LS-DYNA®
KEYWORD USER'S MANUAL

VOLUME I

May 2010
Version 971 / Rev 5
(Beta)

LIVERMORE SOFTWARE TECHNOLOGY CORPORATION (LSTC)

Corporate Address
Livermore Software Technology Corporation
P. O. Box 712
Livermore, California 94551-0712

Support Addresses
Livermore Software Technology Corporation
7374 Las Positas Road
Livermore, California 94551
Tel: 925-449-2500 ♦ Fax: 925-449-2507
Email: sales@lstc.com
Website: www.lstc.com

Livermore Software Technology Corporation
1740 West Big Beaver Road
Suite 100
Troy, Michigan 48084
Tel: 248-649-4728 ♦ Fax: 248-649-6328

Disclaimer

ISBN 0-9778540-2-7

Issue Date: 21/01/2002

This file contains the code for implementing the key schedule for AES (Rijndael) for block and key sizes of 16, 24, and 32 bytes.

TABLE OF CONTENTS

TABLE OF CONTENTS

TABLE OF CONTENTS

TABLE OF CONTENTS

TABLE OF CONTENTS

TABLE OF CONTENTS

TABLE OF CONTENTS

TABLE OF CONTENTS

TABLE OF CONTENTS

TABLE OF CONTENTS

LS-DYNA USER'S MANUAL

INTRODUCTION

CHRONOLOGICAL HISTORY

DYNA3D originated at the Lawrence Livermore National Laboratory [Hallquist 1976]. The early applications were primarily for the stress analysis of structures subjected to a variety of impact loading. These applications required what was then significant computer resources, and the need for a much faster version was immediately obvious. Part of the speed problem was related to the inefficient implementation of the element technology which was further aggravated by the fact that supercomputers in 1976 were much slower than today's PC. Furthermore, the primitive sliding interface treatment could only treat logically regular interfaces that are uncommon in most finite element discretizations of complicated three-dimensional geometries; consequently, defining a suitable mesh for handling contact was often very difficult. The first version contained trusses, membranes, and a choice of solid elements. The solid elements ranged from a one-point quadrature eight-noded element with hourglass control to a twenty-noded element with eight integration points. Due to the high cost of the twenty node solid, the zero energy modes related to the reduced 8-point integration, and the high frequency content which drove the time step size down, higher order elements were all but abandoned in later versions of DYNA3D. A two-dimensional version, DYNA2D, was developed concurrently.

A new version of DYNA3D was released in 1979 that was programmed to provide near optimal speed on the CRAY-1 supercomputers, contained an improved sliding interface treatment that permitted triangular segments and was an order of magnitude faster than the previous contact treatment. The 1979 version eliminated structural and higher order solid elements and some of the material models of the first version. This version also included an optional element-wise implementation of the integral difference method developed by Wilkins et al. [1974].

The 1981 version [Hallquist 1981a] evolved from the 1979 version. Nine additional material models were added to allow a much broader range of problems to be modeled including explosive-structure and soil-structure interactions. Body force loads were implemented for angular velocities and base accelerations. A link was also established from the 3D Eulerian code, JOY [Couch, et. al., 1983] for studying the structural response to impacts by penetrating projectiles. An option was provided for storing element data on disk thereby doubling the capacity of DYNA3D.

The 1982 version of DYNA3D [Hallquist 1982] accepted DYNA2D [Hallquist 1980] material input directly. The new organization was such that equations of state and constitutive models of any complexity could be easily added. Complete vectorization of the material models had been nearly achieved with about a 10 percent increase in execution speed over the 1981 version.

In the 1986 version of DYNA3D [Hallquist and Benson 1986], many new features were added, including beams, shells, rigid bodies, single surface contact, interface friction, discrete springs and dampers, optional hourglass treatments, optional exact volume integration, and VAX/ VMS, IBM, UNIX, COS operating systems compatibility, that greatly expanded its range of applications. DYNA3D thus became the first code to have a general single surface contact algorithm.

In the 1987 version of DYNA3D [Hallquist and Benson 1987] metal forming simulations and composite analysis became a reality. This version included shell thickness changes, the Belytschko-Tsay shell element [Belytschko and Tsay, 1981], and dynamic relaxation. Also

included were non-reflecting boundaries, user specified integration rules for shell and beam elements, a layered composite damage model, and single point constraints.

New capabilities added in the 1988 DYNA3D [Hallquist 1988] version included a cost effective resultant beam element, a truss element, a C^0 triangular shell, the BCIZ triangular shell [Bazeley et al. 1965], mixing of element formulations in calculations, composite failure modeling for solids, noniterative plane stress plasticity, contact surfaces with spot welds, tie break sliding surfaces, beam surface contact, finite stonewalls, stonewall reaction forces, energy calculations for all elements, a crushable foam constitutive model, comment cards in the input, and one-dimensional slidelines.

By the end of 1988 it was obvious that a much more concentrated effort would be required in the development of this software if problems in crashworthiness were to be properly solved; therefore, Livermore Software Technology Corporation was founded to continue the development of DYNA3D as a commercial version called LS-DYNA3D which was later shortened to LS-DYNA. The 1989 release introduced many enhanced capabilities including a one-way treatment of slide surfaces with voids and friction; cross-sectional forces for structural elements; an optional user specified minimum time step size for shell elements using elastic and elastoplastic material models; nodal accelerations in the time history database; a compressible Mooney-Rivlin material model; a closed-form update shell plasticity model; a general rubber material model; unique penalty specifications for each slide surface; external work tracking; optional time step criterion for 4-node shell elements; and internal element sorting to allow full vectorization of right-hand-side force assembly.

During the last ten years, considerable progress has been made as may be seen in the chronology of the developments which follows.

Capabilities added in 1989-1990:

- arbitrary node and element numbers,
- fabric model for seat belts and airbags,
- composite glass model,
- vectorized type 3 contact and single surface contact,
- many more I/O options,
- all shell materials available for 8 node thick shell,
- strain rate dependent plasticity for beams,
- fully vectorized iterative plasticity,
- interactive graphics on some computers,
- nodal damping,
- shell thickness taken into account in shell type 3 contact,
- shell thinning accounted for in type 3 and type 4 contact,
- soft stonewalls,
- print suppression option for node and element data,
- massless truss elements, rivets – based on equations of rigid body dynamics,
- massless beam elements, spot welds – based on equations of rigid body dynamics,
- expanded databases with more history variables and integration points,
- force limited resultant beam,
- rotational spring and dampers, local coordinate systems for discrete elements,
- resultant plasticity for C^0 triangular element,
- energy dissipation calculations for stonewalls,
- hourglass energy calculations for solid and shell elements,
- viscous and Coulomb friction with arbitrary variation over surface,
- distributed loads on beam elements,
- Cowper and Symonds strain rate model,
- segmented stonewalls,
- stonewall Coulomb friction,
- stonewall energy dissipation,
- airbags (1990),

- nodal rigid bodies,
- automatic sorting of triangular shells into C^0 groups,
- mass scaling for quasi static analyses,
- user defined subroutines,
- warpage checks on shell elements,
- thickness consideration in all contact types,
- automatic orientation of contact segments,
- sliding interface energy dissipation calculations,
- nodal force and energy database for applied boundary conditions,
- defined stonewall velocity with input energy calculations,

Capabilities added in 1991-1992:

- rigid/deformable material switching,
- rigid bodies impacting rigid walls,
- strain-rate effects in metallic honeycomb model 26,
- shells and beams interfaces included for subsequent component analyses,
- external work computed for prescribed displacement/velocity/accelerations,
- linear constraint equations,
- MPGS database,
- MOVIE database,
- Slideline interface file,
- automated contact input for all input types,
- automatic single surface contact without element orientation,
- constraint technique for contact,
- cut planes for resultant forces,
- crushable cellular foams,
- urethane foam model with hysteresis,
- subcycling,
- friction in the contact entities,
- strains computed and written for the 8 node thick shells,
- “good” 4 node tetrahedron solid element with nodal rotations,
- 8 node solid element with nodal rotations,
- 2x2 integration for the membrane element,
- Belytschko-Schwer integrated beam,
- thin-walled Belytschko-Schwer integrated beam,
- improved TAURUS database control,
- null material for beams to display springs and seatbelts in TAURUS,
- parallel implementation on Crays and SGI computers,
- coupling to rigid body codes,
- seat belt capability.

Capabilities added in 1993-1994:

- Arbitrary Lagrangian Eulerian brick elements,
- Belytschko-Wong-Chiang quadrilateral shell element,
- Warping stiffness in the Belytschko-Tsay shell element,
- Fast Hughes-Liu shell element,
- Fully integrated thick shell element,
- Discrete 3D beam element,
- Generalized dampers,
- Cable modeling,
- Airbag reference geometry,
- Multiple jet model,
- Generalized joint stiffnesses,
- Enhanced rigid body to rigid body contact,
- Orthotropic rigid walls,

- Time zero mass scaling,
- Coupling with USA (Underwater Shock Analysis),
- Layered spot welds with failure based on resultants or plastic strain,
- Fillet welds with failure,
- Butt welds with failure,
- Automatic eroding contact,
- Edge-to-edge contact,
- Automatic mesh generation with contact entities,
- Drawbead modeling,
- Shells constrained inside brick elements,
- NIKE3D coupling for springback,
- Barlat's anisotropic plasticity,
- Superplastic forming option,
- Rigid body stoppers,
- Keyword input,
- Adaptivity,
- First MPP (Massively Parallel) version with limited capabilities.
- Built in least squares fit for rubber model constitutive constants,
- Large hysteresis in hyperelastic foam,
- Bilhku/Dubois foam model,
- Generalized rubber model,

Capabilities added in 1995:

- Belytschko - Leviathan Shell
- Automatic switching between rigid and deformable bodies.
- Accuracy on SMP machines to give identical answers on one, two or more processors.
- Local coordinate systems for cross-section output can be specified.
- Null material for shell elements.
- Global body force loads now may be applied to a subset of materials.
- User defined loading subroutine.
- Improved interactive graphics.
- New initial velocity options for specifying rotational velocities.
- Geometry changes after dynamic relaxation can be considered for initial velocities..
- Velocities may also be specified by using material or part ID's.
- Improved speed of brick element hourglass force and energy calculations.
- Pressure outflow boundary conditions have been added for the ALE options.
- More user control for hourglass control constants for shell elements.
- Full vectorization in constitutive models for foam, models 57 and 63.
- Damage mechanics plasticity model, material 81,
- General linear viscoelasticity with 6 term prony series.
- Least squares fit for viscoelastic material constants.
- Table definitions for strain rate effects in material type 24.
- Improved treatment of free flying nodes after element failure.
- Automatic projection of nodes in CONTACT_TIED to eliminate gaps in the surface.
- More user control over contact defaults.
- Improved interpenetration warnings printed in automatic contact.
- Flag for using actual shell thickness in single surface contact logic rather than the default.
- Definition by exempted part ID's.
- Airbag to Airbag venting/segmented airbags are now supported.
- Airbag reference geometry speed improvements by using the reference geometry for the time step size calculation.
- Isotropic airbag material may now be directly for cost efficiency.
- Airbag fabric material damping is specified as the ratio of critical damping.

- Ability to attach jets to the structure so the airbag, jets, and structure to move together.
- PVM 5.1 Madymo coupling is available.
- Meshes are generated within LS-DYNA3D for all standard contact entities.
- Joint damping for translational motion.
- Angular displacements, rates of displacements, damping forces, etc. in JNTFORC file.
- Link between LS-NIKE3D to LS-DYNA3D via *INITIAL_STRESS keywords.
- Trim curves for metal forming springback.
- Sparse equation solver for springback.
- Improved mesh generation for IGES and VDA provides a mesh that can directly be used to model tooling in metal stamping analyses.

Capabilities added in 1996-1997 in Version 940:

- Part/Material ID's may be specified with 8 digits.
- Rigid body motion can be prescribed in a local system fixed to the rigid body.
- Nonlinear least squares fit available for the Ogden rubber model.
- Lease squares fit to the relaxation curves for the viscoelasticity in rubber.
- Fu-Chang rate sensitive foam.
- 6 term Prony series expansion for rate effects in model 57-now 73
- Viscoelastic material model 76 implemented for shell elements.
- Mechanical threshold stress (MTS) plasticity model for rate effects.
- Thermoelastic-plastic material model for Hughes-Liu beam element.
- Ramberg-Osgood soil model
- Invariant local coordinate systems for shell elements are optional.
- Second order accurate stress updates.
- Four noded, linear, tetrahedron element.
- Co-rotational solid element for foam that can invert without stability problems.
- Improved speed in rigid body to rigid body contacts.
- Improved searching for the a_3, a_5 and a10 contact types.
- Invariant results on shared memory parallel machines with the *a_n* contact types.
- Thickness offsets in type 8 and 9 tie break contact algorithms.
- Bucket sort frequency can be controlled by a load curve for airbag applications.
- In automatic contact each part ID in the definition may have unique:
  -Static coefficient of friction
  -Dynamic coefficient of friction
  -Exponential decay coefficient
  -Viscous friction coefficient
  -Optional contact thickness
  -Optional thickness scale factor
  -Local penalty scale factor
- Automatic beam-to-beam, shell edge-to-beam, shell edge-to-shell edge and single surface contact algorithm.
- Release criteria may be a multiple of the shell thickness in types a_3, a_5, a10, 13, and 26 contact.
- Force transducers to obtain reaction forces in automatic contact definitions. Defined manually via segments, or automatically via part ID's.
- Searching depth can be defined as a function of time.
- Bucket sort frequency can be defined as a function of time.
- Interior contact for solid (foam) elements to prevent "negative volumes."
- Locking joint
- Temperature dependent heat capacity added to Wang-Nefske inflator models.
- Wang Hybrid inflator model [Wang, 1996] with jetting options and bag-to-bag venting.
- Aspiration included in Wang's hybrid model [Nusholtz, Wang, Wylie, 1996].

- Extended Wang's hybrid inflator with a quadratic temperature variation for heat capacities [Nusholtz, 1996].
- Fabric porosity added as part of the airbag constitutive model .
- Blockage of vent holes and fabric in contact with structure or itself considered in venting with leakage of gas.
- Option to delay airbag liner with using the reference geometry until the reference area is reached.
- Birth time for the reference geometry.
- Multi-material Euler/ALE fluids,
  -2nd order accurate formulations.
  -Automatic coupling to shell, brick, or beam elements
  -Coupling using LS-DYNA contact options.
  -Element with fluid + void and void material
  -Element with multi-materials and pressure equilibrium
- Nodal inertia tensors.
- 2D plane stress, plane strain, rigid, and axisymmetric elements
- 2D plane strain shell element
- 2D axisymmetric shell element.
- Full contact support in 2D, tied, sliding only, penalty and constraint techniques.
- Most material types supported for 2D elements.
- Interactive remeshing and graphics options available for 2D.
- Subsystem definitions for energy and momentum output.
- Boundary element method for incompressible fluid dynamics and fluid-structure interaction problems.

Capabilities added during 1997-1998 in Version 950:

- Adaptive refinement can be based on tooling curvature with FORMING contact.
- The display of drawbeads is now possible since the drawbead data is output into the D3PLOT database.
- An adaptive box option, *DEFINE_BOX_ADAPTIVE, allows control over the refinement level and location of elements to be adapted.
- A root identification file, ADAPT.RID, gives the parent element ID for adapted elements.
- Draw bead box option,*DEFINE_BOX_DRAWBEAD, simplifies drawbead input.
- The new control option, CONTROL_IMPLICIT, activates an implicit solution scheme.
- 2D Arbitrary-Lagrangian-Eulerian elements are available.
- 2D automatic contact is defined by listing part ID's.
- 2D r-adaptivity for plane strain and axisymmetric forging simulations is available.
- 2D automatic non-interactive rezoning as in LS-DYNA2D.
- 2D plane strain and axisymmetric element with 2x2 selective-reduced integration are implemented.
- Implicit 2D solid and plane strain elements are available.
- Implicit 2D contact is available.
- The new keyword, *DELETE_CONTACT_2DAUTO, allows the deletion of 2D automatic contact definitions.
- The keyword, *LOAD_BEAM is added for pressure boundary conditions on 2D elements.
- A viscoplastic strain rate option is available for materials:
  *MAT_PLASTIC_KINEMATIC
  *MAT_JOHNSON_COOK
  *MAT_POWER_LAW_PLASTICITY
  *MAT_STRAIN_RATE_DEPENDENT_PLASTICITY
  *MAT_PIECEWISE_LINEAR_PLASTICITY
  *MAT_RATE_SENSITIVE_POWERLAW_PLASTICITY

*MAT_ZERILLI-ARMSTRONG
*MAT_PLASTICITY_WITH_DAMAGE
*MAT_PLASTICITY_COMPRESSION_TENSION

- Material model, *MAT_PLASTICITY_WITH_DAMAGE, has a piecewise linear damage curve given by a load curve ID.
- The Arruda-Boyce hyper-viscoelastic rubber model is available, see *MAT_ ARRUDA_BOYCE.
- Transverse-anisotropic-viscoelastic material for heart tissue, see *MAT_HEART_ TISSUE.
- Lung hyper-viscoelastic material, see *MAT_LUNG_TISSUE.
- Compression/tension plasticity model, see *MAT_PLASTICITY_COMPRESSION_ TENSION.
- The Lund strain rate model, *MAT_STEINBERG_LUND, is added to Steinberg-Guinan plasticity model.
- Rate sensitive foam model, *MAT_FU_CHANG_FOAM, has been extended to include engineering strain rates, etc.
- Model, *MAT_MODIFIED_PIECEWISE_LINEAR_PLASTICITY, is added for modeling the failure of aluminum.
- Material model, *MAT_SPECIAL_ ORTHOTROPIC, added for television shadow mask problems.
- Erosion strain is implemented for material type, *MAT_BAMMAN_DAMAGE.
- The equation of state, *EOS_JWLB, is available for modeling the expansion of explosive gases.
- The reference geometry option is extended for foam and rubber materials and can be used for stress initialization, see *INITIAL_FOAM_REFERENCE_GEOMETRY.
- A vehicle positioning option is available for setting the initial orientation and velocities, see *INITIAL_VEHICLE_KINEMATICS.
- A boundary element method is available for incompressible fluid dynamics problems.
- The thermal materials work with instantaneous coefficients of thermal expansion:
  *MAT_ELASTIC_PLASTIC_THERMAL
  *MAT_ORTHOTROPIC_THERMAL
  *MAT_TEMPERATURE_DEPENDENT_ORTHOTROPIC
  *MAT_ELASTIC_WITH_VISCOSITY.
- Airbag interaction flow rate versus pressure differences.
- Contact segment search option, [bricks first optional]
- A through thickness Gauss integration rule with 1-10 points is available for shell elements. Previously, 5 were available.
- Shell element formulations can be changed in a full deck restart.
- The tied interface which is based on constraint equations, TIED_SURFACE_TO_ SURFACE, can now fail if _FAILURE, is appended.
- A general failure criteria for solid elements is independent of the material type, see *MAT_ADD_EROSION
- Load curve control can be based on thinning and a flow limit diagram, see *DEFINE_ CURVE_FEEDBACK.
- An option to filter the spotweld resultant forces prior to checking for failure has been added the the option, *CONSTRAINED_SPOTWELD, by appending, _FILTERED_ FORCE, to the keyword.
- Bulk viscosity is available for shell types 1, 2, 10, and 16.
- When defining the local coordinate system for the rigid body inertia tensor a local coordinate system ID can be used. This simplifies dummy positioning.
- Prescribing displacements, velocities, and accelerations is now possible for rigid body nodes.
- One way flow is optional for segmented airbag interactions.
- Pressure time history input for airbag type, LINEAR_FLUID, can be used.

- An option is available to independently scale system damping by part ID in each of the global directions.
- An option is available to independently scale global system damping in each of the global directions.
- Added option to constrain global DOF along lines parallel with the global axes. The keyword is *CONSTRAINED_GLOBAL. This option is useful for adaptive remeshing.
- Beam end code releases are available, see *ELEMENT_BEAM.
- An initial force can be directly defined for the cable material, *MAT_CABLE_ DISCRETE_BEAM. The specification of slack is not required if this option is used.
- Airbag pop pressure can be activated by accelerometers.
- Termination may now be controlled by contact, via *TERMINATION_CONTACT.
- Modified shell elements types 8, 10 and the warping stiffness option in the Belytschko-Tsay shell to ensure orthogonality with rigid body motions in the event that the shell is badly warped. This is optional in the Belytschko-Tsay shell and the type 10 shell.
- A one point quadrature brick element with an exact hourglass stiffness matrix has been implemented for implicit and explicit calculations.
- Automatic file length determination for D3PLOT binary database is now implemented. This insures that at least a single state is contained in each D3PLOT file and eliminates the problem with the states being split between files.
- The dump files, which can be very large, can be placed in another directory by specifying *d=/home/user /test/d3dump* on the execution line.
- A print flag controls the output of data into the MATSUM and RBDOUT files by part ID's. The option, PRINT, has been added as an option to the *PART keyword.
- Flag has been added to delete material data from the D3THDT file. See *DATABASE_ EXTENT_BINARY and column 25 of the 19th control card in the structured input.
- After dynamic relaxation completes, a file is written giving the displaced state which can be used for stress initialization in later runs.

Capabilities added during 1998-2000 in Version 960. Most new capabilities work on both the MPP and SMP versions; however, the capabilities that are implemented for the SMP version only, which were not considered critical for this release, are flagged below. These SMP unique capabilities are being extended for MPP calculations and will be available in the near future. The implicit capabilities for MPP require the development of a scalable eigenvalue solver, which is under development for a later release of LS-DYNA.

- Incompressible flow solver is available. Structural coupling is not yet implemented.
- Adaptive mesh coarsening can be done before the implicit springback calculation in metal forming applications.
- Two-dimensional adaptivity can be activated in both implicit and explicit calculations. (SMP version only)
- An internally generated smooth load curve for metal forming tool motion can be activated with the keyword: *DEFINE_CURVE_SMOOTH.
- Torsional forces can be carried through the deformable spot welds by using the contact type: *CONTACT_SPOTWELD_WITH_TORSION (SMP version only with a high priority for the MPP version if this option proves to be stable.)
- Tie break automatic contact is now available via the *CONTACT_AUTOMATIC_..._ TIEBREAK options. This option can be used for glued panels. (SMP only)
- *CONTACT_RIGID_SURFACE option is now available for modeling road surfaces (SMP version only).
- Fixed rigid walls PLANAR and PLANAR_FINITE are represented in the binary output file by a single shell element.
- Interference fits can be modeled with the INTERFERENCE option in contact.

- A layered shell theory is implemented for several constitutive models including the composite models to more accurately represent the shear stiffness of laminated shells.
- Damage mechanics is available to smooth the post-failure reduction of the resultant forces in the constitutive model *MAT_SPOTWELD_DAMAGE.
- Finite elastic strain isotropic plasticity model is available for solid elements. *MAT_FINITE_ELASTIC_STRAIN_PLASTICITY.
- A shape memory alloy material is available: *MAT_SHAPE_MEMORY.
- Reference geometry for material, *MAT_MODIFIED_HONEYCOMB, can be set at arbitrary relative volumes or when the time step size reaches a limiting value. This option is now available for all element types including the fully integrated solid element.
- Non orthogonal material axes are available in the airbag fabric model. See *MAT_FABRIC.
- Other new constitutive models include for the beam elements:
  *MAT_MODIFIED_FORCE_LIMITED
  *MAT_SEISMIC_BEAM
  *MAT_CONCRETE_BEAM
  for shell and solid elements:
  *MAT_ELASTIC_VISCOPLASTIC_THERMAL
  for the shell elements:
  *MAT_GURSON
  *MAT_GEPLASTIC_SRATE2000
  *MAT_ELASTIC_VISCOPLASTIC_THERMAL
  *MAT_COMPOSITE_LAYUP
  *MAT_COMPOSITE_LAYUP
  *MAT_COMPOSITE_DIRECT
  for the solid elements:
  *MAT_JOHNSON_HOLMQUIST_CERAMICS
  *MAT_JOHNSON_HOLMQUIST_CONCRETE
  *MAT_INV_HYPERBOLIC_SIN
  *MAT_UNIFIED_CREEP
  *MAT_SOIL_BRICK
  *MAT_DRUCKER_PRAGER
  *MAT_RC_SHEAR_WALL
  and for all element options a very fast and efficient version of the Johnson-Cook plasticity model is available:
  *MAT_SIMPLIFIED_JOHNSON_COOK
- A fully integrated version of the type 16 shell element is available for the resultant constitutive models.
- A nonlocal failure theory is implemented for predicting failure in metallic materials. The keyword *MAT_NONLOCAL activates this option for a subset of elastoplastic constitutive models.
- A discrete Kirchhoff triangular shell element (DKT) for explicit analysis with three in plane integration points is flagged as a type 17 shell element. This element has much better bending behavior than the C0 triangular element.
- A discrete Kirchhoff linear triangular and quadrilateral shell element is available as a type 18 shell. This shell is for extracting normal modes and static analysis.
- A C0 linear 4-node quadrilateral shell element is implemented as element type 20 with drilling stiffness for normal modes and static analysis.
- An assumed strain linear brick element is available for normal modes and statics.
- The fully integrated thick shell element has been extended for use in implicit calculations.
- A fully integrated thick shell element based on an assumed strain formulation is now available. This element uses a full 3D constitutive model which includes the normal stress component and, therefore, does not use the plane stress assumption.

- The 4-node constant strain tetrahedron element has been extended for use in implicit calculations.
- Relative damping between parts is available, see *DAMPING_RELATIVE (SMP only).
- Preload forces are can be input for the discrete beam elements.
- Objective stress updates are implemented for the fully integrated brick shell element.
- Acceleration time histories can be prescribed for rigid bodies.
- Prescribed motion for nodal rigid bodies is now possible.
- Generalized set definitions, i.e., SET_SHELL_GENERAL etc. provide much flexibility in the set definitions.
- The command "sw4." will write a state into the dynamic relaxation file, D3DRLF, during the dynamic relaxation phase if the D3DRLF file is requested in the input.
- Added mass by PART ID is written into the MATSUM file when mass scaling is used to maintain the time step size, (SMP version only).
- Upon termination due to a large mass increase during a mass scaled calculation a print summary of 20 nodes with the maximum added mass is printed.
- Eigenvalue analysis of models containing rigid bodies is now available using BCSLIB-EXT solvers from Boeing. (SMP version only).
- Second order stress updates can be activated by part ID instead of globally on the *CONTROL_ACCURACY input.
- Interface frictional energy is optionally computed for heat generation and is output into the interface force file (SMP version only).
- The interface force binary database now includes the distance from the contact surface for the FORMING contact options. This distance is given after the nodes are detected as possible contact candidates. (SMP version only).
- Type 14 acoustic brick element is implemented. This element is a fully integrated version of type 8, the acoustic element (SMP version only).
- A flooded surface option for acoustic applications is available (SMP version only).
- Attachment nodes can be defined for rigid bodies. This option is useful for NVH applications.
- CONSTRAINED_POINTS tie any two points together. These points must lie on a shell elements.
- Soft constraint is available for edge to edge contact in type 26 contact.
- CONSTAINED_INTERPOLATION option for beam to solid interfaces and for spreading the mass and loads. (SMP version only).
- A database option has been added that allows the output of added mass for shell elements instead of the time step size.
- A new contact option allows the inclusion of all internal shell edges in contact type *CONTACT_GENERAL, type 26. This option is activated by adding _INTERIOR after the GENERAL keyword.
- A new option allows the use deviatoric strain rates rather than total rates in material model 24 for the Cowper-Symonds rate model.
- The CADFEM option for ASCII databases is now the default. Their option includes more significant figures in the output files.
- When using deformable spot welds, the added mass for spot welds is now printed for the case where global mass scaling is activated. This output is in the log file, D3HSP file, and the MESSAG file.
- Initial penetration warnings for edge-to-edge contact are now written into the MESSAG file and the D3HSP file.
- Each compilation of LS-DYNA is given a unique version number.
- Finite length discrete beams with various local axes options are now available for material types 66, 67, 68, 93, and 95. In this implementation the absolute value of SCOOR must be set to 2 or 3 in the *SECTION_BEAM input.
- New discrete element constitutive models are available:
  *MAT_ELASTIC_SPRING_DISCRETE_BEAM

*MAT_INELASTIC_SPRING_DISCRETE_BEAM
*MAT_ELASTIC_6DOF_SPRING_DISCRETE_BEAM
*MAT_INELASTIC_6DOF_SPRING_DISCRETE_BEAM

The latter two can be used as finite length beams with local coordinate systems.

- Moving SPC's are optional in that the constraints are applied in a local system that rotates with the 3 defining nodes.
- A moving local coordinate system, CID, can be used to determine orientation of discrete beam elements.
- Modal superposition analysis can be performed after an eigenvalue analysis. Stress recovery is based on type 18 shell and brick (SMP only).
- Rayleigh damping input factor is now input as a fraction of critical damping, i.e. 0.10. The old method required the frequency of interest and could be highly unstable for large input values.
- Airbag option "SIMPLE_PRESSURE_VOLUME" allows for the constant CN to be replaced by a load curve for initialization. Also, another load curve can be defined which allows CN to vary as a function of time during dynamic relaxation. After dynamic relaxation CN can be used as a fixed constant or load curve.
- Hybrid inflator model utilizing CHEMKIN and NIST databases is now available. Up to ten gases can be mixed.
- Option to track initial penetrations has been added in the automatic SMP contact types rather than moving the nodes back to the surface. This option has been available in the MPP contact for some time. This input can be defined on the fourth card of the *CONTROL_CONTACT input and on each contact definition on the third optional card in the *CONTACT definitions.
- If the average acceleration flag is active, the average acceleration for rigid body nodes is now written into the D3THDT and NODOUT files. In previous versions of LS-DYNA, the accelerations on rigid nodes were not averaged.
- A capability to initialize the thickness and plastic strain in the crash model is available through the option *INCLUDE_STAMPED_PART, which takes the results from the LS-DYNA stamping simulation and maps the thickness and strain distribution onto the same part with a different mesh pattern.
- A capability to include finite element data from other models is available through the option, *INCLUDE_TRANSFORM. This option will take the model defined in an INCLUDE file: offset all ID's; translate, rotate, and scale the coordinates; and transform the constitutive constants to another set of units.

Many new capabilities were added during 2001-2002 to create version 970 of LS-DYNA. Some of the new features, which are also listed below, were also added to later releases of version 960. Most new explicit capabilities work for both the MPP and SMP versions; however, the implicit capabilities for MPP require the development of a scalable eigenvalue solver and a parallel implementation of the constraint equations into the global matrices. This work is underway. A later release of version 970 is planned in 2003 that will be scalable for implicit solutions.
Below is list of new capabilities and features:

- MPP decomposition can be controlled using *CONTROL_MPP_ DECOMPOSITION commands in the input deck.
- The MPP arbitrary Lagrangian-Eulerian fluid capability now works for airbag deployment in both SMP and MPP calculations.
- Euler-to-Euler coupling is now available through the keyword *CONSTRAINED_EULER_ TO_EULER.
- Up to ten ALE multi-material groups may now be defined. The previous limit was three groups.
- Volume fractions can be automatically assigned during initialization of multi-material cells. See the GEOMETRY option of *INITIAL_VOLUME_ FRACTION.
- A new ALE smoothing option is available to accurately predict shock fronts.

- DATABASE_FSI activates output of fluid-structure interaction data to ASCII file DBFSI.
- Point sources for airbag inflators are available. The origin and mass flow vector of these inflators are permitted to vary with time.
- A majority of the material models for solid materials are available for calculations using the SPH (Smooth Particle Hydrodynamics) option.
- The Element Free Galerkin method (EFG or meshfree) is available for two-dimensional and three-dimensional solids. This new capability is not yet implemented for MPP applications.
- A binary option for the ASCII files is now available. This option applies to all ASCII files and results in one binary file that contains all the information normally spread between a large number of separate ASCII files.
- Material models can now be defined by numbers rather than long names in the keyword input. For example the keyword *MAT_PIECEWISE_LINEAR_ PLASTICITY can be replaced by the keyword: *MAT_024.
- An embedded NASTRAN reader for direct reading of NASTRAN input files is available. This option allows a typical input file for NASTRAN to be read directly and used without additional input. See the *INCLUDE_NASTRAN keyword.
- Names in the keyword input can represent numbers if the *PARAMETER option is used to relate the names and the corresponding numbers.
- Model documentation for the major ASCII output files is now optional. This option allows descriptors to be included within the ASCII files that document the contents of the file.
- ID's have been added to the following keywords:
  *BOUNDARY_PRESCRIBED_MOTION
  *BOUNDARY_PRESCRIBED_SPC
  *CONSTRAINED_GENERALIZED_WELD
  *CONSTRAINED_JOINT
  *CONSTRAINED_NODE_SET
  *CONSTRAINED_RIVET
  *CONSTRAINED_SPOTWELD
  *DATABASE_CROSS_SECTION
  *ELEMENT_MASS
- The *DATABASE_ADAMS keyword is available to output a modal neutral file d3mnf. This will is available upon customer request since it requires linking to an ADAMS library file.
- Penetration warnings for the contact option, ignore initial penetration, î are added as an option. Previously, no penetration warnings were written when this contact option was activated.
- Penetration warnings for nodes in-plane with shell mid-surface are printed for the AUTOMATIC contact options. Previously, these nodes were ignored since it was assumed that they belonged to a tied interface where an offset was not used; consequently, they should not be treated in contact.
- For the arbitrary spot weld option, the spot welded nodes and their contact segments are optionally written into the D3HSP file. See *CONTROL_CONTACT.
- For the arbitrary spot weld option, if a segment cannot be found for the spot welded node, an option now exists to error terminate. See *CONTROL_CONTACT.
- Spot weld resultant forces are written into the SWFORC file for solid elements used as spot welds.
- Solid materials have now been added to the failed element report.
- A new option for terminating a calculation is available, *TERMINATION_CURVE.
- A 10-noded tetrahedron solid element is available with either a 4 or 5 point integration rule. This element can also be used for implicit solutions.
- A new 4 node linear shell element is available that is based on Wilson's plate element combined with a Pian-Sumihara membrane element. This is shell type 21.

- A shear panel element has been added for linear applications. This is shell type 22. This element can also be used for implicit solutions.
- A null beam element for visualization is available. The keyword to define this null beam is *ELEMENT_PLOTEL. This element is necessary for compatibility with NASTRAN.
- A scalar node can be defined for spring-mass systems. The keyword to define this node is *NODE_SCALAR. This node can have from 1 to 6 scalar degrees-of-freedom.
- A thermal shell has been added for through-thickness heat conduction. Internally, 8 additional nodes are created, four above and four below the mid-surface of the shell element. A quadratic temperature field is modeled through the shell thickness. Internally, the thermal shell is a 12 node solid element.
- A beam OFFSET option is available for the *ELEMENT_BEAM definition to permit the beam to be offset from its defining nodal points. This has the advantage that all beam formulations can now be used as shell stiffeners.
- A beam ORIENTATION option for orienting the beams by a vector instead of the third node is available in the *ELEMENT_BEAM definition for NASTRAN compatibility.
- Non-structural mass has been added to beam elements for modeling trim mass and for NASTRAN compatibility.
- An optional checking of shell elements to avoid abnormal terminations is available. See *CONTROL_SHELL. If this option is active, every shell is checked each time step to see if the distortion is so large that the element will invert, which will result in an abnormal termination. If a bad shell is detected, either the shell will be deleted or the calculation will terminate. The latter is controlled by the input.
- An offset option is added to the inertia definition. See *ELEMENT_ INERTIA_OFFSET keyword. This allows the inertia tensor to be offset from the nodal point.
- Plastic strain and thickness initialization is added to the draw bead contact option. See *CONTACT_DRAWBEAD_INITIALIZE.
- Tied contact with offsets based on both constraint equations and beam elements for solid elements and shell elements that have 3 and 6 degrees-of-freedom per node, respectively. See BEAM_OFFSET and CONSTRAINED_ OFFSET contact options. These options will not cause problems for rigid body motions.
- The segment-based (SOFT=2) contact is implemented for MPP calculations. This enables airbags to be easily deployed on the MPP version.
- Improvements are made to segment-based contact for edge-to-edge and sliding conditions, and for contact conditions involving warped segments.
- An improved interior contact has been implemented to handle large shear deformations in the solid elements. A special interior contact algorithm is available for tetrahedron elements.
- Coupling with MADYMO 6.0 uses an extended coupling that allows users to link most MADYMO geometric entities with LS-DYNA FEM simulations. In this coupling MADYMO contact algorithms are used to calculate interface forces between the two models.
- Release flags for degrees-of-freedom for nodal points within nodal rigid bodies are available. This makes the nodal rigid body option nearly compatible with the RBE2 option in NASTRAN.
- Fast updates of rigid bodies for metalforming applications can now be accomplished by ignoring the rotational degrees-of-freedom in the rigid bodies that are typically inactive during sheet metal stamping simulations. See the keyword: *CONTROL_RIGID.
- Center of mass constraints can be imposed on nodal rigid bodies with the SPC option in either a local or a global coordinate system.

- Joint failure based on resultant forces and moments can now be used to simulate the failure of joints.
- CONSTRAINED_JOINT_STIFFNESS now has a TRANSLATIONAL option for the translational and cylindrical joints.
- Joint friction has been added using table look-up so that the frictional moment can now be a function of the resultant translational force.
- The nodal constraint options *CONSTRAINED_INTERPOLATION and *CONSTRAINED_LINEAR now have a local option to allow these constraints to be applied in a local coordinate system.
- Mesh coarsening can now be applied to automotive crash models at the beginning of an analysis to reduce computation times. See the new keyword: *CONTROL_COARSEN.
- Force versus time seatbelt pretensioner option has been added.
- Both static and dynamic coefficients of friction are available for seat belt slip rings. Previously, only one friction constant could be defined.
- *MAT_SPOTWELD now includes a new failure model with rate effects as well as additional failure options.
- Constitutive models added for the discrete beam elements:
  *MAT_1DOF_GENERALIZED_SPRING
  *MAT_GENERAL_NONLINEAR_6dof_DISCRETE_BEAM
  *MAT_GENERAL_NONLINEAR_1dof_DISCRETE_BEAM
  *MAT_GENERAL_SPRING_DISCRETE_BEAM
  *MAT_GENERAL_JOINT_DISCRETE_BEAM
  *MAT_SEISMIC_ISOLATOR
  for shell and solid elements:
  *MAT_plasticity_with_damage_ortho
  *MAT_simplified_johnson_cook_orthotropic_damage
  *MAT_HILL_3R
  *MAT_GURSON_RCDC
  for the solid elements:
  *MAT_SPOTWELD
  *MAT_HILL_FOAM
  *MAT_WOOD
  *MAT_VISCOELASTIC_HILL_FOAM
  *MAT_LOW_DENSITY_SYNTHETIC_FOAM
  *MAT_RATE_SENSITIVE_POLYMER
  *MAT_QUASILINEAR VISCOELASTIC
  *MAT_TRANSVERSELY_ANISOTROPIC_CRUSHABLE_FOAM
  *MAT_VACUUM
  *MAT_MODIFIED_CRUSHABLE_FOAM
  *MAT_PITZER_CRUSHABLE FOAM
  *MAT_JOINTED_ROCK
  *MAT_SIMPLIFIED_RUBBER
  *MAT_FHWA_SOIL
  *MAT_SCHWER_MURRAY_CAP_MODEL
- Failure time added to MAT_EROSION for solid elements.
- Damping in the material models *MAT_LOW_DENSITY_FOAM and *MAT_LOW_ DENSITY_VISCOUS_FOAM can now be a tabulated function of the smallest stretch ratio.
- The material model *MAT_PLASTICITY_WITH_DAMAGE allows the table definitions for strain rate.
- Improvements in the option *INCLUDE_STAMPED_PART now allow all history data to be mapped to the crash part from the stamped part. Also, symmetry planes can be used to allow the use of a single stamping to initialize symmetric parts.

- Extensive improvements in trimming result in much better elements after the trimming is completed. Also, trimming can be defined in either a local or global coordinate system. This is a new option in *DEFINE_CURVE_TRIM.
- An option to move parts close before solving the contact problem is available, see *CONTACT_AUTO_MOVE.
- An option to add or remove discrete beams during a calculation is available with the new keyword: *PART_SENSOR.
- Multiple jetting is now available for the Hybrid and Chemkin airbag inflator models.
- Nearly all constraint types are now handled for implicit solutions.
- Calculation of constraint and attachment modes can be easily done by using the option: *CONTROL_IMPLICIT_MODES.
- Penalty option, see *CONTROL_CONTACT, now applies to all *RIGIDWALL options and is always used when solving implicit problems.
- Solid elements types 3 and 4, the 4 and 8 node elements with 6 degrees-of-freedom per node are available for implicit solutions.
- The warping stiffness option for the Belytschko-Tsay shell is implemented for implicit solutions. The Belytschko-Wong-Chang shell element is now available for implicit applications. The full projection method is implemented due to it accuracy over the drill projection.
- Rigid to deformable switching is implemented for implicit solutions.
- Automatic switching can be used to switch between implicit and explicit calculations. See the keyword: *CONTROL_IMPLICIT_GENERAL.
- Implicit dynamics rigid bodies are now implemented. See the keyword *CONTROL_IMPLICIT_DYNAMIC.
- Eigenvalue solutions can be intermittently calculated during a transient analysis.
- A linear buckling option is implemented. See the new control input: *CONTROL_ IMPLICIT_BUCKLE
- Implicit initialization can be used instead of dynamic relaxation. See the keyword *CONTROL_DYNAMIC_RELAXATION where the parameter, IDFLG, is set to 5.
- Superelements, i.e., *ELEMENT_DIRECT_MATRIX_INPUT, are now available for implicit applications.
- There is an extension of the option, *BOUNDARY_CYCLIC, to symmetry planes in the global Cartesian system. Also, automatic sorting of nodes on symmetry planes is now done by LS-DYNA.
- Modeling of wheel-rail contact for railway applications is now available, see *RAIL_TRACK and *RAIL_TRAIN.
- A new, reduced CPU, element formulation is available for vibration studies when elements are aligned with the global coordinate system. See *SECTION_SOLID and *SECTION_SHELL formulation 98.
- An option to provide approximately constant damping over a range of frequencies is implemented, see *DAMPING_FREQUENCY_RANGE.

Many new capabilities were added during 2003-2005 to create version 971 of LS-DYNA. Initially, the intent was to quickly release version 971 after 970 with the implicit capabilities fully functional for distributed memory processing using MPI. Unfortunately, the effort required for parallel implicit was grossly underestimated, and, as a result, the release has been delayed. Because of the delay, version 971 has turned into a major release. Some of the new features, listed below, were also added to later releases of version 970. The new explicit capabilities are implemented in the MPP version and except for one case, in the SMP version as well.
Below is list of new capabilities and features:

- A simplified method for using the ALE capability with airbags is now available with the keyword *AIRBAG_ALE.
- Case control using the *CASE keyword, which provides a way of running multiple load cases sequentially within a single run

- New option to forming contact: *CONTACT_FORMING_ONE_WAY_SURFACE_TO_SURFACE_SMOOTH, which use fitted surface in contact calculation.
- Butt weld definition by using the *CONSTRAINED_BUTT_WELD option which makes the definition of butt welds simple relative to the option: *CONSTRAINED_GENERALIZED_WELD_BUTT.
- H-adaptive fusion is now possible as an option with the control input, *CONTROL_ADAPTIVE.
- Added a parameter on, *CONTROL_ADAPTIVE, to specify the number of elements generated around a 90 degree radius. A new option to better calculate the curvature was also implemented.
- Added a new keyword: *CONTROL_ADAPTIVE_CURVE, to refine the element along trimming curves
- Birth and death times for implicit dynamics on the keyword *CONTROL_IMPLICIT_DYNAMICS.
- Added an option to scale the spot weld failure resultants to account for the location of the weld on the segment surface, see *CONTROL_SPOTWELD_BEAM.
- Added an option which automatically replaces a single beam spot weld by an assembly of solid elements using the same ID as the beam that was replaced, see *CONTROL_SPOTWELD_BEAM.
- Boundary constraint in a local coordinate system using *CONSTRAINED_LOCAL keyword.
- A cubic spline interpolation element is now available, *CONSTRAINED_SPLINE.
- Static implicit analyses in of a structure with rigid body modes is possible using the option, *CONTROL_IMPLICIT_INERTIA_RELIEF.
- Shell element thickness updates can now be limited to part ID's within a specified set ID, see the *CONTROL_SHELL keyword. The thickness update for shells can now be optionally limited to the plastic part of the strain tensor for better stability in crash analysis.
- Solid element stresses in spot welds are optionally output in the local system using the SWLOCL parameter on the *CONTROL_SOLID keyword.
- SPOTHIN option on the *CONTROL_CONTACT keyword cards locally thins the spot welded parts to prevent premature breakage of the weld by the contact treatments.
- New function: *CONTROL_FORMING_PROJECT, which can initial move the penetrating slave nodes to the master surface
- New function *CONTROL_FORMING_TEMPLATE, which allows user to easily set up input deck. Its function includes auto-position, define travel curve, termination time, and most of the forming parameters for most of the typical forming process.
- New function *CONTROL_FORMING_USER, *CONTROL_FORMING_POSITION, and *CONTROL_FORMING_TRAVEL, when used together, can allow the user to define atypical forming process.
- Added new contact type *CONTACT_GUIDED_CABLE.
- Circular cut planes are available for *DATABASE_CROSS_SECTION definitions.
- New binary database FSIFOR for fluid structure coupling.
- Added *DATABASE_BINARY_D3PROP for writing the material and property data to the first D3PLOT file or to a new database D3PROP.
- DATABASE_EXTENT_BINARY has new flags to output peak pressure, surface energy density, nodal mass increase from mass scaling, thermal fluxes, and temperatures at the outer surfaces of the thermal shell.
- Eight-character alphanumeric labels can now be used for the parameters SECID, MID, EOSID, HGID, and TMID on the *PART keyword.
- Two NODOUT files are now written: one for high frequency output and a second for low frequency output.
- Nodal mass scaling information can now be optionally written to the D3PLOT file.

- Added option, MASS_PROPERTIES, to include the mass and inertial properties in the GLSTAT and SSSTAT files.
- Added option in *CONTROL_CPU to output the cpu and elapsed time into the GLSTAT file.
- Added an option, IERODE, on the *CONTROL_OUTPUT keyword to include eroded energies by part ID into the MATSUM file. Lumped mass kinetic energy is also in the MATSUM file as part ID 0.
- Added an option, TET10, on the *CONTROL_OUTPUT keyword to output ten connectivity nodes into D3PLOT database rather than 4.
- New keyword, *ELEMENT_SOLID_T4TOT10 to convert 4 node tetrahedron elements to 10 node tetrahedron elements.
- New keyword, *ELEMENT_MASS_PART defines the total additional non-structural mass to be distributed by an area weighted distribution to all nodes of a given part ID.
- New keyword option, SET, for *INTIAL_STRESS_SHELL_SET allows a set of shells to be initialized with the state of stress.
- New option allows the number of cpu's to be specified on the *KEYWORD input.
- Tubular drawbead box option for defining the elements that are included in the drawbead contact, see *DEFINE_BOX_DRAWBEAD.
- New function: *DEFINE_CURVE_DRAWBEAD, allow user to conveniently define drawbead by using curves (in x, y format or iges format)
- New function: *DEFINE_DRAWBEAD_BEAM, which allows user to conveniently define drawbead by using beam part ID, and specify the drawbead force.
- Analytic function can be used in place of load curves with the option *DEFINE_ CURVE_FUNCTION.
- Friction can now be defined between part pair using the *DEFINE_FRICTION input.
- New keyword: *DEFINE_CURVE_TRIM_3D, to allow trimming happens based on blank element normal, rather than use pre-defined direction
- A new trimming algorithm was added: *DEFINE_CURVE_TRIM_NEW, which allow seed node to be input and is much faster then the original algorithm.
- A new keyword, *DEFINE_HEX_SPOTWELD_ASSEMBLY, is available to define a cluster of solid elements that comprise a single spot weld.
- The definition of a vector, see *DEFINE_VECTOR, can be done by defining coordinates in a local coordinate system.
- The definition of a failure criteria between part pairs is possible with a table defined using the keyword, *DEFINE_SPOTWELD_FAILURE_RESULTANTS.
- A new keyword, *DEFINE_CONNECTION_PROPERTIES is available for defining failure properties of spot welds.
- Added *DEFINE_SET_ADAPTIVE to allow the adaptive level and element size to be specified by part ID or element set ID.
- Static rupture stresses for beam type spot welds can be defined in the keyword input, *DEFINE_SPOTWELD_RUPTURE_STRESS.
- Section properties can be define in the *ELEMENT_BEAM definitions for resultant beam elements using the SECTION option.
- Physical offsets of the shell reference surface can be specified on the shell element cards, see the OFFSET option on *ELEMENT_SHELL.
- File names can be located in remote directories and accessed through the *INCLUDE _PART keyword.
- New features to *INCLUDE_STAMPED_PART: two different mirror options, user-defined searching radius.
- *INTIAL_STRESS_SECTION allows for stress initialization across a cross-section, which consists of solid elements.
- An option, IVATN, is available for setting the velocities of slaved nodes and parts for keyword, *INITIAL_VELOCITY_GENERATION.
- Twenty-two built-in cross-section are now available in the definition of beam integration rules, see *INTEGRATION_BEAM.

- The possibility of changing material types is now available for shells using the user defined integration rule, see *INTEGRATION_SHELL.
- The interface springback file created by using the keyword, *INTERFACE_ SPRINGBACK is now optionally written as a binary file.
- An optional input line for *KEYWORD allows the definition of a prefix for all file names created during a simulation. This allows multiple jobs to be executed in the same directory.
- Body force loads can now be applied in a local coordinate system for *LOAD_ BODY.
- A pressure loading feature allows moving pressures to be applied to a surface to simulate spraying a surface with stream of fluid through a nozzle. See keyword *LOAD_MOVING_PRESSURE.
- Thermal expansion can be added to any material by the keyword, *MAT_ADD_ THERMAL_EXPANSION.
- Curves can now be used instead of eight digitized data points in the material model *MAT_ELASTIC_WITH_VISCOSITY_CURVE
- New options for spot weld failure in *MAT_SPOTWELD, which apply to beam and solid elements.
- Failure criteria based on plastic strain to failure is added to material *MAT_ ANISOTROPIC_VISCOPLASTIC.
- Strain rate failure criterion is added to material *MAT_MODIFIED_PIECEWISE_ LINEAR_PLASTICITY.
- Strain rate scaling of the yield stress can now be done differently in tension and compression in material with separate pressure cut-offs in tension and compression in material model *MAT_PLASTICITY_TENSION_COMPRESSION.
- The RCDC model is now available to predict failure in material *MAT_ PLASTICITY_WITH_DAMAGE.
- Two additional yield surfaces have been added to material *MAT_MODIFIED_ HONEYCOMB to provide more accurate predictions of the behavior of honeycomb barrier models.
- Unique coordinate systems can be assigned to the two nodal points of material *MAT_1DOF_GENERALIZED_SPRING.
- Poisson's ratio effects are available in foam defined by load curves in the material *MAT_SIMPLIFIED_RUBBER/FOAM
- Failure effects are available in the rubber/foam material defined by load curves in the *MAT_SIMPLIFIED_RUBBER/FOAM_WITH_FAILURE.
- The material option *MAT_ADD_EROSION now allows the maximum pressure at failure and the minimum principal strain at failure to be specified.
- Strains rather than displacements can now be used with the material model for discrete beams, *MAT_GENERAL_NONLINEAR_6DOF_DISCRETE_BEAM.
- New option for MAT_TRANSVERSELY_ANISOTROPIC_ELASTIC_PLASTIC_ (ECHANGE), which allow two ways to change the Young's modulus during forming simulation.
- New Material model: *MAT_HILL_3R: includes the shear term in the yield surface calculation by using Hill's 1948 an-isotropic material model.
- New Material model: *MAT_KINEMATIC_HARDENING_TRANSVERSELY_ ANISOTROPIC: which integrates Mat #37 with Yoshida's two-surface kinematic hardening model.
- Improved formulation for the fabric material, *MAT_FABRIC for formulations 2, 3, and 4. The improved formulations are types 12, 13, and 14.
- Constitutive models added for truss elements:
  *MAT_MUSCLE
  For beam elements
  *MAT_MOMENT-CURVATURE

For shell elements
*MAT_RESULTANT_ANISOTROPIC
*MAT_RATE_ SENSITIVE_COMPOSITE_FABRIC.
*MAT_SAMP-1
*MAT_SHAPE_MEMORY is now implemented for shells.
for shell and solid elements:
*MAT_ BARLAT_YLD2000 for anisotropic aluminum alloys.
*MAT_SIMPLIFIED_RUBBER_WITH_DAMAGE
*MAT_VISCOELASTIC_THERMAL
*MAT_THERMO_ELASTO_VISCOPLASTIC_CREEP
for the solid elements:
*MAT_ARUP_ADHESIVE
*MAT_BRAIN_ LINEAR_VISCOELASTIC.
*MAT_CSCM for modeling concrete.
*MAT_PLASTICITY_COMPRESSION_TENSION_EOS for modeling ice.
*MAT_COHESIVE_ELASTIC
*MAT_COHESIVE_TH
*MAT_COHESIVE_GENERAL
*MAT_EOS_GASKET
*MAT_SIMPLIFIED_JOHNSON_COOK is now implemented for solids.
*MAT_PLASTICITY_WITH_DAMAGE is now implemented for solids.
*MAT_SPOTWELD_DAIMLERCHRYSLER

- User defined equations-of-state are now available.
- There is now an interface with the MOLDFLOW code.
- Damping defined in *DAMPING_PART_STIFFNESS now works for the Belytschko –Schwer beam element.
- The option *NODE_TRANSFORMATION allows a node set to be transformed based on a transformation defined in *DEFINE_TRANSFORMATION.
- Parameters can be defined in FORTRAN like expressions using *PARAMETER_ EXPRESSION.
- A part can be moved in a local coordinate system in *PART_MOVE.
- A simplified method for defining composite layups is available with *PART_ COMPOSITE
- The rigid body inertia can be changed in restart via *CHANGE_RIGID_BODY_ INERTIA.
- A part set can now be defined by combining other part sets in *SET_PART_ADD.
- Termination of the calculation is now possible if a specified number of shell elements are deleted in a give part ID. See *TERMINATION_DELETED_SHELLS.
- Added hourglass control type 7 for solid elements for use when modeling hyperelastic materials.
- Shell formulations 4, 11, 16, and 17 can now model rubber materials.
- Added a new seatbelt pretensioner type 7 in which the pretensioner and retractor forces are calculated independently and added.
- A new composite tetrahedron element made up from 12 tetrahedron is now available as solid element type 17.
- Shell thickness offsets for *SECTION_SHELL now works for most shell elements, not just the Hughes-Liu shell.
- The Hughes-Liu beam has been extended to include warpage for open cross-sections.
- A resultant beam formulation with warpage is available as beam type 12.
- Two nonlinear shell elements are available with 8 degrees-of-freedom per node to include thickness stretch.
- Tetrahedron type 13, which uses nodal pressures, is now implemented for implicit applications.
- Cohesive solid elements are now available for treating failure.
- Seatbelt shell elements are available for use with the all seatbelt capabilities.

- Superelements can now share degrees-of-freedom and are implemented for implicit applications under MPI.
- A user defined element interface is available for solid and shell elements.
- Thermal shells are available for treating heat flow through shell elements.
- EFG shell formulations 41 and 42 are implemented for explicit analysis.
- EFGPACK is implemented in addition to BCSLIB-EXT solver on the keyword *CONTROL_EFG.
- EFG MPP version is available for explicit analysis.
- EFG fast transformation method is implemented in the EFG solid formulation.
- EFG Semi-Lagrangian kernel and Eulerian kernel options are added for the foam materials.
- EFG 3D adaptivity is implemented for the metal materials.
- EFG E.O.S. and *MAT_ELASTIC_FLUID materials are included in the 4-noded background element formulation.
- Airbag simulations by using ALE method can be switched to control volume method by *ALE_CV_SWITCH.
- *MAT_ALE_VISCOUS now supports Non-Newtonian viscosity by power law or load curve.
- *DATABASE_BINARY_FSIFOR outputs fluid-structure interaction data to binary file.
- *DATABASE_FSI_SENSOR outputs ALE element pressure to ASCII file dbsor.
- *MAT_GAS_MIXTURE supports nonlinear heat capacities.
- *INITIAL_VOLUME_FRACTION_GEOMETRY uses an enhanced algorithm to handle both concave and convex geometries and substantially reduce run time.
- A new keyword *DELETE_FSI allows the deletion of coupling definitions.
- Convection heat transfer activates by *LOAD_ALE_CONVECTION in ALE FSI analysis.
- *ALE_FSI_SWITCH_MMG is implemented to switch between ALE multi-material groups to treat immersed FSI problems.
- Type 9 option is added in *ALE_REFERENCE_SYSTEM_GROUP to deal complex ALE mesh motions including translation, rotation, expansion and contraction, etc.
- New options in *CONSTRAINED_LAGRANGE_IN_SOLID
  - Shell thickness option for coupling type 4.
  - Bulk modulus based coupling stiffness.
  - Shell erosion treatment.
  - Enable/disable interface force file.
- New coupling method for fluid flowing through porous media are implemented as type 11 (shell) and type 12 (solid) in *CONSTRAINED_LAGRANGE_IN_SOLID.
- *ALE_MODIFIED_STRAIN allows multiple strain fields in certain ALE elements to solve sticking behavior in FSI. (MPP underdevelopment)
- *ALE_FSI_PROJECTION is added as a new constraint coupling method to solve small pressure variation problem. (MPP underdevelopment)
- *BOUNDARY_PRESCRIBED_ORIENTATION_RIGID is added as a means to prescribe as a function of time the general orientation of a rigid body using a variety of methods. This feature is available in release R3 and higher of Version 971.
- *BOUNDARY_PRESCRIBED_ACCELEROMETER_RIGID is added as a means to prescribe the motion of a rigid body based un experimental data gathered from accelerometers affixed to the rigid body. . This feature is available in release R3 and higher of Version 971.

MATERIAL MODELS

Some of the material models presently implemented are:

- elastic,
- orthotropic elastic,
- kinematic/isotropic plasticity [Krieg and Key 1976],
- thermoelastoplastic [Hallquist 1979],
- soil and crushable/non-crushable foam [Key 1974],
- linear viscoelastic [Key 1974],
- Blatz-Ko rubber [Key 1974],
- high explosive burn,
- hydrodynamic without deviatoric stresses,
- elastoplastic hydrodynamic,
- temperature dependent elastoplastic [Steinberg and Guinan 1978],
- isotropic elastoplastic,
- isotropic elastoplastic with failure,
- soil and crushable foam with failure,
- Johnson/Cook plasticity model [Johnson and Cook 1983],
- pseudo TENSOR geological model [Sackett 1987],
- elastoplastic with fracture,
- power law isotropic plasticity,
- strain rate dependent plasticity,
- rigid,
- thermal orthotropic,
- composite damage model [Chang and Chang 1987a 1987b],
- thermal orthotropic with 12 curves,
- piecewise linear isotropic plasticity,
- inviscid, two invariant geologic cap [Sandler and Rubin 1979, Simo et al, 1988a 1988b],
- orthotropic crushable model,
- Mooney-Rivlin rubber,
- resultant plasticity,
- force limited resultant formulation,
- closed form update shell plasticity,
- Frazer-Nash rubber model,
- laminated glass model,
- fabric,
- unified creep plasticity,
- temperature and rate dependent plasticity,
- elastic with viscosity,
- anisotropic plasticity,
- user defined,
- crushable cellular foams [Neilsen, Morgan, and Krieg 1987],
- urethane foam model with hysteresis,

and some more foam and rubber models, as well as many materials models for springs and dampers. The hydrodynamic material models determine only the deviatoric stresses. Pressure is determined by one of ten equations of state including:

- linear polynomial [Woodruff 1973],
- JWL high explosive [Dobratz 1981],
- Sack "Tuesday" high explosive [Woodruff 1973],
- Gruneisen [Woodruff 1973],
- ratio of polynomials [Woodruff 1973],
- linear polynomial with energy deposition,
- ignition and growth of reaction in HE [Lee and Tarver 1980, Cochran and Chan 1979],
- tabulated compaction,
- tabulated,
- TENSOR pore collapse [Burton et al. 1982].

The ignition and growth EOS was adapted from KOVEC [Woodruff 1973]; the other subroutines, programmed by the authors, are based in part on the cited references and are nearly 100 percent vectorized. The forms of the first five equations of state are also given in the KOVEC user's manual and are retained in this manual. The high explosive programmed burn model is described by Giroux [Simo et al. 1988].

The orthotropic elastic and the rubber material subroutines use Green-St. Venant strains to compute second Piola-Kirchhoff stresses, which transform to Cauchy stresses. The Jaumann stress rate formulation is used with all other materials with the exception of one plasticity model which uses the Green-Naghdi rate.

SPATIAL DISCRETIZATION

The elements shown in Figure I.1 are presently available. Currently springs, dampers, beams, membranes, shells, bricks, thick shells and seatbelt elements are included.

The first shell element in DYNA3D was that of Hughes and Liu [Hughes and Liu 1981a, 1981b, 1981c], implemented as described in [Hallquist et al. 1985, Hallquist and Benson 1986]. This element [designated as HL] was selected from among a substantial body of shell element literature because the element formulation has several desirable qualities:

- It is incrementally objective (rigid body rotations do not generate strains), allowing for the treatment of finite strains that occur in many practical applications;

- It is compatible with brick elements, because the element is based on a degenerated brick element formulation. This compatibility allows many of the efficient and effective techniques developed for the DYNA3D brick elements to be used with this shell element;

- It includes finite transverse shear strains;

- A through-the-thickness thinning option (see [Hughes and Carnoy 1981]) is also available.

All shells in our current LS-DYNA code must satisfy these desirable traits to at least some extent to be useful in metalforming and crash simulations.

The major disadvantage of the HL element turned out to be cost related and, for this reason, within a year of its implementation we looked at the Belytschko-Tsay [BT] shell [Belytschko and Tsay 1981, 1983, 1984] as a more cost effective, but possibly less accurate alternative. In the BT shell the geometry of the shell is assumed to be perfectly flat, the local coordinate system originates at the first node of the connectivity, and the co-rotational stress update does not use the costly Jaumann stress rotation. With these and other simplifications, a very cost effective shell was derived which today has become perhaps the most widely used shell elements in both metalforming and crash applications. Results generated by the BT shell usually compare favorably with those of the more costly HL shell. Triangular shell elements are implemented, based on work by Belytschko and co-workers [Belytschko and Marchertas 1974, Bazeley et al. 1965, Belytschko et al. 1984], and are frequently used since collapsed quadrilateral shell elements tend to lock and give very bad results. LS-DYNA automatically treats collapsed quadrilateral shell elements as C^0 triangular elements

Since the Belytschko-Tsay element is based on a perfectly flat geometry, warpage is not considered. Although this generally poses no major difficulties and provides for an efficient element, incorrect results in the twisted beam problem and similar situations are obtained where the nodal points of the elements used in the discretization are not coplanar. The Hughes-Liu shell element considers non-planar geometries and gives good results on the twisted beam. The effect of neglecting warpage in a typical application cannot be predicted beforehand and may lead to less than accurate results, but the latter is only speculation and is difficult to verify in practice. Obviously, it would be better to use shells that consider warpage if the added costs are reasonable and if this unknown effect is eliminated. Another shell published by Belytschko, Wong, and Chiang [Belytschko, Wong, and Chiang 1989, 1992] proposes inexpensive modifications to include the warping stiffness in the Belytschko-Tsay shell. An improved transverse shear treatment also allows the element to pass the Kirchhoff patch test. This element is now available in LS-DYNA. Also, two fully integrated shell elements, based on the Hughes and Liu formulation, are available in LS-DYNA, but are rather expensive. A much faster fully integrated element which is essentially a fully integrated version of the Belytschko, Wong, and Chiang element, type 16, is a more recent addition and is recommended if fully integrated elements are needed due to its cost effectiveness.

Three-dimensional plane stress constitutive subroutines are implemented for the shell elements which iteratively update the stress tensor such that the stress component normal to the shell midsurface is zero. An iterative update is necessary to accurately determine the normal strain component which is necessary to predict thinning. One constitutive evaluation is made for each integration point through the shell thickness.

Zero energy modes in the shell and solid elements are controlled by either an hourglass viscosity or stiffness. Eight node thick shell elements are implemented and have been found to perform well in many applications. All elements are nearly 100% vectorized. All element classes can be included as parts of a rigid body. The rigid body formulation is documented in [Benson and Hallquist 1986]. Rigid body point nodes, as well as concentrated masses, springs and dashpots can be added to this rigid body.

Membrane elements can be either defined directly as shell elements with a membrane formulation option or as shell elements with only one point for through thickness integration. The latter choice includes transverse shear stiffness and may be inappropriate. For airbag material a special fully integrated three and four node membrane element is available.

Two different beam types are available: a stress resultant beam and a beam with cross section integration at one point along the axis. The cross section integration allows for a more general definition of arbitrarily shaped cross sections taking into account material nonlinearities.

Spring and damper elements can be translational or rotational. Many behavior options can be defined, e.g., arbitrary nonlinear behavior including locking and separation.

Solid elements in LS-DYNA may be defined using from 4 to 8 nodes. The standard elements are based on linear shape functions and use one point integration and hourglass control. A selective-reduced integrated (called fully integrated) 8 node solid element is available for situations when the hourglass control fails. Also, two additional solid elements, a 4 noded tetrahedron and an 8 noded hexahedron, with nodal rotational degrees of freedom, are implemented based on the idea of Allman [1984] to replace the nodal midside translational degrees of freedom of the elements with quadratic shape functions by corresponding nodal rotations at the corner nodes. The latter elements, which do not need hourglass control, require many numerical operations compared to the hourglass controlled elements and should be used at places where the hourglass elements fail. However, it is well known that the elements using more than one point integration are more sensitive to large distortions than one point integrated elements.

The thick shell element is a shell element with only nodal translations for the eight nodes. The assumptions of shell theory are included in a non-standard fashion. It also uses hourglass control or selective-reduced integration. This element can be used in place of any four node shell element. It is favorably used for shell-brick transitions, as no additional constraint conditions are necessary. However, care has to be taken to know in which direction the shell assumptions are made; therefore, the numbering of the element is important.

Seatbelt elements can be separately defined to model seatbelt actions combined with dummy models. Separate definitions of seatbelts, which are one-dimensional elements, with accelerometers, sensors, pretensioners, retractors, and sliprings are possible. The actions of the various seatbelt definitions can also be arbitrarily combined.

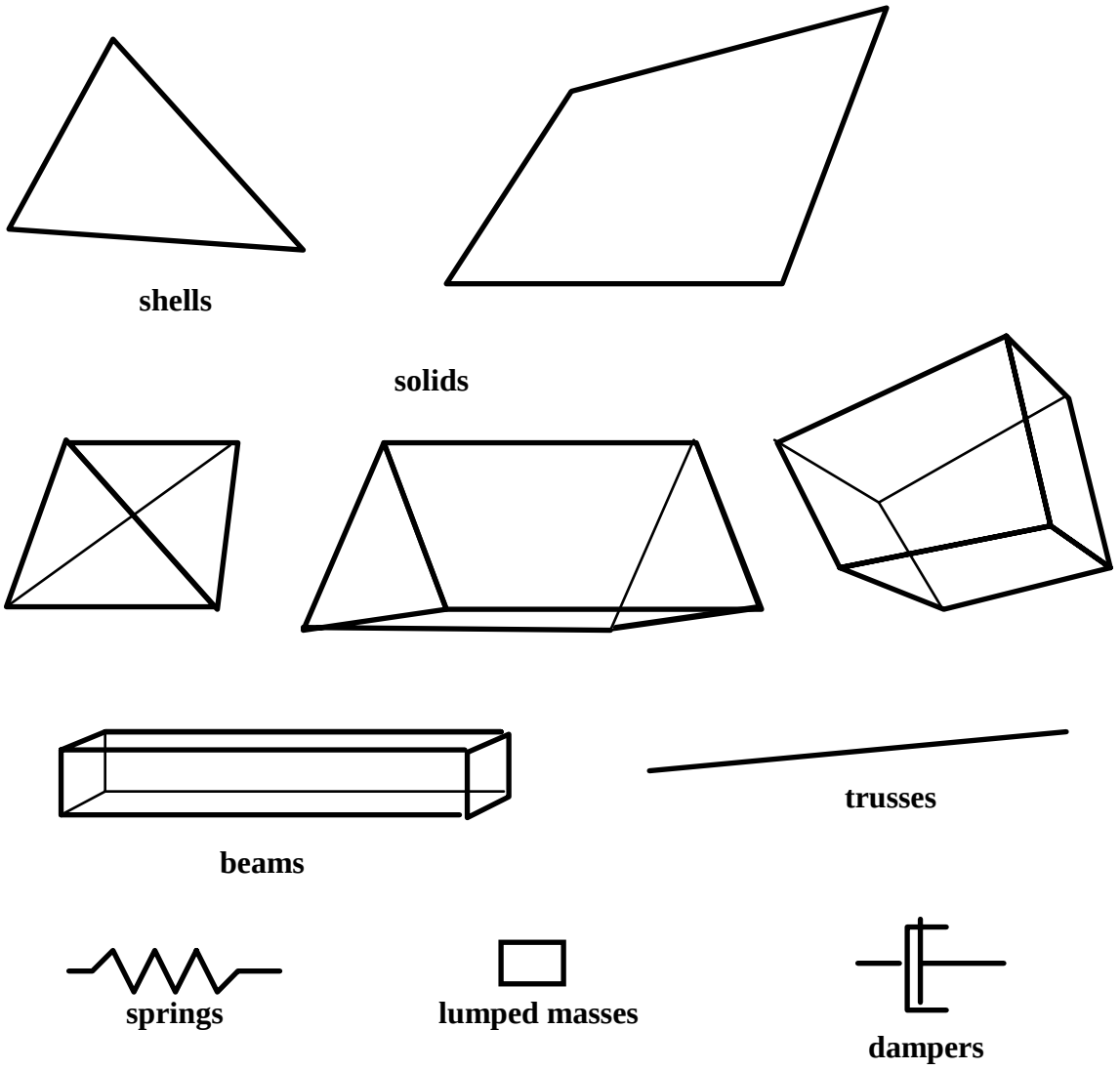


Figure I.1. Elements in LS-DYNA.

CONTACT-IMPACT INTERFACES

The three-dimensional contact-impact algorithm was originally an extension of the NIKE2D [Hallquist 1979] two-dimensional algorithm. As currently implemented, one surface of the interface is identified as a master surface and the other as a slave. Each surface is defined by a set of three or four node quadrilateral segments, called master and slave segments, on which the nodes of the slave and master surfaces, respectively, must slide. In general, an input for the contact-impact algorithm requires that a list of master and slave segments be defined. For the single surface algorithm only the slave surface is defined and each node in the surface is checked each time step to ensure that it does not penetrate through the surface. Internal logic [Hallquist 1977, Hallquist et al. 1985] identifies a master segment for each slave node and a slave segment for each master node and updates this information every time step as the slave and master nodes slide along their respective surfaces. It must be noted that for general automatic definitions only parts/materials or three-dimensional boxes have to be given. Then the possible contacting outer surfaces are identified by the internal logic in LS-DYNA. More than 20 types of interfaces can presently be defined including:

sliding only for fluid/structure or gas/structure interfaces,
tied,
sliding, impact, friction,
single surface contact,
discrete nodes impacting surface,
discrete nodes tied to surface,
shell edge tied to shell surface,
nodes spot welded to surface,
tiebreak interface,
one way treatment of sliding, impact, friction,
box/material limited automatic contact for shells,
automatic contact for shells (no additional input required),
automatic single surface with beams and arbitrary orientations,
surface to surface eroding contact,
node to surface eroding contact,
single surface eroding contact,
surface to surface symmetric constraint method [Taylor and Flanagan 1989],
node to surface constraint method [Taylor and Flanagan 1989],
rigid body to rigid body contact with arbitrary force/deflection curve,
rigid nodes to rigid body contact with arbitrary force/deflection curve,
edge-to-edge,
draw beads.

Interface friction can be used with most interface types. The tied and sliding only interface options are similar to the two-dimensional algorithm used in LS-DYNA2D [Hallquist 1976, 1978, 1980]. Unlike the general option, the tied treatments are not symmetric; therefore, the surface which is more coarsely zoned should be chosen as the master surface. When using the one-way slide surface with rigid materials, the rigid material should be chosen as the master surface.

For geometric contact entities, contact has to be separately defined. It must be noted that for the contact of a rigid body with a flexible body, either the sliding interface definitions as explained above or the geometric contact entity contact can be used. Currently, the geometric

contact entity definition is recommended for metalforming problems due to high accuracy and computational efficiency.

INTERFACE DEFINITIONS FOR COMPONENT ANALYSIS

Interface definitions for component analyses are used to define surfaces, nodal lines, or nodal points (*INTERFACE_COMPONENTS) for which the displacement and velocity time histories are saved at some user specified frequency (*CONTROL_OUTPUT). This data may then used to drive interfaces (*INTERFACE_LINKING) in subsequent analyses. This capability is especially useful for studying the detailed response of a small member in a large structure. For the first analysis, the member of interest need only be discretized sufficiently that the displacements and velocities on its boundaries are reasonably accurate. After the first analysis is completed, the member can be finely discretized and interfaces defined to correspond with the first analysis. Finally, the second analysis is performed to obtain highly detailed information in the local region of interest.

When starting the analysis, specify a name for the interface segment file using the Z = parameter on the LS-DYNA command line. When starting the second analysis, the name of the interface segment file (created in the first run) should be specified using the L = parameter on the LS-DYNA command line.

Following the above procedure, multiple levels of sub-modeling are easily accommodated. The interface file may contain a multitude of interface definitions so that a single run of a full model can provide enough interface data for many component analyses. The interface feature represents a powerful extension of LS-DYNA's analysis capability.

CAPACITY

Storage allocation is dynamic. The only limit that exists on the number of boundary condition cards, number of material cards, number of pressure cards, etc., is the capacity of the computer. Typical LS-DYNA calculations may have 10,000 to 500,000 elements. Memory allocation is dynamic and can be controlled during execution.

PRECISION

The explicit time integration algorithms used in LS-DYNA are in general much less sensitive to machine precision than other finite element solution methods. Consequently, double precision is not used. The benefits of this are greatly improved utilization of memory and disk. When problems have been found we have usually been able to overcome them by reorganizing the algorithm or by converting to double precision locally in the subroutine where the problem occurs. A few of the known problems include: **(32-bit computers only!):**

- Round-off errors can cause difficulties with extremely small deflection problems. (Maximum vibration amplitudes are $<10^{-6}$ times nodal coordinates).
  Workaround: Increase the load.
- Buckling problems, which are very sensitive to small imperfections.

However, the users of LS-DYNA have to be aware of potential problems.

A major reorganization of LS-DYNA has led to a version using double precision throughout the full program. As memory and disk space of the computer is less of a problem, we prefer to provide this version for all machines. It also allows LS-DYNA to take advantage of the 64-bit technology offered by many computer manufacturers.

INTRODUCTION

GETTING STARTED

DESCRIPTION OF KEYWORD INPUT

The keyword input provides a flexible and logically organized database that is simple to understand. Similar functions are grouped together under the same keyword. For example, under the keyword *ELEMENT are included solid, beam, shell elements, spring elements, discrete dampers, seat belts, and lumped masses. Many keywords have options that are identified as follows: "*OPTIONS*" and "{*OPTIONS*}". The difference is that "*OPTIONS*" requires that one of the options must be selected to complete the keyword command. The option <BLANK> is included when {} are used to further indicate that these particular options are not necessary to complete the keyword.

LS-DYNA User's Manual is alphabetically organized in logical sections of input data. Each logical section relates to a particular input. There is a control section for resetting LS-DYNA defaults, a material section for defining constitutive constants, an equation-of-state section, an element section where element part identifiers and nodal connectivities are defined, a section for defining parts, and so on. Nearly all model data can be input in block form. For example, consider the following where two nodal points with their respective coordinates and shell elements with their part identity and nodal connectivities are defined:

```
$       DEFINE TWO NODES
$
*NODE
        10101       x       y       z
        10201       x       y       z
$       DEFINE TWO SHELL ELEMENTS
$
*ELEMENT_SHELL
        10201       pid     n1      n2      n3      n4
        10301       pid     n1      n2      n3      n4
```

Alternatively, acceptable input could also be of the form:

```
$       DEFINE ONE NODE
$
*NODE
        10101       x       y       z
$       DEFINE ONE SHELL ELEMENT
$
*ELEMENT_SHELL
        10201       pid     n1      n2      n3      n4
$
$       DEFINE ONE MORE NODE
$
*NODE
        10201       x       y       z
$       DEFINE ONE MORE SHELL ELEMENT
$
*ELEMENT_SHELL
        10301       pid     n1      n2      n3      n4
```

A data block begins with a keyword followed by the data pertaining to the keyword. The next keyword encountered during the reading of the block data defines the end of the block and the beginning of a new block. A keyword must be left justified with the "*" contained in column one. A dollar sign "$" in column one precedes a comment and causes the input line to be ignored. Data blocks are not a requirement for LS-DYNA but they can be used to group nodes and elements for user convenience. Multiple blocks can be defined with each keyword if desired as shown above. It would be possible to put all nodal points definitions under one keyword *NODE, or to define one *NODE keyword prior to each node definition. The entire LS-DYNA input is order independent with the exception of the optional keyword, *END, which defines the end of input stream. Without the *END termination is assumed to occur when an end-of-file is encountered during the reading.

Figure GS.1 attempts to show the general philosophy of the input organization and how various entities relate to each other. In this figure the data included for the keyword, *ELEMENT, is the element identifier, EID, the part identifier, PID, and the nodal points identifiers, the NID's, defining the element connectivity: N1, N2, N3, and N4. The nodal point identifiers are defined in the *NODE section where each NID should be defined just once. A part defined with the *PART keyword has a unique part identifier, PID, a section identifier, SID, a material or constitutive model identifier, MID, an equation of state identifier, EOSID, and the hourglass control identifier, HGID. The *SECTION keyword defines the section identifier, SID, where a section has an element formulation specified, a shear factor, SHRF, a numerical integration rule, NIP, and so on. The constitutive constants are defined in the *MAT section where constitutive data is defined for all element types including solids, beams, shells, thick shells, seat belts, springs, and dampers. Equations of state, which are used only with certain *MAT materials for solid elements, are defined in the *EOS section. Since many elements in LS-DYNA use uniformly reduced numerical integration, zero energy deformation modes may develop. These modes are controlled numerically by either an artificial stiffness or viscosity which resists the formation of these undesirable modes. The hourglass control can optionally be user specified using the input in the *HOURGLASS section.

During the keyword input phase where data is read, only limited checking is performed on the data since the data must first be counted for the array allocations and then reordered. Considerably more checking is done during the second phase where the input data is printed out. Since LS-DYNA has retained the option of reading older non-keyword input files, we print out the data into the output file D3HSP (default name) as in previous versions of LS-DYNA. An attempt is made to complete the input phase before error terminating if errors are encountered in the input. Unfortunately, this is not always possible and the code may terminate with an error message. The user should always check either output file, D3HSP or MESSAG, for the word "Error".

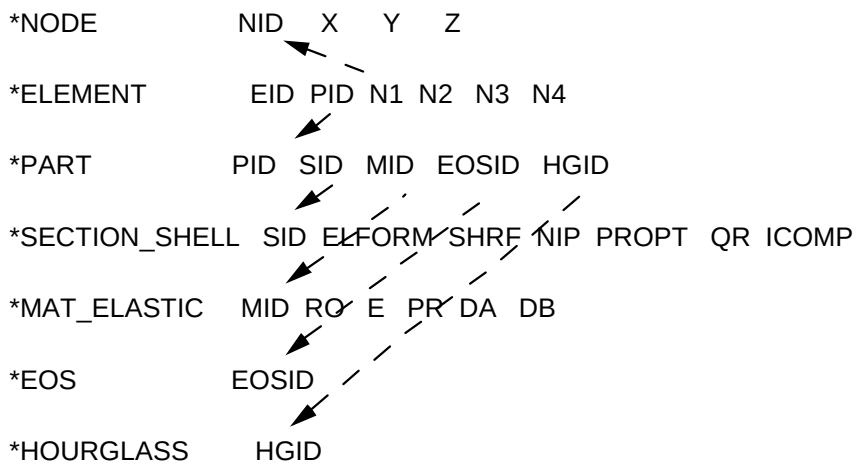


Figure GS.1 Organization of the keyword input.

The input data following each keyword can be input in free format. In the case of free format input the data is separated by commas, i.e.,

*NODE

10101,x ,y ,z

10201,x ,y ,z

*ELEMENT_SHELL

10201,pid,n1,n2,n3,n4

10301,pid,n1,n2,n3,n4

When using commas, the formats **must not** be violated. An I8 integer is limited to a maximum positive value of 99999999, and larger numbers having more than eight characters are unacceptable. The format of the input can change from free to fixed anywhere in the input file. The input is case insensitive and keywords can be given in either upper or lower case. THE ASTERISKS "*" PRECEDING EACH KEYWORD MUST BE IN COLUMN ONE.

To provide a better understanding behind the keyword philosophy and how the options work, a brief review the keywords is given below.

***AIRBAG**
The geometric definition of airbags and the thermodynamic properties for the airbag inflator models can be made in this section. This capability is not necessarily limited to the modeling of automotive airbags, but it can also be used for many other applications such as tires and pneumatic dampers.

***ALE**
This keyword provides a way of defining input data pertaining to the Arbitrary-Lagrangian-Eulerian capability.

***BOUNDARY**
This section applies to various methods of specifying either fixed or prescribed boundary conditions. For compatibility with older versions of LS-DYNA it is still possible to specify some nodal boundary conditions in the *NODE card section.

***CASE**
This keyword option provides a way of running multiple load cases sequentially. Within each case, the input parameters, which include loads, boundary conditions, control cards, contact definitions, initial conditions, etc., can change. If desired, the results from a previous case can be used during initialization. Each case creates unique file names for all output results files by appending "**CID*n***." to the default file name.

***COMPONENT**
This section contains analytical rigid body dummies that can be placed within vehicle and integrated implicitly.

***CONSTRAINED**
This section applies constraints within the structure between structural parts. For example, nodal rigid bodies, rivets, spot welds, linear constraints, tying a shell edge to a shell edge with failure, merging rigid bodies, adding extra nodes to rigid bodies and defining rigid body joints are all options in this section.

***CONTACT**
This section is divided in to three main sections. The *CONTACT section allows the user to define many different contact types. These contact options are primarily for treating contact of deformable to deformable bodies, single surface contact in deformable bodies, deformable body to rigid body contact, and tying deformable structures with an option to release the tie based on plastic strain. The surface definition for contact is made up of segments on the shell or solid element surfaces. The keyword options and the corresponding numbers in previous code versions are:

STRUCTURED INPUT TYPE ID	KEYWORD NAME
1	SLIDING_ONLY
p 1	SLIDING_ONLY_PENALTY
2	TIED_SURFACE_TO_SURFACE
3	SURFACE_TO_SURFACE
a 3	AUTOMATIC_SURFACE_TO_SURFACE
4	SINGLE_SURFACE
5	NODES_TO_SURFACE
a 5	AUTOMATIC_NODES_TO_SURFACE
6	TIED_NODES_TO_SURFACE
7	TIED_SHELL_EDGE_TO_SURFACE
8	TIEBREAK_NODES_TO_SURFACE
9	TIEBREAK_SURFACE_TO_SURFACE
10	ONE_WAY_SURFACE_TO_SURFACE
a 10	AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE
13	AUTOMATIC_SINGLE_SURFACE
a 13	AIRBAG_SINGLE_SURFACE
14	ERODING_SURFACE_TO_SURFACE
15	ERODING_SINGLE_SURFACE
16	ERODING_NODES_TO_SURFACE
17	CONSTRAINT_SURFACE_TO_SURFACE
18	CONSTRAINT_NODES_TO_SURFACE
19	RIGID_BODY_TWO_WAY_TO_RIGID_BODY
20	RIGID_NODES_TO_RIGID_BODY
21	RIGID_BODY_ONE_WAY_TO_RIGID_BODY
22	SINGLE_EDGE
23	DRAWBEAD

The *CONTACT_ENTITY section treats contact between a rigid surface, usually defined as an analytical surface, and a deformable structure. Applications of this type of contact exist in the metal forming area where the punch and die surface geometries can be input as VDA surfaces which are treated as rigid. Another application is treating contact between rigid body occupant dummy hyper-ellipsoids and deformable structures such as airbags and instrument panels. This

option is particularly valuable in coupling with the rigid body occupant modeling codes MADYMO and CAL3D. The *CONTACT_1D is for modeling rebars in concrete structure.

***CONTROL**
Options available in the *CONTROL section allow the resetting of default global parameters such as the hourglass type, the contact penalty scale factor, shell element formulation, numerical damping, and termination time.

***DAMPING**
Defines damping either globally or by part identifier.

***DATABASE**
This keyword with a combination of options can be used for controlling the output of ASCII databases and binary files output by LS-DYNA. With this keyword the frequency of writing the various databases can be determined.

***DEFINE**
This section allows the user to define curves for loading, constitutive behaviors, etc.; boxes to limit the geometric extent of certain inputs; local coordinate systems; vectors; and orientation vectors specific to spring and damper elements. Items defined in this section are referenced by their identifiers throughout the input. For example, a coordinate system identifier is sometimes used on the *BOUNDARY cards, and load curves are used on the *AIRBAG cards.

***DEFORMABLE_TO_RIGID**
This section allows the user to switch parts that are defined as deformable to rigid at the start of the analysis. This capability provides a cost efficient method for simulating events such as rollover events. While the vehicle is rotating the computation cost can be reduced significantly by switching deformable parts that are not expected to deform to rigid parts. Just before the vehicle comes in contact with ground, the analysis can be stopped and restarted with the part switched back to deformable.

***EF**
Exchange factors characterize radiative heat transfer between collections of flat surfaces, the union of which is a closed surface (an enclosure). LS-DYNA can calculate exchange factors and then use them as boundary conditions for thermal runs. The $(i,j)^{th}$ element of an exchange factor matrix, E_{ij}, is the fraction of the Stefan-Boltzman surface energy radiated from surface i that is absorbed by surface j. LS-DYNA employs a Monte Carlo algorithm to calculate these exchange factors. For each surface, LS-DYNA simulates photon emission one photon at a time. For each photon, LS-DYNA generates a random initial position on the emitting surfaces as well as a random initial direction that points into the enclosure. LS-DYNA ray traces each photon until it is absorbed. The path of a simulated photon can be complex involving multiple diffuse and specular reflections as well as multiple diffuse and specular transmissions. The results of this Monte Carlo algorithm are used to assemble a matrix that is related to the exchange factor matrix, for which, the $(i,j)^{th}$ entry contains the number of photons emitted from surface i that are absorbed by surface j. From this matrix LS-DYNA then assembles the exchange factor matrix.

***ELEMENT**
Define identifiers and connectivities for all elements which include shells, beams, solids, thick shells, springs, dampers, seat belts, and concentrated masses in LS-DYNA.

***EOS**
This section reads the equations of state parameters. The equation of state identifier, EOSID, points to the equation of state identifier on the *PART card.

***HOURGLASS**
Defines hourglass and bulk viscosity properties. The identifier, HGID, on the *HOURGLASS card refers to HGID on *PART card.

***INCLUDE**
To make the input file easy to maintain, this keyword allows the input file to be split into subfiles. Each subfile can again be split into sub-subfiles and so on. This option is beneficial when the input data deck is very large.

***INITIAL**
Initial velocity and initial momentum for the structure can be specified in this section. The initial velocity specification can be made by *INITIAL_VELOCITY_NODE card or *INITIAL_ VELOCITY cards. In the case of *INITIAL_VELOCITY_NODE nodal identifiers are used to specify the velocity components for the node. Since all the nodes in the system are initialized to zero, only the nodes with non-zero velocities need to be specified. The *INITIAL_VELOCITY card provides the capability of being able to specify velocities using the set concept or boxes.

***INTEGRATION**
In this section the user defined integration rules for beam and shell elements are specified. IRID refers to integration rule number IRID on *SECTION_BEAM and *SECTION_SHELL cards respectively. Quadrature rules in the *SECTION_SHELL and *SECTION_BEAM cards need to be specified as a negative number. The absolute value of the negative number refers to user defined integration rule number. Positive rule numbers refer to the built in quadrature rules within LS-DYNA.

***INTERFACE**
Interface definitions are used to define surfaces, nodal lines, and nodal points for which the displacement and velocity time histories are saved at some user specified frequency. This data may then used in subsequent analyses as an interface ID in the *INTERFACE_LINKING_DISCRETE_ NODE as master nodes, in *INTERFACE_LINKING_ SEGMENT as master segments and in *INTERFACE_LINKING_EDGE as the master edge for a series of nodes. This capability is especially useful for studying the detailed response of a small member in a large structure. For the first analysis, the member of interest need only be discretized sufficiently that the displacements and velocities on its boundaries are reasonably accurate. After the first analysis is completed, the member can be finely discretized in the region bounded by the interfaces. Finally, the second analysis is performed to obtain highly detailed information in the local region of interest. When beginning the first analysis, specify a name for the interface segment file using the Z=parameter on the LS-DYNA execution line. When starting the second analysis, the name of the interface segment file created in the first run should be specified using the L=parameter on the LS-DYNA command line. Following the above procedure, multiple levels of sub-modeling are easily accommodated. The interface file may contain a multitude of interface definitions so that a single run of a full model can provide enough interface data for many component analyses. The interface feature represents a powerful extension of LS-DYNA's analysis capabilities. A similar capability using *INTERFACE_SSI may be used for soil-structure interaction analysis under earthquake excitation.

***KEYWORD**
Flags LS-DYNA that the input deck is a keyword deck. To have an effect this must be the very first card in the input deck. Alternatively, by typing "keyword" on the execute line, keyword input formats are assumed and the "*KEYWORD" is not required. If a number is specified on this card after the word KEYWORD it defines the memory size to used in words. The memory size can also be set on the command line. NOTE THAT THE MEMORY SPECIFIED ON THE EXECUTION LINE OVERRIDES MEMORY SPECIFIED ON THE *KEYWORD CARD.

***LOAD**
This section provides various methods of loading the structure with concentrated point loads, distributed pressures, body force loads, and a variety of thermal loadings.

***MAT**
This section allows the definition of constitutive constants for all material models available in LS-DYNA including springs, dampers, and seat belts. The material identifier, MID, points to the MID on the *PART card.

***NODE**
Define nodal point identifiers and their coordinates.

***PARAMETER**
This option provides a way of specifying numerical values of parameter names that are referenced throughout the input file. The parameter definitions, if used, should be placed at the beginning of the input file following *KEYWORD. *PARAMETER_EXPRESSION permits general algebraic expressions to be used to set the values.

***PART**
This keyword serves two purposes.
1. Relates part ID to *SECTION, *MATERIAL, *EOS and *HOURGLASS sections.
2. Optionally, in the case of a rigid material, rigid body inertia properties and initial conditions can be specified. Deformable material repositioning data can also be specified in this section if the reposition option is invoked on the *PART card, i.e., *PART_REPOSITION.

***PERTURBATION**
This keyword provides a way of defining deviations from the designed structure such as, buckling imperfections.

***RAIL**
This keyword provides a way of defining a wheel-rail contact algorithm intended for railway applications but can also be used for other purposes. The wheel nodes (defined on *RAIL_TRAIN) represent the contact patch between wheel and rail.

***RIGIDWALL**
Rigid wall definitions have been divided into two separate sections, _PLANAR and _GEOMETRIC. Planar walls can be either stationary or moving in translational motion with mass and initial velocity. The planar wall can be either finite or infinite. Geometric walls can be planar as well as have the geometric shapes such as rectangular prism, cylindrical prism and sphere. By default, these walls are stationary unless the option MOTION is invoked for either prescribed translational velocity or displacement. Unlike the planar walls, the motion of the

geometric wall is governed by a load curve. Multiple geometric walls can be defined to model combinations of geometric shapes available. For example, a wall defined with the _CYLINDER option can be combined with two walls defined with the _SPHERICAL option to model hemispherical surface caps on the two ends of a cylinder. Contact entities are also analytical surfaces but have the significant advantage that the motion can be influenced by the contact to other bodies, or prescribed with six full degrees-of-freedom.

***SECTION**
In this section, the element formulation, integration rule, nodal thicknesses, and cross sectional properties are defined. All section identifiers (SECID's) defined in this section must be unique, i.e., if a number is used as a section ID for a beam element then this number cannot be used again as a section ID for a solid element.

***SENSOR**
This keyword provides a convenient way of activating and deactivating boundary conditions, airbags, discrete elements, joints, contact, rigid walls, single point constraints, and constrained nodes. The sensor capability is new in the second release of version 971 and will evolve in later releases to encompass many more LS-DYNA capabilities and replace some of the existing capabilities such as the airbag sensor logic.

***SET**
A concept of grouping nodes, elements, materials, etc., in sets is employed throughout the LS-DYNA input deck. Sets of data entities can be used for output. So-called slave nodes used in contact definitions, slaves segment sets, master segment sets, pressure segment sets and so on can also be defined. The keyword, *SET, can be defined in two ways:

1. Option _LIST requires a list of entities, eight entities per card, and define as many cards as needed to define all the entities.

2. Option _COLUMN, where applicable, requires an input of one entity per line along with up to four attribute values which are needed to specify, for example, failure criterion input that is needed for *CONTACT_CONSTRAINT_NODES_TO_SURFACE .

***TERMINATION**
This keyword provides an alternative way of stopping the calculation before the termination time is reached. The termination time is specified on the *CONTROL_TERMINATION input and will terminate the calculation whether or not the options available in this section are active.

***TITLE**
In this section a title for the analysis is defined.

***USER_INTERFACE**
This section provides a method to provide user control of some aspects of the contact algorithms including friction coefficients via user defined subroutines.

RESTART
This section of the input is intended to allow the user to restart the simulation by providing a restart file and optionally a restart input defining changes to the model such as deleting contacts, materials, elements, switching materials from rigid to deformable, deformable to rigid, etc.

***RIGID_TO_DEFORMABLE**
This section switches rigid parts back to deformable in a restart to continue the event of a vehicle impacting the ground which may have been modeled with a rigid wall.

***STRESS_INITIALIZATION**
This is an option available for restart runs. In some cases there may be a need for the user to add contacts, elements, etc., which are not available options for standard restart runs. A full input containing the additions is needed if this option is invoked upon restart.

SUMMARY OF COMMONLY USED OPTIONS

The following table gives a list of the commonly used keywords related by topic.

Table GS.1. Keywords for the most commonly used options.

Topic	Component	Keyword
Geometry	Nodes	*NODE
	Elements	*ELEMENT_BEAM
		*ELEMENT_SHELL
		*ELEMENT_SOLID
		*ELEMENT_TSHELL
	Discrete Elements	*ELEMENT_DISCRETE
		*ELEMENT_MASS
		ELEMENT_SEATBELT_*Option
Materials	Part (which is composed of Material and Section, equation of state and hourglass data)	*PART
	Material	*MAT_*Option*
	Sections	*SECTION_BEAM
		*SECTION_SHELL
		*SECTION_SOLID
		*SECTION_TSHELL
	Discrete sections	*SECTION_DISCRETE
		*SECTION_SEATBELT
	Equation of state	*EOS_*Option*
	Hourglass	*CONTROL_HOURGLASS
		*HOURGLASS
Contacts and Rigid walls	Defaults for contacts	*CONTROL_CONTACT
	Definition of contacts	*CONTACT_*Option*
	Definition of rigid walls	*RIGIDWALL_*Option*

Table GS.1. (continued) Keywords for the most commonly used options.

Topic	Component	Keyword
Boundary Conditions & Loadings	Restraints	*NODE *BOUNDARY_SPC_*Option*
	Gravity (body) load	*LOAD_BODY_*Option*
	Point load	*LOAD_NODE_*Option*
	Pressure load	*LOAD_SEGMENT_*Option* *LOAD_SHELL_*Option*
	Thermal load	*LOAD_THERMAL_*Option*
	Load curves	*DEFINE_CURVE
Constraints and spot welds	Constrained nodes	*CONSTRAINED_NODE_SET
	Welds	*CONSTRAINED_GENERALIZED_WELD_ *Option* *CONSTRAINED_SPOT_WELD
	Rivet	*CONSTRAINED_RIVET
Output Control	Defaults	*CONTROL_OUTPUT
	ASCII time history files	*DATABASE_*Option*
	Binary plot, time history and restart files	*DATABASE_BINARY_*Option*
	Items in time history blocks	*DATABASE_HISTORY_*Option*
	Nodes for nodal reaction output	*DATABASE_NODAL_FORCE_GROUP
Termination	Termination time	*CONTROL_TERMINATION
	Termination cycle	*CONTROL_TERMINATION
	CPU termination	*CONTROL_CPU
	Degree of freedom	*TERMINATION_NODE

EXECUTION SYNTAX

The interactive execution line for LS-DYNA is as follows:

LS-DYNA I=**inf** O=**otf** G=**ptf** D=**dpf** F=**thf** U=**xtf** T=**tpf** A=**rrd** M=**sif** J=**jif** S=**iff** Z=**isf1** L=**isf2** B=**rlf** W=**root** E=**efl** X=**scl** C=**cpu** K=**kill** V=**vda** Y=**c3d BEM=bof {KEYWORD} {THERMAL} {COUPLE} {INIT}** MEMORY=**nwds** NCPU= **ncpu** PARA=**para** ENDTIME=**time** NCYCLE=**ncycle** JOBID=**jobid** D3PROP=**d3prop GMINP=gminp GMOUT=gmout**

where

inf = input file (user specified)
otf = high speed printer file (default=D3HSP)
ptf = binary plot file for graphics (default=D3PLOT)
dpf = dump file for restarting (default=D3DUMP). This file is written at the end of every run and during the run as requested in the input. To stop the generation of this file set the file name to NODUMP.

thf = binary plot file for time histories of selected data (default=D3THDT)
xtf = binary plot file for time extra data (default-XTFILE)
tpf = optional temperature file
rrd = running restart dump file (default=RUNRSF)
sif = stress initialization file (user specified)
jif = optional JOY interface file
iff = interface force file (user specified)
isf1 = interface segment save file to be created (user specified)
isf2 = existing interface segment save file to be used (user specified)
rlf = binary plot file for dynamic relaxation (default=D3DRFL)
efl = echo file containing optional input echo with or without node/element data
root = root file name for general print option
scl = scale factor for binary file sizes (default=7)
cpu = cpu limit in seconds, applies to total calculation not just cpu from a restart
kill = if LS-DYNA encounters this file name it will terminate with a restart file (default=D3KIL)
vda = VDA/IGES database for geometrical surfaces
c3d = CAL3D input file
bof = *BOUNDARY_ELEMENT_METHOD_ACOUSTIC output file
nwds = Number of words to be allocated. On engineering workstations a word is usually 32bits. **This number overwrites the memory size specified on the *KEYWORD card at the beginning of the input deck.**
ncpu = Overrides **NCPU** and **CONST** defined in *CONTROL_PARALLEL. A positive value sets CONST=2 and a negative values sets CONST=1. See the *CONTROL_PARALLEL command for an explanation of these parameters. The *KEYWORD command provides an alternative way to set the number of CPUs.
npara = Overrides **PARA** defined in *CONTROL_PARALLEL.
time = Overrides **ENDTIM** defined in *CONTROL_TERMINATION.
ncycle = Overrides **ENDCYC** defined in *CONTROL_TERMINATION.
jobid = Character string which acts as a prefix for all output files. Maximum length is 72 characters. **Do not** include the following characters:) (* / ? \.
d3prop = See *DATABASE_BINARY_D3PROP input parameter IFILE for options.
gminp = Input file for reading recorded motions in *INTERFACE_SSI (default=GMBIN).
gmout = Output file for writing recorded motions in *INTERFACE_SSI_AUX (default=GMBIN).

In order to avoid undesirable or confusing results, each LS-DYNA run should be performed in a separate directory, unless using the command line parameter “jobid” described above. If rerunning a job in the same directory, old files should first be removed or renamed to avoid confusion since the possibility exists that the binary database may contain results from both the old and new run.

By including **KEYWORD** anywhere on the execute line or instead if ***KEYWORD** is the first card in the input file, the keyword formats are expected; otherwise, the older structured input file will be expected.

To run a coupled thermal analysis the command **COUPLE** must be in the execute line. A thermal only analysis may be run by including the word **THERMAL** in the execution line.

The **INIT** (or **sw1.** can be used instead) command on the execution line causes the calculation to run just one cycle followed by termination with a full restart file. No editing of the

input deck is required. The calculation can then be restarted with or without any additional input. Sometimes this option can be used to reduce the memory on restart if the required memory is given on the execution line and is specified too large in the beginning when the amount of required memory is unknown. Generally, this option would be used at the beginning of a new calculation.

If the word **MEMORY** is found anywhere on the execution line and if it is not set via (**=nwds**) LS-DYNA will give the default size of memory, request, and then read in the desired memory size. This option is necessary if the default value is insufficient memory and termination occurs as a result. Occasionally, the default value is too large for execution and this option can be used to lower the default size. Memory can also be specified on the ***KEYWORD** card.

SENSE SWITCH CONTROLS

The status of an in-progress LS-DYNA simulation can be determined by using the sense switch. On UNIX versions, this is accomplished by first typing a "^C" (Control-C). This sends an interrupt to LS-DYNA which is trapped and the user is prompted to input the sense switch code. LS-DYNA has nine terminal sense switch controls that are tabulated below:

Type	Response
SW1.	A restart file is written and LS-DYNA terminates.
SW2.	LS-DYNA responds with time and cycle numbers.
SW3.	A restart file is written and LS-DYNA continues.
SW4.	A plot state is written and LS-DYNA continues.
SW5.	Enter interactive graphics phase and real time visualization.
SW7.	Turn off real time visualization.
SW8.	Interactive 2D rezoner for solid elements and real time visualization.
SW9.	Turn off real time visualization (for option SW8).
SWA.	Flush ASCII file buffers.

Type	Response (Implicit Mode Only)
lprint	Enable/Disable printing of equation solver memory, cpu requirements.
nlprint	Enable/Disable printing of nonlinear equilibrium iteration information.
iter	Enable/Disable output of binary plot database "d3iter" showing mesh after each equilibrium iteration. Useful for debugging convergence problems.
conv	Temporarily override nonlinear convergence tolerances.
stop	Halt execution immediately, closing open files.

On UNIX/LINUX systems the sense switches can still be used if the job is running in the background or in batch mode. To interrupt LS-DYNA simply create a file called D3KIL containing the desired sense switch, e.g., "sw1." LS-DYNA periodically looks for this file and if found, the sense switch contained therein is invoked and the D3KIL file is deleted. A null D3KIL file is equivalent to a "sw1."

When LS-DYNA terminates, all scratch files are destroyed: the restart file, plot files, and high-speed printer files remain on disk. Of these, only the restart file is needed to continue the interrupted analysis.

Procedure for LS-DYNA/MPP

As described above the serial/SMP code supports the use of the SIGINT signal (usually Ctrl-C) to interrupt the execution and prompt the user for a "sense switch." The MPP code also supports this capability. However, on many systems a shell script or front end program (generally "mpirun") is required to start MPI applications. Pressing Ctrl-C on some systems will kill this process, and thus kill the running MPP-DYNA executable. As a workaround, when the MPP code begins execution it creates a file named "bg_switch" in the current working directory. This file contains the following single line:

rsh <machine name> kill -INT <PID>

where <machine name> is the hostname of the machine on which the root MPP-DYNA process is running, and <PID> is its process id. (on HP systems, "rsh" is replaced by "remsh"). Thus, simply executing this file will send the appropriate signal.

For more information about running the LS-DYNA/MPP Version see Appendix O.

Files: Input and Output

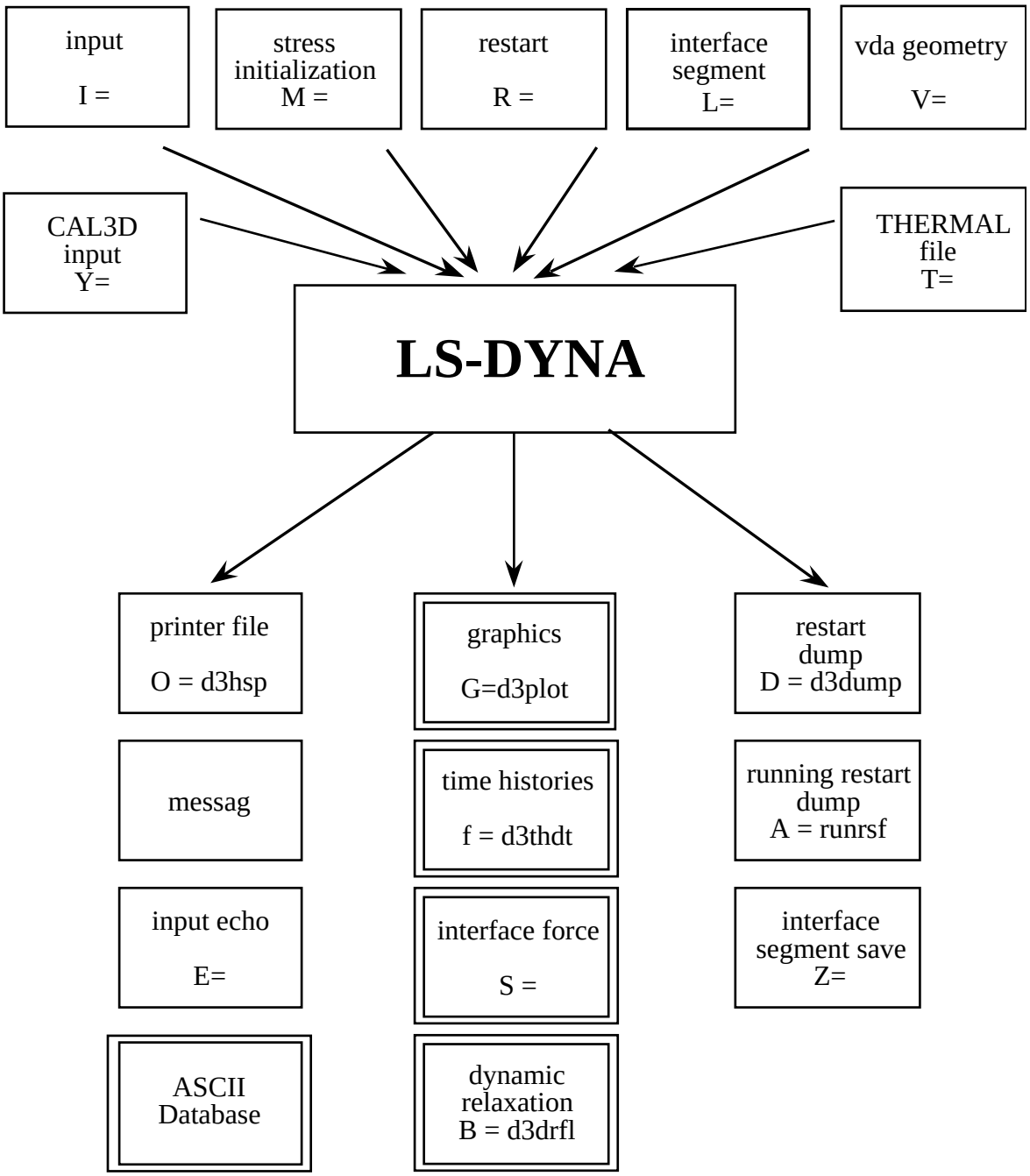


Figure GS.2

File names must be unique. The interface force file is created only if it is specified on the execution line (S=iff). On large problems the default file sizes may not be large enough for a single file to hold either a restart dump or a plot state. Then the file size may be increased by specifying the file size on the execute line using X=**scl**. The default file size holds seven times one-million octal word (262144) or 1835008 words. If the core required by LS-DYNA requires more space, it is recommended that the **scl** be increased appropriately. Using C=**cpu** defines the maximum cpu usage allowed that if exceeded will cause LS-DYNA to terminate with a restart file. During a restart, **cpu** should be set to the total cpu used up to the current restart plus whatever amount of additional time is wanted.

When **restarting from a dump file**, the execution line becomes

LS-DYNA I=**inf** O=**otf** G=**ptf** D=**dpf** R=**rtf** F=**thf** U=**xtf** T=**tpf** A=**rrd** J=**jif** S=**iff** Z=**isf1** L=**isf2** B=**rlf** W=**root** E=**efl** X=**scl** C=**cpu** K=**kill** Q=*option* **KEYWORD MEMORY=nwds**

where

rtf = restart filename.

The adaptive dump files contain all information required to successfully restart so that no other files are needed except when CAD surface data is used. When restarting a problem that uses VDA/IGES surface data, the vda input file must be specified, e.g.:

LS-DYNA R=**d3dump01** V=**vda .....**

If the data from the last run is to be remapped onto a new mesh, then specify: Q=**remap.** The remap file is the dump file from which the remapping data is taken. The remap option is available for brick elements only. File name dropouts are permitted; for example, the following execution lines are acceptable.

LS-DYNA I=**inf**
LS-DYNA R=**rtf**

Default names for the output file, binary plot files, and the dump file are D3HSP, D3PLOT, D3THDT, and D3DUMP, respectively.

For an analysis using interface segments the execution line in the first analysis is given by:

LS-DYNA **I=inf Z=isf1**

and in the second by:

LS-DYNA **I=inf L=isf1**

Batch execution in some installations (e.g., GM) is controlled by file NAMES on unit 88. NAMES is a 2 line file in which the second line is blank. The first line of NAMES contains the execution line:

I=inf

if this is the initial run. For a restart the execution line becomes:

I=inf R=rtf

Remark: No stress initialization is possible at restart. Also the VDA files and the CAL3D files cannot be changed.

RESTART ANALYSIS

The LS-DYNA restart capability allows analyses to be broken down into stages. After the completion of each stage in the calculation a "restart dump" is written that contains all information necessary to continue the analysis. The size of this "dump" file is roughly the same size as the memory required for the calculation. Results can be checked at each stage by post-processing the output databases in the normal way, so the chance of wasting computer time on incorrect analyses is reduced. The restart capability is frequently used to modify models by deleting excessively distorted elements, materials that are no longer important, and contact surfaces that are no longer needed. Output frequencies of the various databases can also be altered. Often, these simple modifications permit the calculation to continue on to a successful completion. Restarting can also help to diagnose why a model is giving problems. By restarting from a dump that is written before the occurrence of a numerical problem and obtaining output at more frequent intervals, it is often possible to identify where the first symptoms appear and what aspect of the model is causing them.

The format of the restart input file is described in this manual. If, for example, the user wishes to restart the analysis from dump state *nn*, contained in file *D3DUMPnn*, then the following procedure is followed:

1. Create the restart input deck, if required, as described in the Restart Section of this manual. Call this file *restartinput.*

2. By invoking the execution line:

   LS-DYNA I=*restartinput* R=*D3DUMPnn*

   execution begins. If no alterations to the model are made, then the execution line:

   LS-DYNA R=*D3DUMPnn*

   will suffice. Of course, the other output files should be assigned names if the defaults have been changed in the original run.

The R=D3DUMPnn on the status line informs the program that this is a restart analysis.

The full deck restart option allows the user to begin a new analysis, with deformed shapes and stresses carried forward from a previous analysis for selected materials. The new analysis can be different from the original, e.g., more contact surfaces, different geometry (of parts which are not carried forward), etc. Examples of applications include:

- Crash analysis continued with extra contact surfaces;
- Sheet metalforming continued with different tools for modeling a multi-stage forming process.

Assume an analysis is run using the input file, job1.inf, and a restart dump named d3dump01 is created. A new input file job2.inf is generated and submitted as a restart with R=d3dump01 as the dump file. The input file job2.inf contains the entire model in its original undeformed state but with more contact surfaces, new output databases, and so on. Since this is a restart job, information must be given to tell LS-DYNA which parts of the model should be initialized in the full deck restart. When the calculation begins the restart database contained in

the file d3dump01 is read, and a new database is created to initialize the model in the input file, job2.inf. The data in file job2.inf is read and the LS-DYNA proceeds through the entire input deck and initialization. At the end of the initialization process, all the parts selected are initialized from the data saved from d3dump01. This means that the deformed position and velocities of the nodes on the elements of each part, and the stresses and strains in the elements (and, if the material of the part is rigid, the rigid body properties) will be assigned.

It is assumed during this process that any initialized part has the same elements, in the same order, with the same topology, in job1 and job2. If this is not the case, the parts cannot be initialized. However, the parts may have different identifying numbers.

For discrete elements and seat belts, the choice is all or nothing. All discrete and belt elements, retractors, sliprings, pretensioners and sensors must exist in both files and will be initialized.

Materials which are not initialized will have no initial deformations or stresses. However, if initialized and non-initialized materials have nodes in common, the nodes will be moved by the initialized material causing a sudden strain in the non-initialized material. This effect could give rise to sudden spikes in loading.

Points to note are:

- Time and output intervals are continuous with job1, i.e., the time is not reset to zero.
- Don’t try to use the restart part of the input to change anything since this will be overwritten by the new input file.
- Usually, the complete input file part of job2.in1 will be copied from job1.inf, with the required alterations. We again mention that there is no need to update the nodal coordinates since the deformed shapes of the initialized materials will be carried forward from job1.
- Completely new databases will be generated with the time offset.

VDA/IGES DATABASES

VDA surfaces are surfaces of geometric entities which are given in the form of polynomials. The format of these surfaces is as defined by the German automobile and supplier industry in the VDA guidelines, [VDA 1987].

The advantage of using VDA surfaces is twofold. First, the problem of meshing the surface of the geometric entities is avoided and, second, smooth surfaces can be achieved which are very important in metalforming. With smooth surfaces, artificial friction introduced by standard faceted meshes with corners and edges can be avoided. This is a big advantage in springback calculations.

A very simple and general handling of VDA surfaces is possible allowing arbitrary motion and generation of surfaces. For a detailed description, see Appendix L.

LS-PrePost®

LS-DYNA is designed to operate with a variety of commercial pre- and post-processing packages. Currently, direct support is available from TRUEGRID, PATRAN, eta/VPG, HYPERMESH, EASi-CRASH DYNA and FEMAP. Several third-party translation programs are available for PATRAN and IDEAS.

Alternately, the pre- and post-processor LS-PrePost is available from LSTC and is specialized for LS-DYNA. LS-PrePost is an advanced pre- and post-processor that is delivered free with LS-DYNA. The user interface is designed to be both efficient and intuitive. LS-PrePost runs on Windows, Linux, and Unix, utilizing OpenGL graphics to achieve fast model rendering and XY plotting.

Some of the capabilities available in LS-PrePost are:

- Complete support for all LS-DYNA keyword data.
- Importing and combining multiple models from many sources (LS-DYNA keyword, IDEAS neutral file, NASTRAN bulk data, STL ascii, and STL binary formats).
- Improved renumbering of model entities.
- Model Manipulation: Translate, Rotate, Scale, Project, Offset, Reflect
- LS-DYNA Entity Creation: Coordinate Systems, Sets, Parts, Masses, CNRBs, Boxes, Spot welds, SPCs, Rigidwalls, Rivets, Initial Velocity, Accelerometers, Cross Sections, etc.
- Mesh Generation: 2Dmesh Sketchboard, nLine Meshing, Line sweep into shell, Shell sweep into solid, Tet-Meshing, Automatic surface meshing of IGES and VDA data, Meshing of simple geometric objects (Plate, Sphere, Cylinder)
- Special Applications: Airbag folding, Dummy positioning, Seatbelt fitting, Initial penetration check, Spot weld generation using MAT_100
- Complete support of LS-DYNA results data file: d3plot file, d3thdt file, All ascii time history data file, Interface force file

LS-PrePost processes output from LS-DYNA. LS-PrePost reads the binary plot-files generated by LS-DYNA and plots contours, fringes, time histories, and deformed shapes. Color contours and fringes of a large number of quantities may be interactively plotted on meshes consisting of plate, shell, and solid type elements. LS-PrePost can compute a variety of strain measures, reaction forces along constrained boundaries.

LS-DYNA generates three binary databases. One contains information for complete states at infrequent intervals; 50 to 100 states of this sort is typical in a LS-DYNA calculation. The second contains information for a subset of nodes and elements at frequent intervals; 1000 to 10,000 states is typical. The third contains interface data for contact surfaces.

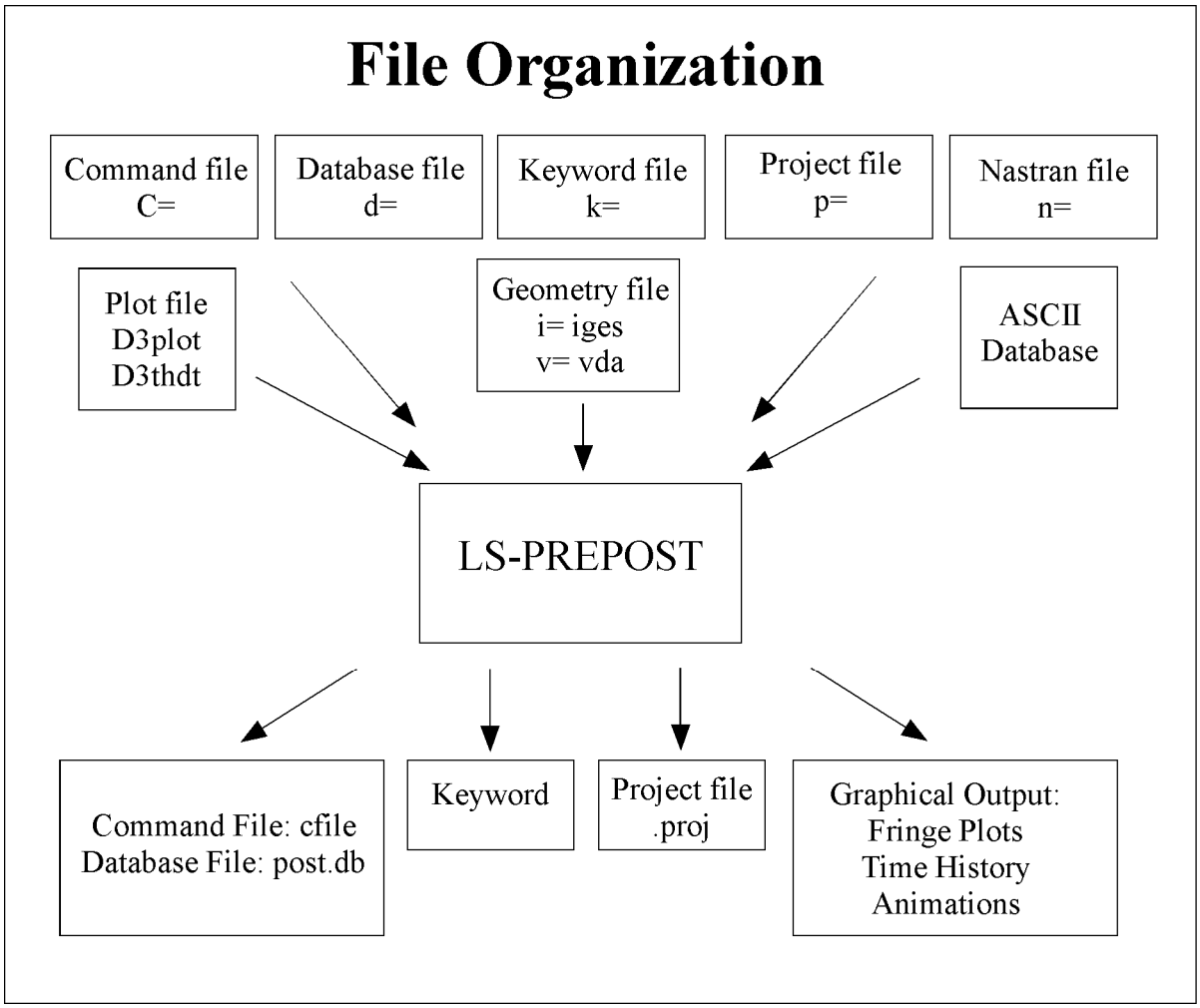

File Organization
Command file
C=
Database file
d=
Keyword file
k=
Project file
p=
Nastran file
n=
Plot file
D3plot
D3thdt
Geometry file
i= iges
v= vda
ASCII
Database
LS-PREPOST
Command File: cfile
Database File: post.db
Keyword
Project file
.proj
Graphical Output:
Fringe Plots
Time History
Animations

Figure GS.3

EXECUTION SPEEDS

The relative execution speeds for various elements in LS-DYNA are tabulated below:

Element Type	Relative Cost
8 node solid with 1 point integration and default hourglass control	4
as above but with Flanagan-Belytschko hourglass control	5
constant stress and Flanagan-Belytschko hourglass control, i.e., the Flanagan-Belytschko element	7
4 node Belytschko-Tsay shell with four thickness integration points	4
4 node Belytschko-Tsay shell with resultant plasticity	3
BCIZ triangular shell with four thickness integration points	7
C^0 triangular shell with four thickness integration points	4
2 node Hughes-Liu beam with four integration points	9
2 node Belytschko-Schwer beam	2
2 node simple truss elements	1
8 node solid-shell with four thickness integration points	11

These relative timings are very approximate. Each interface node of the sliding interfaces is roughly equivalent to one-half zone cycle in cost. Figure GS.3 illustrates the relative cost of the various shell formulations in LS-DYNA.

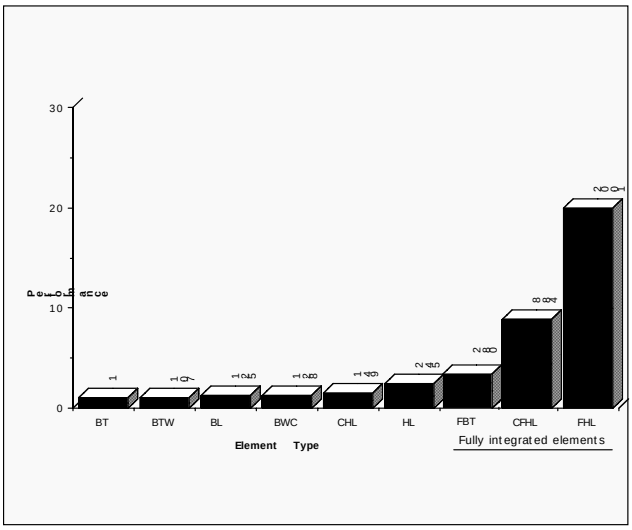


Figure GS.4 Relative cost of the four noded shells available in LS-DYNA where BT is the Belytschko-Tsay shell, BTW is the Belytschko-Tsay shell with the warping stiffness taken from the Belytschko-Wong-Chiang, BWC, shell. The BL shell is the Belytschko-Leviathan shell. CHL denotes the Hughes-Liu shell, HL, with one point quadrature and a co-rotational formulation. FBT is a Belytschko-Tsay like shell with full integration, FHL is the fully integrated Hughes-Liu shell, and the CFHL shell is its co-rotational version.

UNITS

The units in LS-DYNA must be consistent. One way of testing whether a set of units is consistent is to check that:

1 (force unit) = 1 (mass unit) x 1 (acceleration unit)

and that 1 (acceleration unit) = $\frac{1(\text{lengthunit})}{\left[1\left(\text{timeunit}\right)\right]^2}$

Examples of sets of consistent units are:

	(a)	(b)	(c)
Length unit	meter	millimeter	millimeter
Time unit	second	second	millisecond
Mass unit	kilogram	tonne	kilogram
Force unit	Newton	Newton	kiloNewton
Young's Modulus of Steel	210.0E+09	210.0E+03	210.0
Density of Steel	7.85E+03	7.85E–09	7.85E–06
Yield stress of Mild Steel	200.0E+06	200.0	0.200
Acceleration due to gravity	9.81	9.81E+03	9.81E-03
Velocity equivalent to 30 mph	13.4	13.4E+03	13.4

GENERAL CARD FORMAT

The following sections specify for each keyword the cards that have to be defined. Each card is defined in its rigid format form and is shown as a number of fields in an 80 character string. **Most cards are 8 fields with a length of 10 and a sample card is shown below.**

Card Format

	1	2	3	4	5	6	7	8
Variable	NSID	PSID	A1	A2	A3	KAT		
Type	I	I	F	F	F	I		
Default	none	none	1.0	1.0	0	1		
Remarks	1			2		3		

The type is the variable type and is either F, for floating point or I, for an integer. The default gives the value set if zero is specified, the field is left blank or the card is not defined. The remarks refer to comments at the end of the section. The card format is given above the card if it is other than eight fields of 10. Free formats may be used with the data separated by commas. When using comma format, the number of characters used to specify a number must not exceed

the number which would fit into the equivalent rigid format field. An I8 number is limited to a number of 99999999 and larger numbers with more than eight characters are unacceptable. Rigid and free formats can be mixed throughout the deck but not within a card.

*AIRBAG

Purpose: Define an airbag or control volume.

The keyword *AIRBAG provides a way of defining thermodynamic behavior of the gas flow into the airbag as well as a reference configuration for the fully inflated bag. The keyword cards in this section are defined in alphabetical order:

AIRBAG_OPTION1*_{*OPTION2*}_{*OPTION3*}_{*OPTION4*}
***AIRBAG_ADVANCED_ALE**
***AIRBAG_ALE**
***AIRBAG_INTERACTION**
***AIRBAG_PARTICLE**
AIRBAG_REFERENCE_GEOMETRY_OPTION_OPTION*
***AIRBAG_SHELL_REFERENCE_GEOMETRY**

AIRBAG_OPTION1*_{*OPTION2*}_{*OPTION3*}_{*OPTION4*}

OPTION1 specifies one of the following thermodynamic relationships:

SIMPLE_PRESSURE_VOLUME
SIMPLE_AIRBAG_MODEL
ADIABATIC_GAS_MODEL
WANG_NEFSKE
WANG_NEFSKE_JETTING
WANG_NEFSKE_MULTIPLE_JETTING
LOAD_CURVE
LINEAR_FLUID
HYBRID
HYBRID_JETTING
HYBRID_CHEMKIN

OPTION2 specifies that an additional line of data is read for the WANG_NEFSKE type thermodynamic relationships. The additional data controls the initiation of exit flow from the airbag. *OPTION2* takes the single option:

POP

OPTION3 specifies that a constant momentum formulation is used to calculate the jetting load on the airbag an additional line of data is read in: *OPTION3* takes the single option:

CM

OPTION4 given by:

ID

Specifies that an airbag ID and heading information will be the first card of the airbag definition. This ID is a unique number that is necessary for the identification of the airbags in the definition of airbag interaction via *AIRBAG_INTERACTION keyword. The numeric ID's and heading are written into the ABSTAT and D3HSP files.

The following card is read if and only if the ID option is specified.
Note: An ID is necessary for *AIRBAG_INTERACTION.

Optional 1 2-8

Variable	ABID	HEADING
Type	I	A70

VARIABLE	DESCRIPTION
ABID	Contact interface ID. This must be a unique number.
HEADING	Airbag descriptor. It is suggested that unique descriptions be used.

Card	1	2	3	4	5	6	7	8
Variable	SID	SIDTYP	RBID	VSCA	PSCA	VINI	MWD	SPSF
Type	I	I	I	F	F	F	F	F
Default	none	0	0	1.	1.	0.	0.	0.
Remarks			optional					

VARIABLE	DESCRIPTION
SID	Set ID
SIDTYP	Set type: EQ.0: segment, NE.0: part set ID.
RBID	Rigid body part ID for user defined activation subroutine: EQ.-*RBID*: Sensor subroutine flags initiates the inflator. Load curves are offset by initiation time, EQ.0: the control volume is active from time zero, EQ. *RBID*: User sensor subroutine flags the start of the inflation. Load curves are offset by initiation time. See Appendix D.
VSCA	Volume scale factor, V_{sca} (default=1.0)
PSCA	Pressure scale factor, P_{sca} (default=1.0)
VINI	Initial filled volume, V_{ini}
MWD	Mass weighted damping factor, D
SPSF	Stagnation pressure scale factor, $0 <= \gamma <= 1$

Remarks:

The first card is necessary for all airbag options. The sequence for the following cards which is different for each option is explained on the next pages.

Lumped parameter control volumes are a mechanism for determining volumes of closed surfaces and applying a pressure based on some thermodynamic relationships. The volume is specified by a list of polygons similar to the pressure boundary condition cards or by specifying a material subset which represents shell elements which form the closed boundary. All polygon normals must be oriented to face outwards from the control volume. If holes are detected, they are assumed to be covered by planar surfaces.

V_{sca} and P_{sca} allow for unit system changes from the inflator to the finite element model. There are two sets of volume and pressure used for each control volume. First, the finite element model computes a volume ($V_{femodel}$) and applies a pressure ($P_{femodel}$). The thermodynamics of a control volume may be computed in a different unit system; thus, there is a separate volume ($V_{cvolume}$) and pressure ($P_{cvolume}$) which are used for integrating the differential equations for the control volume. The conversion is as follows:

$$V_{cvolume} = \left(V_{sca} V_{femodel}\right) - V$$

$$P_{femodel} = P_{sca} P_{cvolume}$$

Damping can be applied to the structure enclosing a control volume by using a mass weighted damping formula:

$$F_i^d = m_i D\left(v_i - v_{cg}\right)$$

where F_i^d is the damping force, m_i is the nodal mass, v_i is the velocity for a node, v_{cg} is the mass weighted average velocity of the structure enclosing the control volume, and D is the damping factor.

An alternative, separate damping is based on the stagnation pressure concept. The stagnation pressure is roughly the maximum pressure on a flat plate oriented normal to a steady state flow field. The stagnation pressure is defined as $p = \gamma \rho V^2$ where V is the normal velocity of the control volume relative to the ambient velocity, ρ is the ambient air density, and γ is a factor which varies from 0 to 1 and has to be chosen by the user. Small values are recommended to avoid excessive damping.

Sensor Input to Activate Inflator

Define if and only if *RBID* nonzero.

Skip this input if *RBID*=0. If the rigid body ID is non-zero then define either the input for the user defined sensor subroutine (A) or define the data for the default sensor (B).

The sensor is mounted on a rigid body which is attached to the structure. The motion of the sensor is provided in the local coordinate system defined for the rigid body in the definition of the rigid material, see *MAT_RIGID. This is important since the default local system is taken as the principal axes of the inertia tensor. The local system rotates and translates with the rigid material. When the user defined criterion is met for the deployment of the airbag, a flag is set and the deployment begins. All load curves relating to the mass flow rate versus time are then shifted by the initiation time.

A. Sensor Input for User Subroutine (*RBID*>0)
See Appendix D. A user supplied subroutine must be provided.

Define the following card sets which provide the input parameters for the user defined subroutine. Up to 25 parameters may be used with each control volume.

Card	1	2	3	4	5	6	7	8
Variable	N							
Type	I							
Default	none							

Card Format (Define up to 25 constants for the user subroutine. Input only the number of cards necessary, i.e. for nine constants use 2 cards)

Card	1	2	3	4	5	6	7	8
Variable	C1	C2	C3	C4	C5			
Type	F	F	F	F	F			
Default	0.	0.	0.	0.	0.			

VARIABLE	DESCRIPTION
N	Number of input parameters (not to exceed 25).
C1,...CN	Up to 25 constants for the user subroutine.

B. LS-DYNA Sensor Input (*RBID*<0)

Define three cards which provide the input parameters for the built in sensor subroutine.

Acceleration/Velocity/Displacement Activation

Card	1	2	3	4	5	6	7	8
Variable	AX	AY	AZ	AMAG	TDUR			
Type	F	F	F	F	F			
Default	0.	0.	0.	0.	0.			

Card

Variable	DVX	DVY	DVZ	DVMAG				
Type	F	F	F	F				
Default	0.	0.	0.	0.				

Card

Variable	UX	UY	UZ	UMAG				
Type	F	F	F	F				
Default	0.	0.	0.	0.				

VARIABLE	DESCRIPTION
AX	Acceleration level in local x-direction to activate inflator. The absolute value of the x-acceleration is used. EQ.0: inactive.
AY	Acceleration level in local y-direction to activate inflator. The absolute value of the y-acceleration is used. EQ.0: inactive.
AZ	Acceleration level in local z-direction to activate inflator. The absolute value of the z-acceleration is used. EQ.0: inactive.
AMAG	Acceleration magnitude required to activate inflator. EQ.0: inactive.
TDUR	Time duration acceleration must be exceeded before the inflator activates. This is the cumulative time from the beginning of the calculation, i.e., it is not continuous.
DVX	Velocity change in local x-direction to activate the inflator. (The absolute value of the velocity change is used.) EQ.0: inactive.
DVY	Velocity change in local y-direction to activate the inflator. (The absolute value of the velocity change is used.) EQ.0: inactive.
DVZ	Velocity change in local z-direction to activate the inflator. (The absolute value of the velocity change is used.) EQ.0: inactive.
DVMAG	Velocity change magnitude required to activate the inflator. EQ.0: inactive.
UX	Displacement increment in local x-direction to activate the inflator. (The absolute value of the x-displacement is used.) EQ.0: inactive.
UY	Displacement increment in local y-direction to activate the inflator. (The absolute value of the y-displacement is used.) EQ.0: inactive.
UZ	Displacement increment in local z-direction to activate the inflator. (The absolute value of the z-displacement is used.) EQ.0: inactive.
UMAG	Displacement magnitude required to activate the inflator. EQ.0: inactive.

Additional card required for SIMPLE_PRESSURE_VOLUME option

Card	1	2	3	4	5	6	7	8
Variable	CN	BETA	LCID	LCIDDR				
Type	F	F	I	I				
Default	none	none	none	0				

VARIABLE	DESCRIPTION
CN	Coefficient. Define if the load curve ID, LCID, is unspecified. LT.0.0: |CN| is the load curve ID, which defines the coefficient as a function of time.
BETA	Scale factor, β. Define if a load curve ID is not specified.
LCID	Optional load curve ID defining pressure versus relative volume.
LCIDDR	Optional load curve ID defining the coefficient, CN, as a function of time during the dynamic relaxation phase.

Remarks:

The relationship is the following:

$$Pressure = \beta \frac{CN}{RelativeVolume}$$

$$RelativeVolume = \frac{CurrentVolume}{InitialVolume}$$

The pressure is then a function of the ratio of current volume to the initial volume. The constant, CN, is used to establish a relationship known from the literature. The scale factor β is simply used to scale the given values. This simple model can be used when an initial pressure is given and no leakage, no temperature, and no input mass flow is assumed. A typical application is the modeling of air in automobile tires.

The load curve, LCIDDR, can be used to ramp up the pressure during the dynamic relaxation phase in order to avoid oscillations after the desired gas pressure is reached. In the DEFINE_CURVE section this load curve must be flagged for dynamic relaxation. After initialization either the constant or load curve ID, |CN| is used to determine the pressure.

Additional cards required for SIMPLE_AIRBAG_MODEL option

Card 1

Variable	CV	CP	T	LCID	MU	A	PE	RO
Type	F	F	F	I	F	F	F	F
Default	none	none	none	none	none	none	none	none

Card 2

Variable	LOU	TEXT	A	B	MW	GASC		
Type	I	F	F	F	F	F		
Default	0	0.	0.	0.	0.	0.		
Remarks	0	optional	optional	optional	optional	optional		

VARIABLE	DESCRIPTION
CV	Heat capacity at constant volume
CP	Heat capacity at constant pressure
T	Temperature of input gas
LCID	Load curve ID specifying input mass flow rate. See *DEFINE_CURVE.
MU	Shape factor for exit hole, μ: LT.0.0: $\lvert\mu\rvert$ is the load curve number defining the shape factor as a function of absolute pressure.
A	Exit area, A: GE.0.0: A is the exit area and is constant in time, LT.0.0: |A| is the load curve number defining the exit area as a function of absolute pressure.
PE	Ambient pressure, p_e

VARIABLE	DESCRIPTION
RO	Ambient density, ρ
LOU	**Optional** load curve ID giving mass flow out versus gauge pressure in bag. See *DEFINE_CURVE.
TEXT	Ambient temperature. (Define if and only if CV=0.)
A	First heat capacity coefficient of inflator gas (e.g., Joules/mole/^{o}K). (Define if and only if CV=0.)
B	Second heat capacity coefficient of inflator gas, (e.g., Joules/mole/^{o}K^2). (Define if and only if CV=0.)
MW	Molecular weight of inflator gas (e.g., Kg/mole). (Define if and only if CV=0.)
GASC	Universal gas constant of inflator gas (e.g., 8.314 Joules/mole/^{o}K). (Define if and only if CV=0.)

Remarks:

The gamma law equation of state used to determine the pressure in the airbag:

$$p = (\gamma - 1)\rho e$$

where p is the pressure, ρ is the density, e is the specific internal energy of the gas, and γ is the ratio of the specific heats:

$$\gamma = \frac{c_p}{c_v}$$

From conservation of mass, the time rate of change of mass flowing into the bag is given as:

$$\frac{dM}{dt} = \frac{dM_{in}}{dt} - \frac{dM_{out}}{dt}$$

The inflow mass flow rate is given by the load curve ID, LCID. Leakage, the mass flow rate out of the bag, can be modeled in two alternative ways. One is to give an exit area with the corresponding shape factor, then the load curve ID, LOU, must be set to zero. The other is to define a mass flow out by a load curve, then μ and A have to both be set to zero.

If CV=0. then the constant-pressure specific heat is given by:

$$c_p = \frac{(a + bT)}{MW}$$

and the constant-volume specific heat is then found from:

$$c_v = c_p - \frac{R}{MW}$$

Additional card required for ADIABATIC_GAS_MODEL option

Card	1	2	3	4	5	6	7	8
Variable	PSF	LCID	GAMMA	P0	PE	RO		
Type	F	I	F	F	F	F		
Default	1.0	none	none	none	none	none		

VARIABLE	DESCRIPTION
PSF	Pressure scale factor
LCID	Optional load curve for preload flag. See *DEFINE_CURVE.
GAMMA	Ratio of specific heats
P0	Initial pressure (gauge)
PE	Ambient pressure
RO	Initial density of gas

Remarks:

The optional load curve ID, LCID, defines a preload flag. During the preload phase the function value of the load curve versus time is zero, and the pressure in the control volume is given as:

$$p = PSF\ p_0$$

When the **first nonzero** function value is encountered, the preload phase stops and the ideal gas law applies for the rest of the analysis. If LCID is zero, no preload is performed.

The gamma law equation of state for the adiabatic expansion of an ideal gas is used to determine the pressure after preload:

$$p = (\gamma - 1)\rho e$$

where p is the pressure, ρ is the density, e is the specific internal energy of the gas, and γ is the ratio of the specific heats:

$$\gamma = \frac{c_p}{c_v}$$

The pressure above is the absolute pressure, the resultant pressure acting on the control volume is:

$$p_s = PSF\left(p - p_e\right)$$

where PSF is the pressure scale factor. Starting from the initial pressure p_0 an initial internal energy is calculated:

$$e_0 = \frac{p_0 + p_e}{\rho\left(\gamma - 1\right)}$$

Additional 4 cards are required for all WANG_NEFSKE models

Card 1

	1	2	3	4	5	6	7	8
Variable	CV	CP	T	LCT	LCMT	TVOL	LCDT	IABT
Type	F	F	F	I	I	F	I	F
Default	none	none	0.	0	none	0.	0.	not used

Card 2

Variable	C23	LCC23	A23	LCA23	CP23	LCCP23	AP23	LCAP23
Type	F	I	F	I	F	I	F	I
Default	none	0	none	0	none	0	0.0	0

Card 3

Variable	PE	RO	GC	LCEFR	POVER	PPOP	OPT	KNKDN
Type	F	F	F	I	F	F	F	I
Default	none	none	none	0	0.0	0.0	0.0	0

If the inflator is modeled, LCMT=0, define, the following card. If not, define but leave blank.

Card 4	1	2	3	4	5	6	7	8
Variable	IOC	IOA	IVOL	IRO	IT	LCBF		
Type	F	F	F	F	F	I		
Default	none	none	none	none	none	none		

Define the following card if and only if CV=0. This option allows temperature dependent heat capacities to be defined. See below.

Card 5	1	2	3	4	5	6	7	8
Variable	TEXT	A	B	MW	GASC	HCONV		
Type	F	F	F	F	F	F		
Default	none	none	none	none	none	none		

Define the following card if and only if the POP option is specified. Use this option to specify additional criteria for initiating exit flow from the airbag.

Card 5	1	2	3	4	5	6	7	8
Variable	TDP	AXP	AYP	AZP	AMAGP	TDURP	TDA	RBIDP
Type	F	F	F	F	F	F	F	I
Default	0.0	0.0	0.0	0.0	0.0	0.0	0.0	none

VARIABLE	DESCRIPTION
CV	Heat capacity at constant volume
CP	Heat capacity at constant pressure

VARIABLE	DESCRIPTION
T	Temperature of input gas. For temperature variations a load curve, LCT, may be defined.
LCT	Optional load curve number defining temperature of input gas versus time. This overrides columns T.
LCMT	Load curve specifying input mass flow rate or tank pressure versus time. If the tank volume, TVOL, is nonzero the curve ID is assumed to be tank pressure versus time. If LCMT=0, then the inflator has to be modeled, see Card 4. During the dynamic relaxation phase the airbag is ignored unless the curve is flagged to act during dynamic relaxation.
TVOL	Tank volume which is required only for the tank pressure versus time curve, LCMT.
LCDT	Load curve for time rate of change of temperature (dT/dt) versus time.
IABT	Initial airbag temperature. (Optional, generally not defined.)
C23	Vent orifice coefficient which applies to exit hole. Set to zero if LCC23 is defined below.
LCC23	The absolute value, |LCC23|, is a load curve ID. If the ID is positive, the load curve defines the vent orifice coefficient which applies to exit hole as a function of time. If the ID is negative, the vent orifice coefficient is defined as a function of relative pressure, P_{air} / P_{bag}, see [Anagonye and Wang 1999]. A nonzero value for C23 overrides LCC23.
A23	If defined as a positive number, A23 is the vent orifice area which applies to exit hole. If defined as a negative number, the absolute value |A23| is a part ID, see [Anagonye and Wang, 1999]. The area of this part becomes the vent orifice area. Set A23 to zero if LCA23 is defined below.
LCA23	Load curve number defining the vent orifice area which applies to exit hole as a function of absolute pressure. A nonzero value for A23 overrides LCA23.
CP23	Orifice coefficient for leakage (fabric porosity). Set to zero if LCCP23 is defined below.
LCCP23	Load curve number defining the orifice coefficient for leakage (fabric porosity) as a function of time. A nonzero value for CP23 overrides LCCP23.
AP23	Area for leakage (fabric porosity)

VARIABLE	DESCRIPTION
LCAP23	Load curve number defining the area for leakage (fabric porosity) as a function of (absolute) pressure. A nonzero value for AP23 overrides LCAP23.
PE	Ambient pressure
RO	Ambient density
GC	Gravitational conversion constant (mandatory - no default). If consistent units are being used for all parameters in the airbag definition then unity should be input.
LCEFR	Optional curve for exit flow rate (mass/time) versus (gauge) pressure
POVER	Initial relative overpressure (gauge), P_{over} in control volume
PPOP	Pop Pressure: relative pressure (gauge) for initiating exit flow, P_{pop}
OPT	Fabric venting option, if nonzero CP23, LCCP23, AP23, and LCAP23 are set to zero. EQ. 1: Wang-Nefske formulas for venting through an orifice are used. Blockage is not considered. EQ. 2: Wang-Nefske formulas for venting through an orifice are used. Blockage of venting area due to contact is considered. EQ. 3: Leakage formulas of Graefe, Krummheuer, and Siejak [1990] are used. Blockage is not considered. EQ. 4: Leakage formulas of Graefe, Krummheuer, and Siejak [1990] are used. Blockage of venting area due to contact is considered. EQ. 5: Leakage formulas based on flow through a porous media are used. Blockage is not considered. EQ. 6: Leakage formulas based on flow through a porous media are used. Blockage of venting area due to contact is considered. EQ. 7: Leakage is based on gas volume outflow versus pressure load curve. Blockage of flow area due to contact is not considered. Absolute pressure is used in the porous-velocity-versus-pressure load curve, given as FAC(P) in the *MAT_FABRIC card. EQ. 8: Leakage is based on gas volume outflow versus pressure load curve. Blockage of flow area due to contact is considered. Absolute pressure is used in the porous-velocity-versus-pressure load curve, given as FAC(P) in the *MAT_FABRIC card.
KNKDN	Optional load curve ID defining the knock down pressure scale factor versus time. This option only applies to jetting. The scale factor defined by this load curve scales the pressure applied to airbag segments which

VARIABLE	DESCRIPTION
	do not have a clear line-of-sight to the jet. Typically, at very early times this scale factor will be less than unity and equal to unity at later times. The full pressure is always applied to segments which can see the jets.
IOC	Inflator orifice coefficient
IOA	Inflator orifice area
IVOL	Inflator volume
IRO	Inflator density
IT	Inflator temperature
LCBF	Load curve defining burn fraction versus time
TEXT	Ambient temperature.
A	First heat capacity coefficient of inflator gas (e.g., Joules/mole/^{o}K)
B	Second heat capacity coefficient of inflator gas, (e.g., Joules/mole/^{o}K^{2})
MW	Molecular weight of inflator gas (e.g., Kg/mole).
GASC	Universal gas constant of inflator gas (e.g., 8.314 Joules/mole/^{o}K)
HCONV	Effective heat transfer coefficient between the gas in the air bag and the environment at temperature TEXT. If HCONV<0, then HCONV defines a load curve of data pairs (time, hconv).
TDP	Time delay before initiating exit flow after pop pressure is reached.
AXP	Pop acceleration magnitude in local x-direction. EQ.0.0: Inactive.
AYP	Pop acceleration magnitude in local y-direction. EQ.0.0: Inactive.
AZP	Pop acceleration magnitude in local z-direction. EQ.0.0: Inactive.
AMAGP	Pop acceleration magnitude. EQ.0.0: Inactive.
TDURP	Time duration pop acceleration must be exceeded to initiate exit flow. This is a cumulative time from the beginning of the calculation, i.e., it is not continuous.

TDA	Time delay before initiating exit flow after pop acceleration is exceeded for the prescribed time duration.
RBIDP	Part ID of the rigid body for checking accelerations against pop accelerations.

Remarks:

The gamma law equation of state for the adiabatic expansion of an ideal gas is used to determine the pressure after preload:

$$p = (\gamma - 1)\rho e$$

where p is the pressure, ρ is the density, e is the specific internal energy of the gas, and γ is the ratio of the specific heats:

$$\gamma = \frac{c_p}{c_v}$$

where c_v is the specific heat at constant volume, and c_p is the specific heat at constant pressure. A pressure relation is defined:

$$Q = \frac{p_e}{p}$$

where p_e is the external pressure and p is the internal pressure in the bag. A critical pressure relationship is defined as:

$$Q_{crit} = \left(\frac{2}{\gamma + 1}\right)^{\gamma/\gamma - 1}$$

where γ is the ratio of specific heats:

$$\gamma = \frac{c_p}{c_v}$$

If

$$Q \le Q_{crit} \quad \text{then} \quad Q = Q_{crit}$$

Wang and Nefske define the mass flow through the vents and leakage by

$$\dot{m}_{23} = C_{23} A_{23} \frac{p}{R\sqrt{T_2}} Q^{1/\gamma} \sqrt{2 g_c \left(\frac{\gamma R}{\gamma - 1}\right)\left(1 - Q^{\gamma - 1/\gamma}\right)}$$

and

$$\dot{m}'_{23} = C'_{23}A'_{23}\frac{p}{R\sqrt{T_2}}Q^{1/\gamma}\sqrt{2g_c\left(\frac{\gamma R}{\gamma-1}\right)\left(1-Q^{\gamma-1/\gamma}\right)}$$

It must be noted that the gravitational conversion constant has to be given in consistent units. As an alternative to computing the mass flow out of the bag by the Wang-Nefske model, a curve for the exit flow rate depending on the internal pressure can be taken. Then, no definitions for C23, LCC23, A23, LCA23, CP23, LCCP23, AP23, and LCAP23 are necessary.

The airbag inflator assumes that the control volume of the inflator is constant and that the amount of propellant reacted can be defined by the user as a tabulated curve of fraction reacted versus time. A pressure relation is defined:

$$Q_{crit} = \frac{p_c}{p_i} = \left(\frac{2}{\gamma+1}\right)^{\frac{\gamma}{\gamma-1}}$$

where p_c is a critical pressure at which sonic flow occurs, p_I, is the inflator pressure. The exhaust pressure is given by

$$p_e = p_a \quad if \quad p_a \geq p_c$$
$$p_e = p_c \quad if \quad p_a < p_c$$

where p_a is the pressure in the control volume. The mass flow into the control volume is governed by the equation:

$$\dot{m}_{in} = C_O A_O \sqrt{2p_I\rho_I}\sqrt{\frac{g_c\gamma\left(Q^{\frac{2}{\gamma}} - Q^{\frac{\gamma+1}{\gamma}}\right)}{\gamma-1}}$$

where C_O, A_O, and ρ_I are the inflator orifice coefficient, area, and gas density, respectively.

If OPT is defined, then for OPT set to 1 or 2 the mass flow rate out of the bag, $\dot{m}_{out}$ is given by:

$$\dot{m}_{out} = \sqrt{g_c}\cdot\left[\sum_{n=1}^{nairmats}\left(FLC(t)_n\cdot FAC(p)_n\cdot Area_n\right)\right]\cdot\sqrt{2p\rho}\sqrt{\frac{\gamma\left(Q^{\frac{2}{k}} - Q^{\gamma+1/\gamma}\right)}{\gamma-1}}$$

where, ρ is the density of airbag gas, *nairmats* is the number of fabrics used in the airbag, and $Area_n$ is the current unblocked area of fabric number n.

If OPT set to 3 or 4 then:

$$\dot{m}_{out} = \left[\sum_{n=1}^{nairmats}\left(FLC(t)_n\cdot FAC(p)_n\cdot Area_n\right)\right]\cdot\sqrt{2\left(p-p_{ext}\right)\rho}$$

and for OPT set to 5 or 6:

$$\dot{m}_{out} = \left[\sum_{n=1}^{nairmats} \left(FLC(t)_n \cdot FAC(p)_n \cdot Area_n \right) \right] \cdot \left(p - p_{ext} \right)$$

and for OPT set to 7 or 8 (may be comparable to an equivalent model ALE model):

$$\dot{m}_{out} = \sum_{n=1}^{nairmats} FLC(t)_n \cdot FAC(p)_n \cdot Area_n \cdot \rho_n$$

Note that for different OPT settings, $FAC(p)_n$ has different meanings (all units shown just as demonstrations):

- For OPT of 1, 2, 3 and 4, FAC(P) is unit-less.
- For OPT of 5 and 6, FAC(P) has a unit of (s/m).
- For OPT of 7 or 8, FAC(P) is the gas volume outflow through a unit area per unit time thus has the unit of speed, i.e. $v\dot{o}l \approx m^3/(m^2 \bullet s) \approx m/s \approx vel(P)$.

Multiple airbags may share the same part ID since the area summation is over the airbag segments whose corresponding part ID's are known. Currently, we assume that no more than ten materials are used per bag for purposes of the output. This constraint can be eliminated if necessary.

The total mass flow out will include the portion due to venting, i.e., constants C23 and A23 or their load curves above.

If CV=0. then the constant-pressure specific heat is given by:

$$c_p = \frac{(a + bT)}{MW}$$

and the constant-volume specific heat is then found from:

$$c_v = c_p - \frac{R}{MW}$$

Further additional 2 cards are required for JETTING models

The following additional cards are defined for the WANG_NEFSKE_JETTING and WANG_NEFSKE_MULTIPLE_JETTING options, two further cards are defined for each option. The jet may be defined by specifying either the coordinates of the jet focal point, jet vector head and secondary jet focal point, or by specifying three nodes located at these positions. The nodal point option is recommended when the location of the airbag changes as a function of time.

Define either card below but not both:

1st additional card of 2 required for WANG_NEFSKE_JETTING option

Card 1	1	2	3	4	5	6	7	8
Variable	XJFP	YJFP	ZJFP	XJVH	YJVH	ZJVH	CA	BETA
Type	F	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	none	1.0
Remark	1	1	1	1	1	1		

1st additional card of 2 required for WANG_NEFSKE_MULTIPLE_JETTING option

Card 1	1	2	3	4	5	6	7	8
Variable	XJFP	YJFP	ZJFP	XJVH	YJVH	ZJVH	LCJRV	BETA
Type	F	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	none	1.0
Remark	1	1	1	1	1	1		

2nd additional card of 2 required for WANG_NEFSKE_JETTING and WANG_NEFSKE_MULTIPLE_JETTING option

Card 2	1	2	3	4	5	6	7	8
Variable	XSJFP	YSJFP	ZSJFP	PSID	ANGLE	NODE1	NODE2	NODE3
Type	F	F	F	I	F	I	I	I
Default	none	none	none	none	none	0	0	0
Remark						1	1	1

VARIABLE	DESCRIPTION
XJFP	x-coordinate of jet focal point, i.e., the virtual origin in Figure 1.1. See Remark 1 below.
YJFP	y-coordinate of jet focal point, i.e., the virtual origin in Figure 1.1.
ZJFP	z-coordinate of jet focal point, i.e., the virtual origin in Figure 1.1.
XJVH	x-coordinate of jet vector head to defined code centerline
YJVH	y-coordinate of jet vector head to defined code centerline
ZJVH	z-coordinate of jet vector head to defined code centerline
CA	Cone angle, α, defined in radians. LT.0.0: $\lvert\alpha\rvert$ is the load curve ID defining cone angle as a function of *time*
LCJRV	Load curve ID giving the spatial jet relative velocity distribution, see Figures 1.2 and 1.3. The jet velocity is determined from the inflow mass rate and scaled by the load curve function value corresponding to the value of the angle ψ. Typically, the values on the load curve vary between 0 and unity. See *DEFINE_CURVE.
BETA	Efficiency factor, β, which scales the final value of pressure obtained from Bernoulli's equation. LT.0.0: $\lvert\beta\rvert$ is the load curve ID defining the efficiency factor as a function of *time*

VARIABLE	DESCRIPTION
XSJFP	x-coordinate of secondary jet focal point, passenger side bag. If the coordinates of the secondary point are (0,0,0) then a conical jet (driver's side airbag) is assumed.
YSJFP	y-coordinate of secondary jet focal point
ZSJFP	z-coordinate of secondary jet focal point
PSID	Optional part set ID, see *SET_PART. If zero all elements are included in the airbag.
ANGLE	Cutoff angle in degrees. The relative jet velocity is set to zero for angles greater than the cutoff. See Figure 1.3. This option applies to the MULTIPLE jet only.
NODE1	Node ID located at the jet focal point, i.e., the virtual origin in Figure 1.1. See Remark 1 below.
NODE2	Node ID for node along the axis of the jet.
NODE3	Optional node ID located at secondary jet focal point.

Remarks:

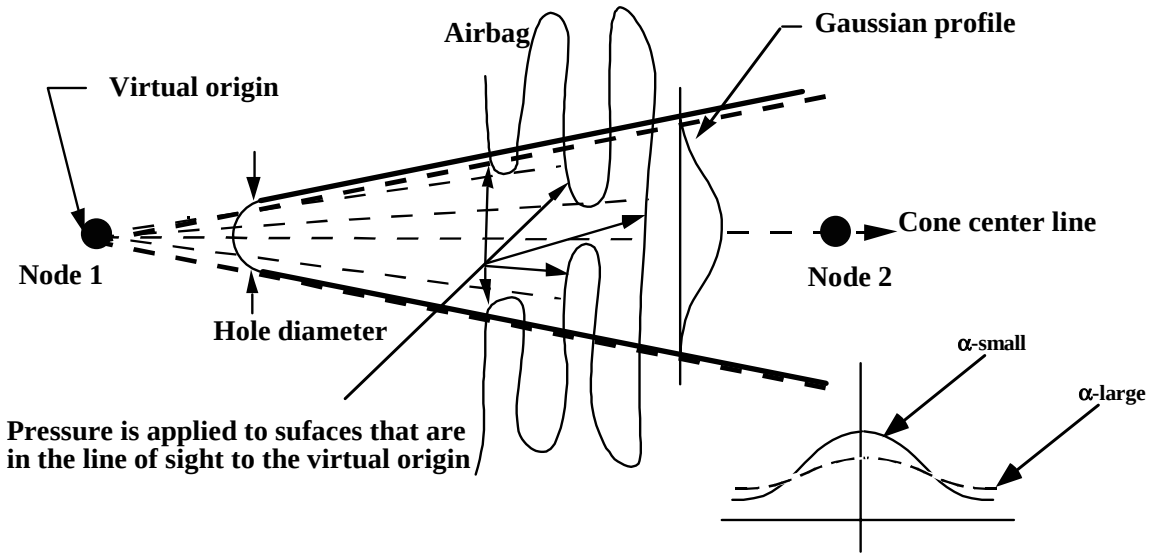
1. It is assumed that the jet direction is defined by the coordinate method (XJFP, YJFP, ZJFP) and (XJVH, YJVH, ZJVH) unless both NODE1 and NODE2 are defined. In which case the coordinates of the nodes give by NODE1, NODE2 and NODE3 will override (XJFP, YJFP, ZJFP) and (XJVH, YJVH, ZJVH). The use of nodes is recommended if the airbag system is undergoing rigid body motion. The nodes should be attached to the vehicle to allow for the coordinates of the jet to be continuously updated with the motion of the vehicle.

   The jetting option provides a simple model to simulate the real pressure distribution in the airbag during the breakout and early unfolding phase. Only the surfaces that are in the line of sight to the virtual origin have an increased pressure applied. With the optional load curve LCRJV, the pressure distribution with the code can be scaled according to the so-called relative jet velocity distribution.

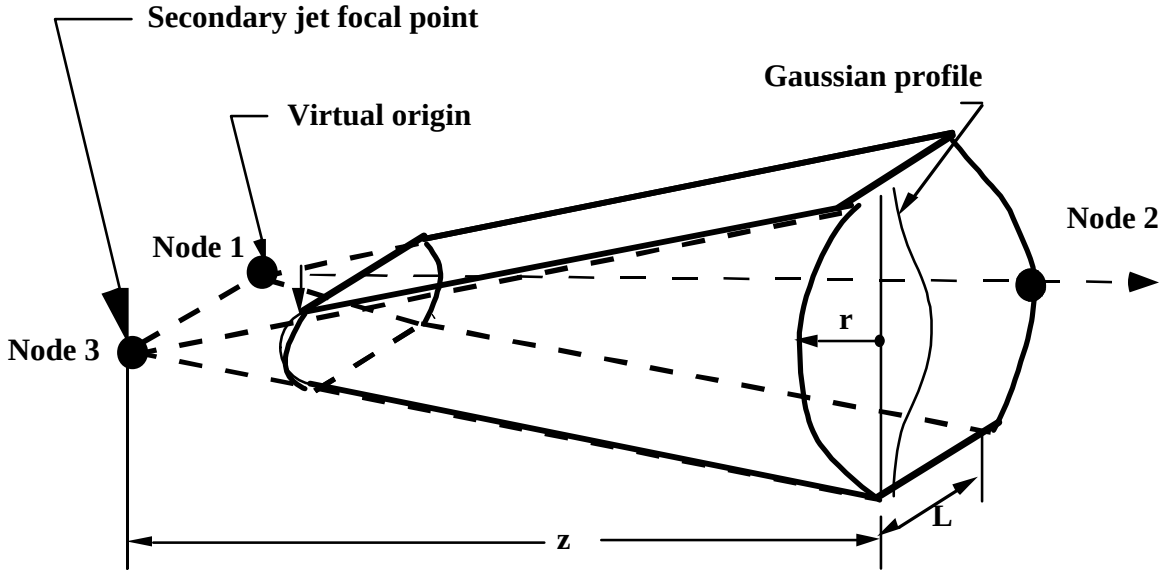
   For passenger side airbags the cone is replaced by a wedge type shape. The first and secondary jet focal points define the corners of the wedge and the angle α then defines the wedge angle.

   Instead of applying pressure to all surfaces in the line of sight of the virtual origin(s), a part set can be defined to which the pressure is applied.

2. Care must be used to place the jet focal point within the bag. If the focal point is outside the bag, inside surfaces will not be visible so jetting pressure will not be applied correctly.



(a.)



(b.)

Figure 1.1 Jetting configuration for (a.) driver's side airbag (pressure applied only if centroid of surface is in line-of-sight) and (b.) the passenger's side bag.

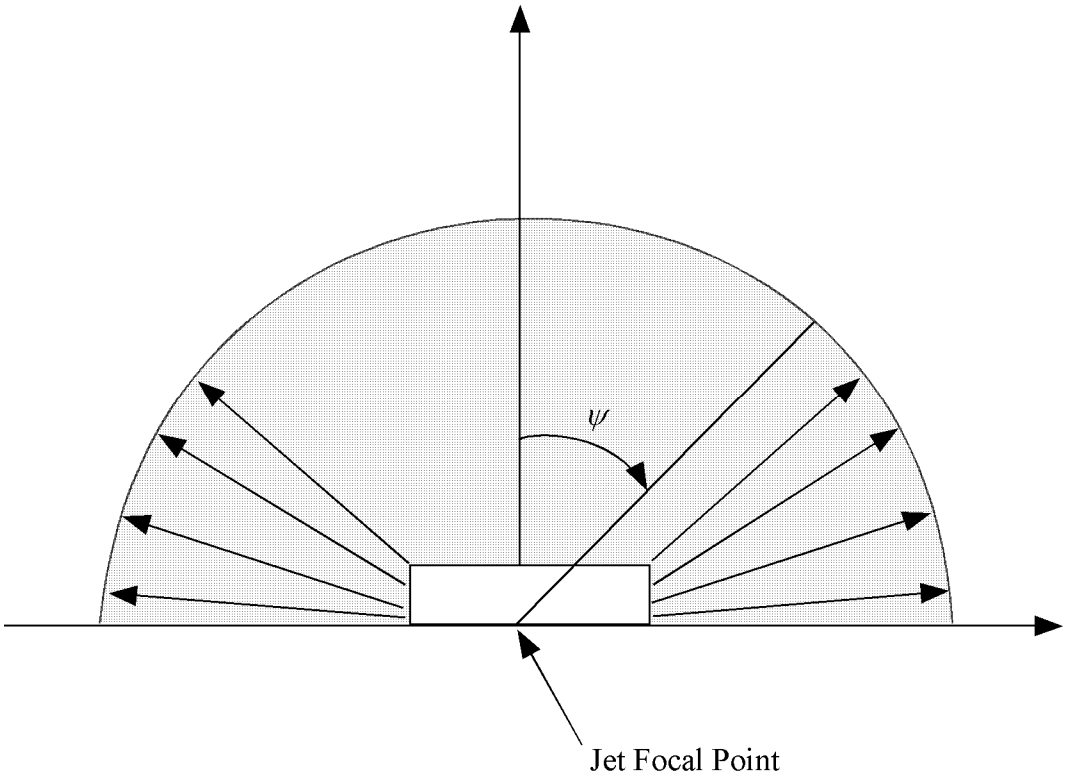


Figure 1.2 Multiple jet model for driver's side airbag.

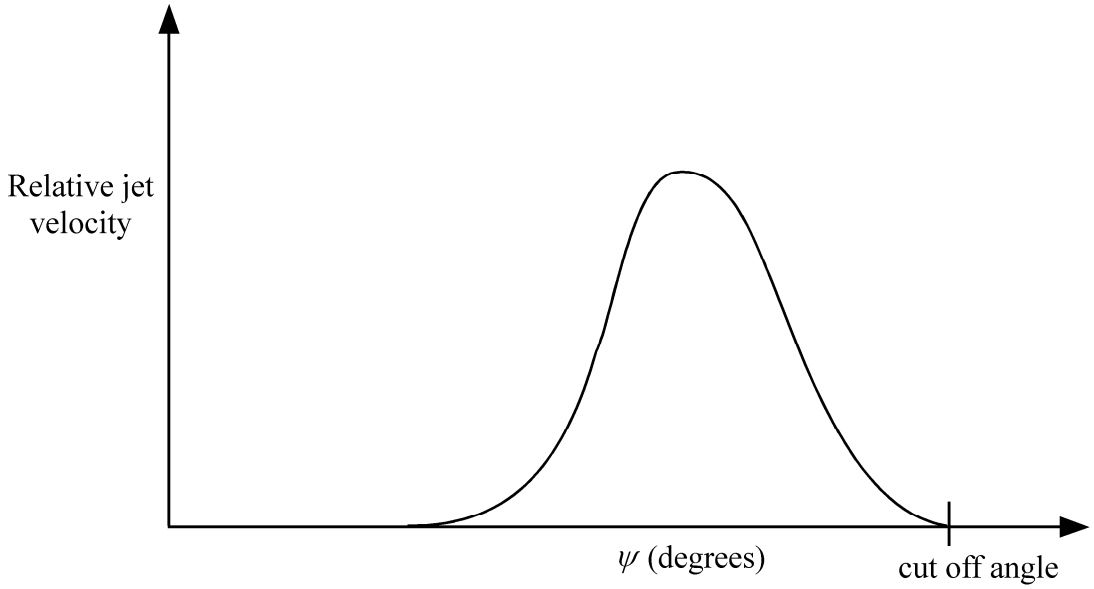


Figure 1.3 Normalized jet velocity versus angle for multiple jet driver's side airbag.

Further additional required for CM option.

The following additional card is defined for the WANG_NEFSKE_JETTING_CM and WANG_ NEFSKE_MULTIPLE_JETTING_CM options.

Additional card required for _CM option

Card 1

	1	2	3	4	5	6	7	8
Variable	NREACT							
Type	I							
Default	None							
Remark								

VARIABLE	DESCRIPTION
NREACT	Node for reacting jet force. If zero the jet force will not be applied.

Remarks:

Compared with the standard LS-DYNA jetting formulation, the Constant Momentum option has several differences. Overall, the jetting usually has a more significant effect on airbag deployment than the standard LS-DYNA jetting: the total force is often greater, and does not reduce with distance from the jet.

The velocity at the jet outlet is assumed to be a choked (sonic) adiabatic flow of a perfect gas. Therefore the velocity at the outlet is given by:

$$v_{outlet} = \sqrt{(\gamma RT)} = \sqrt{\left(\frac{(c_p - c_v) T c_p}{c_v} \right)}$$

The density in the nozzle is then calculated from conservation of mass flow.

$$\rho_0 v_{outlet} A_{outlet} = \dot{m}$$

This is different from the standard LS-DYNA jetting formulation, which assumes that the density of the gas in the jet is the same as atmospheric air, and then calculates the jet velocity from conservation of mass flow.

The velocity distribution at any radius, r, from the jet centerline and distance, z, from the focus, v_{rz}, relates to the velocity of the jet centerline, $v_r = 0, z$, in the same way as the standard LS-DYNA jetting options:

$$v_{r,z} = v_{r=0,z} e^{-\left(\frac{r}{\alpha z}\right)_2}$$

The velocity at the jet centerline, $v_r = 0$, at the distance, z, from the focus of the jet is calculated such that the momentum in the jet is conserved.

Momentum at nozzle = Momentum at z

$$\begin{aligned} \rho_0 v_{outlet}^2 A_{outlet} &= \rho_0 \int v_{jet}^2 dA_{jet} \\ &= \rho_0 v_{r=0,Z}^2 \left\{ b + F\sqrt{b} \right\} \end{aligned}$$

where

$$b = \frac{\pi(\alpha z)^2}{2}$$

$$F = distance\ between\ jet\ focii\ for\ a\ passenger\ jet$$

Finally, the pressure exerted on an airbag element in view of the jet is given as:

$$p_{r,z} = \beta \rho_0 v_{r,z}^2$$

By combining the equations above

$$p_{r,z} = \frac{\beta \dot{m} v_{outlet} \left[e^{-(r/\alpha z)^2} \right]^2}{\left\{ \frac{\pi(\alpha z)^2}{2} + F\sqrt{\frac{\pi(\alpha z)^2}{2}} \right\}}$$

The total force exerted by the jet is given by:

$$F_{jet} = \dot{m} v_{outlet} \quad \text{(independent of distance from the nozzle)}$$

Mass flow in the jet is not necessarily conserved, because gas is entrained into the jet from the surrounding volume. By contrast, the standard LS-DYNA jetting formulation conserves mass flow but not momentum. This has the effect of making the jet force reduce with distance from the nozzle.

The jetting forces can be reacted onto a node (NREACT), to allow the reaction force through the steering column or the support brackets to be modeled. The jetting force is written to the ASCII ABSTAT file and the binary XTF file.

Additional card required for LOAD_CURVE option

Card	1	2	3	4	5	6	7	8
Variable	STIME	LCID	RO	PE	P0	T	T0	
Type	F	I	F	F	F	F	F	
Default	0.0	none	none	none	none	none	none	

VARIABLE	DESCRIPTION
STIME	Time at which pressure is applied. The load curve is offset by this amount.
LCID	Load curve ID defining pressure versus time, see *DEFINE_CURVE.
RO	Initial density of gas (ignored if LCID > 0)
PE	Ambient pressure (ignored if LCID > 0)
P0	Initial gauge pressure (ignored if LCID > 0)
T	Gas Temperature (ignored if LCID > 0)
T0	Absolute zero on temperature scale (ignored if LCID > 0)

Remarks:

Within this simple model the control volume is inflated with a pressure defined as a function of time or calculated using the following equation if LCID = 0.

$$P_{total} = C\rho(T - T_0)$$
$$P_{gauge} = P_{total} - P_{ambient}$$

The pressure is uniform throughout the control volume.

Additional card required for LINEAR_FLUID option

Card	1	2	3	4	5	6	7	8
Variable	BULK	RO	LCINT	LCOUTT	LCOUTP	LCFIT	LCBULK	LCID
Type	F	F	I	I	I	I	I	I
Default	none	none	none	optional	optional	optional	optional	none

If the next card is a "*" keyword card, the following card is not read.

Variable	P_LIMIT	P_LIMLC						
Type	F	I						
Default	optional	optional						

VARIABLE	DESCRIPTION
BULK	K, bulk modulus of the fluid in the control volume. Constant as a function of time. Define if LCBULK=0.
RO	ρ, density of the fluid
LCINT	$F(t)$ input flow curve defining mass per unit time as a function of time, see *DEFINE_CURVE.
LCOUTT	$G(t)$, output flow curve defining mass per unit time as a function of time. This load curve is optional.
LCOUTP	$H(p)$, output flow curve defining mass per unit time as a function of pressure. This load curve is optional.
LFIT	$L(t)$, added pressure as a function of time. This load curve is optional.
LCBULK	Curve defining the bulk modulus as a function of time. This load curve is optional, but if defined, the constant, BULK, is not used.
LCID	Load curve ID defining pressure versus time, see *DEFINE_CURVE.
P_LIMIT	Limiting value on total pressure (optional).

VARIABLE	DESCRIPTION
P_LIMLC	Curve defining the limiting pressure value as a function of time. If nonzero, P_LIMIT is ignored.

Remarks:

If LCID = 0 then the pressure is determined from:

$$P(t) = K(t)\ln\left(\frac{V_0(t)}{V(t)}\right) + L(t)$$

where

$P(t)$ Pressure,

$V(t)$ Volume of fluid in compressed state,

$V_0(t) = V_0(t) = \dfrac{M(t)}{\rho}$ Volume of fluid in uncompressed state,

$M(t) = M(0) + \int F(t)dt - \int G(t)dt - \int H(p)dt$ Current fluid mass,

$M(0) = V(0)\rho$ Mass of fluid at time zero $P(0) = 0$.

By setting LCID ≠ 0 a pressure time history may be specified for the control volume and the mass of fluid within the volume is then calculated from the volume and density.

This model is for the simulation of hydroforming processes or similar problems. The pressure is controlled by the mass flowing into the volume and by the current volume. The pressure is uniformly applied to the control volume.

Note the signs used in the equation for $M(t)$. The mass flow should always be defined as positive since the output flow is subtracted.

Additional cards required for HYBRID and HYBRID_JETTING options

Card 1 2 3 4 5 6 7 8

Variable	ATMOST	ATMOSP	ATMOSD	GC	CC	HCONV		
Type	F	F	F	F	F	F		
Default	none	none	none	none	1.0	none		

Card

Variable	C23	LCC23	A23	LCA23	CP23	LCP23	AP23	LCAP23
Type	F	I	F	I	F	I	F	I
Default	none	0	none	0	none	0	none	0

Card

Variable	OPT	PVENT	NGAS	LCEFR	LCIDM0			
Type	I	F	I	I	I			
Default	none	none	none	0	0			

Define 2*NGAS cards below, two for each gas type.

Card	1	2	3	4	5	6	7	8
Variable	LCIDM	LCIDT		MW	INITM	A	B	C
Type	I	I	F	F	F	F	F	F
Default	none	none	not used	none	none	none	none	none

Card

Variable	FMASS							
Type	F							
Default	none							

VARIABLE	DESCRIPTION
ATMOST	Atmospheric temperature
ATMOSP	Atmospheric pressure
ATMOSD	Atmospheric density
GC	Universal molar gas constant
CC	Conversion constant EQ: .0 Set to 1.0.
HCONV	Effective heat transfer coefficient between the gas in the air bag and the environment at temperature at ATMOST. If HCONV<0, then HCONV defines a load curve of data pairs (time, hconv).
C23	Vent orifice coefficient which applies to exit hole. Set to zero if LCC23 is defined below.
LCC23	The absolute value, |LCC23|, is a load curve ID. If the ID is positive, the load curve defines the vent orifice coefficient which applies to exit hole as a function of time. If the ID is negative, the vent orifice coefficient is

defined as a function of relative pressure, P_{air} / P_{bag}, see [Anagonye and Wang 1999]. A nonzero value for C23 overrides LCC23.

VARIABLE	DESCRIPTION
A23	If defined as a positive number, A23 is the vent orifice area which applies to exit hole. If defined as a negative number, the absolute value |A23| is a part ID, see [Anagonye and Wang 1999]. The area of this part becomes the vent orifice area. Set A23 to zero if LCA23 is defined below.
LCA23	Load curve number defining the vent orifice area which applies to exit hole as a function of absolute pressure. A nonzero value for A23 overrides LCA23.
CP23	Orifice coefficient for leakage (fabric porosity). Set to zero if LCCP23 is defined below.
LCCP23	Load curve number defining the orifice coefficient for leakage (fabric porosity) as a function of time. A nonzero value for CP23 overrides LCCP23.
AP23	Area for leakage (fabric porosity)
LCAP23	Load curve number defining the area for leakage (fabric porosity) as a function of (absolute) pressure. A nonzero value for AP23 overrides LCAP23.
OPT	Fabric venting option, if nonzero CP23, LCCP23, AP23, and LCAP23 are set to zero. EQ. 1: Wang-Nefske formulas for venting through an orifice are used. Blockage is not considered. EQ. 2: Wang-Nefske formulas for venting through an orifice are used. Blockage of venting area due to contact is considered. EQ. 3: Leakage formulas of Graefe, Krummheuer, and Siejak [1990] are used. Blockage is not considered. EQ. 4: Leakage formulas of Graefe, Krummheuer, and Siejak [1990] are used. Blockage of venting area due to contact is considered. EQ. 5: Leakage formulas based on flow through a porous media are used. Blockage due to contact is not considered. EQ. 6: Leakage formulas based on flow through a porous media are used. Blockage due to contact is considered. EQ. 7: Leakage is based on gas volume outflow versus pressure load curve. Blockage of flow area due to contact is not considered. Absolute pressure is used in the porous-velocity-versus-pressure load curve, given as FAC(P) in the *MAT_FABRIC card. EQ. 8: Leakage is based on gas volume outflow versus pressure load curve. Blockage of flow area due to contact is considered. Absolute pressure is used in the porous-velocity-versus-pressure load curve, given as FAC(P) in the *MAT_FABRIC card.

VARIABLE	DESCRIPTION
PVENT	Gauge pressure when venting begins
NGAS	Number of gas inputs to be defined below (Including initial air). The maximum number of gases is 17.
LCEFR	Optional curve for exit flow rate (mass/time) versus (gauge) pressure
LCIDM0	Optional curve representing inflator's total mass inflow rate. When defined, LCIDM in the following 2xNGAS cards defines the molar fraction of each gas component as a function of time and INITM defines the initial molar ratio of each gas component .
LCIDM	Load curve ID for inflator mass flow rate (eq. 0 for gas in the bag at time=0) GT.0: piece wise linear interpolation LT.0: cubic spline interpolation
LCIDT	Load curve ID for inflator gas temperature (eq.0 for gas in the bag at time 0) GT.0: piece wise linear interpolation LT.0: cubic spline interpolation
BLANK	(not used)
MW	Molecular weight
INITM	Initial mass fraction of gas component
A	Coefficient for molar heat capacity of inflator gas at constant pressure, (e.g., Joules/mole/^{o}K)
B	Coefficient for molar heat capacity of inflator gas at constant pressure, (e.g., Joules/mole/^{o}K^{2})
C	Coefficient for molar heat capacity of inflator gas at constant pressure, (e.g., Joules/mole/^{o}K^{3})
FMASS	Fraction of additional aspirated mass.

Further additional cards are required for HYBRID_JETTING and …_CM models

The following two additional cards are defined for the HYBRID_JETTING options. The jet may be defined by specifying either the coordinates of the jet focal point, jet vector head and secondary jet focal point, or by specifying three nodes located at these positions. The nodal point option is recommended when the location of the airbag changes as a function of time.

Card 1

1	2	3	4	5	6	7	8
Variable	XJFP	YJFP	ZJFP	XJVH	YJVH	ZJVH	CA	BETA
Type	F	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	none	none
Remark	1	1	1	1	1	1		

Card 2

Variable	XSJFP	YSJFP	ZSJFP	PSID	IDUM	NODE1	NODE2	NODE3
Type	F	F	F	I	F	I	I	I
Default	none	none	none	none	none	0	0	0
Remark					2	1	1	1

Additional card required for HYBRID_JETTING_CM option

Card 1

1	2	3	4	5	6	7	8
Variable	NREACT							
Type	I							
Default	None							
Remark	4							

VARIABLE	DESCRIPTION
XJFP	x-coordinate of jet focal point, i.e., the virtual origin in Figure 1.1. See Remark 1 below.
YJFP	y-coordinate of jet focal point, i.e., the virtual origin in Figure 1.1.
ZJFP	z-coordinate of jet focal point, i.e., the virtual origin in Figure 1.1.
XJVH	x-coordinate of jet vector head to defined code centerline
YJVH	y-coordinate of jet vector head to defined code centerline
ZJVH	z-coordinate of jet vector head to defined code centerline
CA	Cone angle, α, defined in radians. LT.0.0: $\lvert\alpha\rvert$ is the load curve ID defining cone angle as a function of *time*
BETA	Efficiency factor, β, which scales the final value of pressure obtained from Bernoulli's equation. LT.0.0: $\lvert\beta\rvert$ is the load curve ID defining the efficiency factor as a function of *time*
XSJFP	x-coordinate of secondary jet focal point, passenger side bag. If the coordinate of the secondary point is (0,0,0) then a conical jet (driver's side airbag) is assumed.
YSJFP	y-coordinate of secondary jet focal point
ZSJFP	z-coordinate of secondary jet focal point
PSID	Optional part set ID, see *SET_PART. If zero all elements are included in the airbag.
IDUM	Dummy field (Variable not used)
NODE1	Node ID located at the jet focal point, i.e., the virtual origin in Figure 1.1. See Remark 1 below.
NODE2	Node ID for node along the axis of the jet.
NODE3	Optional node ID located at secondary jet focal point.
NREACT	Node for reacting jet force. If zero the jet force will not be applied.

Remarks:

1. It is assumed that the jet direction is defined by the coordinate method (XJFP, YJFP, ZJFP) and (XJVH, YJVH, ZJVH) unless both NODE1 and NODE2 are defined. In which case the coordinates of the nodes given by NODE1, NODE2 and NODE3 will override (XJFP, YJFP, ZJFP) and (XJVH, YJVH, ZJVH). The use of nodes is recommended if the airbag system is undergoing rigid body motion. The nodes should be attached to the vehicle to allow for the coordinates of the jet to be continuously updated with the motion of the vehicle.

   The jetting option provides a simple model to simulate the real pressure distribution in the airbag during the breakout and early unfolding phase. Only the surfaces that are in the line of sight to the virtual origin have an increased pressure applied. With the optional load curve LCRJV, the pressure distribution with the code can be scaled according to the so-called relative jet velocity distribution.

   For passenger side airbags the cone is replaced by a wedge type shape. The first and secondary jet focal points define the corners of the wedge and the angle α then defines the wedge angle.

   Instead of applying pressure to all surfaces in the line of sight of the virtual origin(s), a part set can be defined to which the pressure is applied.

2. This variable is not used and has been included to maintain the same format as the WANG_NEFSKE_JETTING options.

3. Care must be used to place the jet focal point within the bag. If the focal point is outside the bag, inside surfaces will not be visible so jetting pressure will not be applied correctly.

4. See the description related to the WANG_NEFSKE_JETTING_CM option. For the hybrid inflator model the heat capacities are compute from the combination of gases which inflate the bag.

Additional cards required for HYBRID_CHEMKIN model

The HYBRID_CHEMKIN model includes 3 control cards. For each gas species an additional set of cards must follow consisting of a control card and several thermodynamic property data cards.

Card 1

	1	2	3	4	5	6	7	8
Variable	LCIDM	LCIDT	NGAS	DATA	ATMT	ATMP	RG	
Type	I	I	I	I	F	F	F	
Default	none	none	none	none	none	none	none	

Card 2

Variable	HCONV							
Type	F							
Default	0.							

Card 3

Variable	C23	A23						
Type	F	F						
Default	0.	0.						

VARIABLE	DESCRIPTION
LCIDM	Load curve specifying input mass flow rate versus time. GT.0: piece wise linear interpolation LT.0: cubic spline interpolation
LCIDT	Load curve specifying input gas temperature versus time. GT.0: piece wise linear interpolation LT.0: cubic spline interpolation

VARIABLE	DESCRIPTION
NGAS	Number of gas inputs to be defined below. (Including initial air)
DATA	Thermodynamic database EQ.1. NIST database (3 additional property cards are required below) EQ.2. CHEMKIN database (no additional property cards are required) EQ.3. Polynomial data (1 additional property card is required below)
ATMT	Atmospheric temperature.
ATMP	Atmospheric pressure
RG	Universal gas constant
HCONV	Effective heat transfer coefficient between the gas in the air bag and the environment at temperature ATMT. If HCONV<0, then HCONV defines a load curve of data pairs (time, hconv).
C23	Vent orifice coefficient
A23	Vent orifice area

For each gas species include a set of cards consisting of a control card followed by several thermo-dynamic property data cards. The next "*" card terminates the reading of this data.

Control Card

Card 1	1	2	3	4	5	6	7	8
Variable	CHNAME	MW	LCIDN	FMOLE	FMOLET			
Type	A	F	I	F	F			
Default	none	none	0	none	0.			

VARIABLE	DESCRIPTION
CHNAME	Chemical symbol for this gas species (e.g., N2 for nitrogen, AR for argon). Required for DATA=2 (CHEMKIN), optional for DATA=1 or DATA=3.
MW	Molecular weight of this gas species.

LCIDN	Load curve specifying the input mole fraction versus time for this gas species. If >0, FMOLE is not used.

VARIABLE	DESCRIPTION
FMOLE	Mole fraction of this gas species in the inlet stream.
FMOLET	Initial mole fraction of this gas species in the tank.

Additional thermodynamic data cards for each gas species. No additional cards are needed if using the CHEMKIN database (DATA=2).

If DATA=1, include the following 3 cards for the NIST database. The required data can be found on the NIST web site at http://webbook.nist.gov/chemistry/

Card 1

Variable	1	2	3	4	5	6	7	8
Variable	TLOW	TMID	THIGH					
Type	F	A8	F					
Default	none	none	none					

Card 2

Variable	a low	b low	c low	d low	e low	f low	h low	
Type	F	F	F	F	F	F	F	
Default	none	none	none	none	none	none	none	

Card 3

Variable	a high	b high	c high	d high	e high	f high	h high	
Type	F	F	F	F	F	F	F	
Default	none	none	none	none	none	none	none	

VARIABLE	DESCRIPTION
TLOW	Curve fit low temperature limit.
TMID	Curve fit low-to-high transition temperature.
THIGH	Curve fit high temperature limit.
$a_{low}, \ldots, h_{low}$	Low temperature range NIST polynomial curve fit coefficients (see below).
$a_{high}, \ldots, h_{high}$	High temperature range NIST polynomial curve fit coefficients (see below).

If DATA=3, include the following card for the polynomial curve fit.

Card 1

1	2	3	4	5	6	7	8
Variable	a	b	c	d	e			
Type	F	F	F	F	F			
Default	none	0.	0.	0.	0.			

VARIABLE	DESCRIPTION
a	Coefficient, see below.
b	Coefficient, see below.
c	Coefficient, see below.
d	Coefficient, see below.
e	Coefficient, see below.

Heat capacity curve fits:

NIST $$c_p = \frac{1}{M}\left(a + bT + cT^2 + dT^3 + \frac{e}{T^2}\right)$$

CHEMKIN $$c_p = \frac{\bar{R}}{M}\left(a + bT + cT^2 + dT^3 + eT^4\right)$$

$\bar{R}$ = universal gas constant (8.314 Nm / mole K)
M = gas molecular weight

Polynomial $$c_p = \frac{1}{M}\left(a + bT + cT^2 + dT^3 + eT^4\right)$$

*AIRBAG_ADVANCED_ALE

Purpose: This *AIRBAG_ADVANCED_ALE (AAA) card provides an alternate, simplified approach to simulating airbag deployment with ALE capabilities. Its input is translated into ALE keywords internally inside LS-DYNA (please review an output file called "advalebag.kw" which contains all the translated ALE commands). It has an option to start the simulation with the ALE method then switch to the control volume (CV) method at a chosen switch time. The CV input data may be used directly. A basic Lagrangian airbag shell structure may consist of an inflator compartment, an airbag, and possibly vent holes. The Lagrangian airbag interacts with the ALE inflator gas. The AAA card requires, at the minimum, one or more *DEFINE_ALEBAG_INFLATOR cards (one for each inflator), and one or more *DEFINE_ALEBAG_BAG cards (one for each bag structure). In addition, one optional card, *DEFINE_ALEBAG_HOLE, may be used to define each physical vent hole on the airbag. Multiple fluid-to-structure interactions may be simulated using multiple AAA definitions. However, more complex couplings most likely must be modeled via the traditional ALE approach (remark 1)..

Card 1 Format

Card 1

Variable	BAGID1	BAGID2	BAGID3	BAGID4	BAGID5	BAGID6	BAGID7	BAGID8
Type	I	I	I	I	I	I	I	I
Default	none	none	none	none	none	none	none	none
Remarks								

Card 2

Variable	HOLEID1	HOLEID2	HOLEID3	HOLEID4	HOLEID5	HOLEID6	HOLEID7	HOLEID8
Type	I	I	I	I	I	I	I	I
Default	none	none	none	none	none	none	none	none
Remarks								

Card 3	1	2	3	4	5	6	7	8
Variable	INFLID1	INFLID2	INFLID3	INFLID4	INFLID5	INFLID6	INFLID7	INFLID8
Type	I	I	I	I	I	I	I	I
Default	none	none	none	none	none	none	none	none
Remarks	8							

Card 4 Format (Define information related to the automatic ALE mesh generation)

Card 4	1	2	3	4	5	6	7	8
Variable	NX/IDA	NY	NZ	unused	ARSNID	IDCENT	EXSID	
Type	I	I	I		I	I	I	
Default	none	none	none		none	none	none	
Remarks	4	4	4					

Card 4b (optional, only define if NX & NY & NZ are nonzero, for automatic ALE mesh generation)

Card 4	1	2	3	4	5	6	7	8
Variable	LX	LY	LZ	ITRANS	UIDAIR			
Type	F	F	F	I	I			
Default	none	none	none					
Remarks								

Card 5: Parameters for defining ambient environment CV *AIRBAG_PARAMETERS

Card 5	1	2	3	4	5	6	7	8
Variable	ATMOST	ATMOSP	unused	GC	CC	unused	MWD	SPSF
Type	F	F		F	F		F	F
Default	0.	0.		none	1.0		0.0	0.0
Remarks	5	5						

Card 6: Parameters for defining ambient environment CV *AIRBAG_PARAMETERS

Card 6	1	2	3	4	5	6	7	8
Variable	SWTIME	unused	HG	NAIR				
Type	F		F	I				
Default	0.		0.	0				
Remarks	6							

Card 7+ (Define air: repeat this card "NAIR" times. NAIR is defined in Card 6)

Card	1	2	3	4	5	6	7	8
Variable	unused	unused	unused	MWAIR	INITM	AIRA	AIRB	AIRC
Type				F	F	F	F	F
Default				0	0	0	0.	0.
Remarks						2		

VARIABLE	DESCRIPTION
BAGID#	Each BAGID refers to an airbag definition defined via a corresponding *DEFINE_ALEBAG_BAG (DAB) card. Each DAB card defines (a) one Lagrangian shell structure representing an airbag (or part of one), (b) its venting characteristics, and (c) its coupling behavior..
HOLEID#	Each HOLEID refers to a vent hole definition defined via a corresponding *DEFINE_ALEBAG_HOLE (DAH) card. Each DAH defines the information required for modeling a physical venting hole including the switching of the ALE multi-material group (AMMG) ID when a gas passes through this vent hole. The information in DAH is translated into a simplified form of the *ALE_FSI_SWITCH_MMG_ID (AFSM) card.
INFLID#	Each INFLID refers to one inflator definition defined via a corresponding *DEFINE_ALEBAG_INFLATOR (DAI) card. Each DAI defines the information required for the modeling of a physical inflator (which pumps a gas mixture in to inflate the airbag). INFLID# > 0: INFLID is associated with an inflator definition defined via a corresponding DAI card
NX/IDA	Option 1: NX is defined as the number of ALE elements to be generated by LS-DYNA automatically in the x direction. If option 1 is chosen, option 1 must also be used for NY and NZ, and card 4B must also be defined. Option 2: IDA is the Part ID of the initial background air mesh (remarks 4 and 9). If IDA is a negative integer, the PID of the air mesh is the same as the one used in a previous AAA card. This approach is only used in the cases of (a) bag-in-bag or (b) bag-to-bag model where there may be more than one AAA card defined.
NY	Option 1: NY is defined as the number of ALE elements to be generated by LS-DYNA automatically in the y direction. If option 1 is chosen, option 1 must also be used for NX and NZ, and card 4B must also be defined. Option 2: Leave blank or 0 (remark 4).
NZ	Option 1: NZ is defined as the number of ALE elements to be generated by LS-DYNA automatically in the z direction. If option 1 is chosen, option 1 must also be used for NX and NY, and card 4B must also be defined Option 2: Leave blank or 0 (remark 4).

VARIABLE	DESCRIPTION
ARSNID	An ID associated with an *ALE_REFERENCE_SYSTEM_NODE card defining 3 nodes making up a local coordinate system. The ALE mesh system will automatically follow this local coordinate system. If there is mesh expansion, and if IDCENT > 0, the expansion will occur along these local axes. Else if IDCENT is not defined (0), the expansion will occur along the global coordinate axes.
IDCENT	A Lagrangian node ID defines the center of expansion for the ALE mesh system. For example, this node may be located on a rigid section of a steering wheel, near the inflator region. The ALE mesh can then move with the steering wheel as it expands. This provides a similar expansion mechanism the *ALE_REFERENCE_SYSTEM_GROUP command (PRTYPE=9).
EXSID	A part ID which makes up a portion of the overall the ALE mesh system to be excluded from expansion. This part can still move with the rest of the ALE system, but will not expand with the rest. This may be used to exclude the inner ALE mesh region surrounding the inflator (which is typically finer resolution to better resolve the inflator gas inflow at the orifices).
LX	Absolute length in the global X-coordinate of the ALE mesh
LY	Absolute length in the global Y-coordinate of the ALE mesh
LZ	Absolute length in the global Z-coordinate of the ALE mesh
ITRANS	An integer ID pointing an ID of a *DEFINE_TRANSFORMATION card which may be used to position the automatically generated ALE mesh at its chosen location and orientation. The mesh is first generated with respect to the origin of the global coordinate system. Then it is moved to the t=0 position via this *DEFINE_TRANSFORMATION card.
UIDAIR	User-defined PID number for the background ALE air mesh. If this is left blank, LS-DYNA will assign a PID for this automatically generated ALE mesh. This may be used to actively define the PID number for the ALE mesh ATMOST Atmospheric ambient temperature (See Remark 5).
ATMOST	Atmospheric ambient temperature (See Remark 5).
ATMOSP	Atmospheric ambient pressure (See Remark 5).
GC	Universal molar gas constant.
CC	Conversion constant. EQ: 0.0 Set to 1.0.

VARIABLE	DESCRIPTION
MWD	Mass weighted damping factor, D
SPSF	Stagnation pressure scale factor.
SWTIME	Time to switch from ALE method to control volume (CV) method. Once switched, the airbag inflation is simulated by a control volume method similar to that used by the *AIRBAG_HYBRID card (See Remark 6). EQ.0.0: switch to CV method at time = 0.0. EQ.blank: switch time is set at 1.0E16 (ALE method is used). EQ.t: switch from ALE to CV method at time = t.
HG	Hourglass coefficient for ALE fluid mesh(es). A typical value on the order of 1.0E-5 or 1.0E-6 may be used for gases and liquids.
NAIR	Number of species for air. If air is defined as 1 single gas then NAIR=1. Card 7 is repeated "NAIR" number of times, one for each species. For example, NAIR=2 for a 2-component air model containing 80% of N2 and 20% of O2. Then card 7 is defined twice.
MWAIR	Molecular weight of this air component
INITMFA	Initial Mass Fraction of this air component
AIRA	First Coefficient of molar heat capacity at constant pressure (e.g., J/mole/K, See Remark 2).
AIRB	Second Coefficient of molar heat capacity at constant pressure (e.g., J/mole/K^2).
AIRC	Third Coefficient of molar heat capacity at constant pressure (e.g., J/mole/K^3).

Remarks:

1. This card defines information for simulating an airbag deployment. It defines:
   - The IDs of the bags, vent holes, and inflators;
   - Information about the ALE background (air) mesh;
   - Basic ambient condition (similar to that from *AIRBAG_HYBRID);
   - Switch time (time for switching from ALE to CV method);
   - Material properties of air and some other miscellaneous information.

   As the input to this command, *AIRBAG_ADVANCED_ALE, is simply translated into the traditional ALE keywords, this command will not have all the flexibilities that the general ALE method allows. It does not provide a grammatical framework for understanding the modeling of fluid structure interaction. It is recommended that the users familiarize themselves with the traditional method of modeling airbag deployment using the ALE method. This is critical in understanding the interactions between the ALE inflator gas and the Lagrangian airbag structure.

2. The per-mass-unit, temperature-dependent, constant-pressure heat capacity is

$$C_p(T) = \frac{[A + B*T + C*T^2]}{MW} \sim \frac{J}{kg*K} \qquad B \sim J/(mole*K^2)$$

$$A = \tilde{C}_{P0} \sim J/(mole*K) \qquad C \sim J/(mole*K^3)$$

   The units shown are only for demonstration of the equation.

3. In general, it is best to locate a point source near the center of an ALE element. Associated width each point source is an area and a vector indicating flow direction. Each point source should occupy 1 ALE element by itself, and there should be at least 2 empty ALE elements between any 2 point sources. A point source should be located at least 3 elements away from the free surface of an ALE mesh for stability (see ***SECTION_POINT_SOURCE_MIXTURE** card)

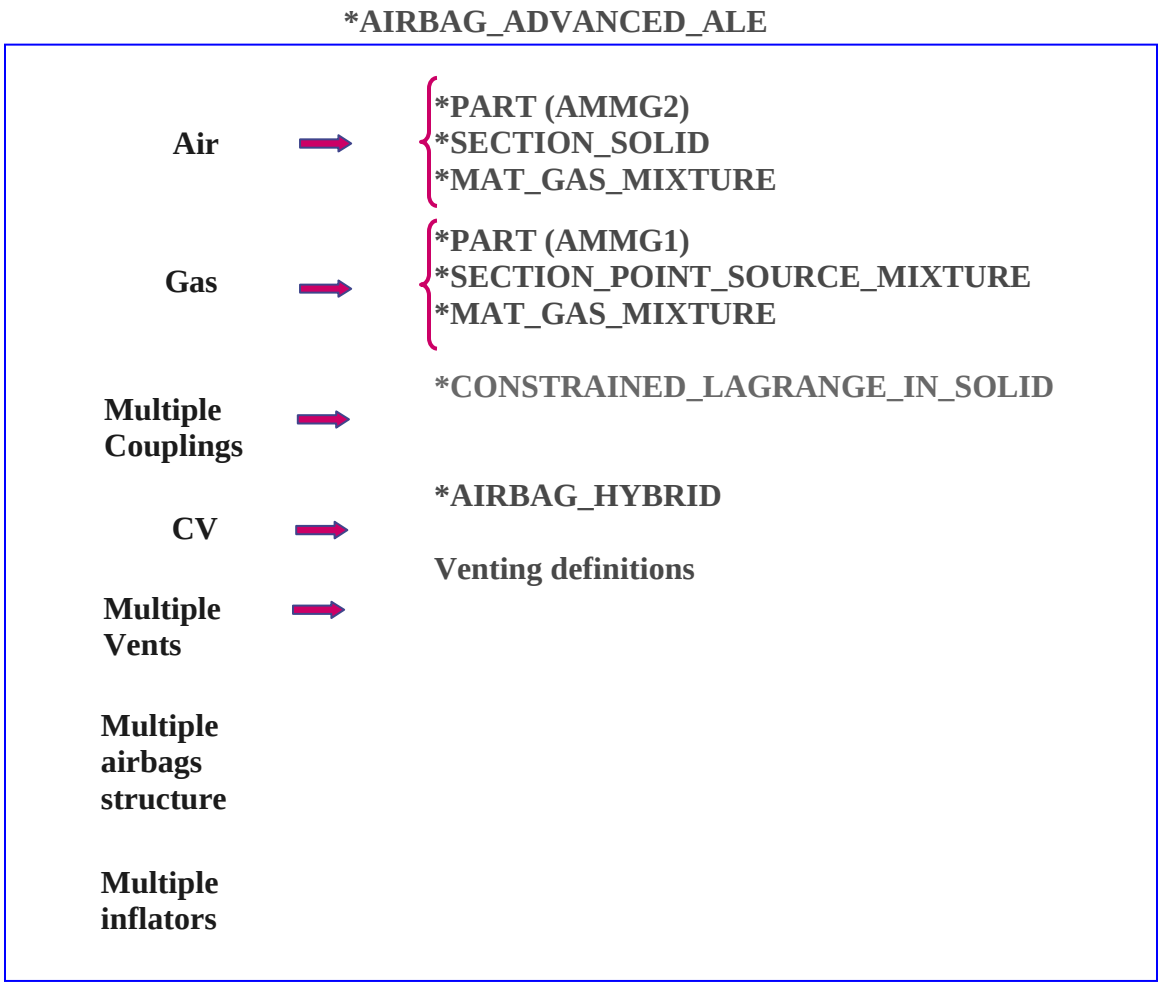
4. There are 2 options for defining the ALE mesh for an airbag deployment simulation. Option 1 lets LS-DYNA generate the background ALE mesh. NX, NY and NZ must be defined. Card 4b must also be defined. Option 2 uses an existing ALE mesh for the background air. The PID of this ALE mesh is defined via IDA (same as the NX field). In this case, the background ALE air mesh should envelope the airbag deployment space. There is no need to define card 4b. If IDA is a negative integer, the PID for the air mesh is the same as the one used in another AAA card (multiple AAA cards implied).

5. Atmospheric density for the ambient gas (air) can be computed from $\rho_{amb} = P_{amb}/(R*T_{amb})$. This card is similar to the 1st additional card for the *AIRBAG_HYBRID command with 2 additional parameters (MWD & SPSF) from the 1st card of the *AIRBAG_ command.

6. Since ALL ALE related activities will be turned off after the switch from ALE method to control-volume method, no other ALE coupling will exist beyond t=SWTIME. Using the traditional ALE modeling method, this switching can also be modeled via the command

*ALE_UP_SWITCH. The user should review its usage as it can provide extensively more coupling controls for general airbag modeling.

7. The *AIRBAG_ADVANCED_ALE (AAA) card may be considered an alternate approach for inputting the information required by the traditional ALE card:



*AIRBAG_ALE

Purpose: The input in this section provides a simplified approach to defining the deployment of the airbag using the ALE capabilities with an option to switch from the initial ALE method to control volume (CV) method (*AIRBAG_HYBRID) at a chosen time. An enclosed airbag (and possibly the airbag canister/compartment and/or a simple representation of the inflator) shell structure interacts with the inflator gas(es). This definition provides a single fluid to structure coupling for the airbag-gas interaction during deployment in which the CV input data may be used directly.

Card 1

1	2	3	4	5	6	7	8
Variable	SID	SIDTYP					MWD	SPSF
Type	I	I					F	F
Default	none	none					0	0
Remarks	1							

VARIABLE	DESCRIPTION
SID	Set ID as defined on *AIRBAG card. This set ID contains the Lagrangian elements (segments) which make up the airbag and possibly the airbag canister/compartment and/or a simple representation of the inflator. See Remark 1.
SIDTYP	Set type: EQ.0: Segment set. EQ.1: Part set.
MWD	Mass weighted damping factor, D. This is used during the CV phase for *AIRBAG_HYBRID.
SPSF	Stagnation pressure scale factor, $0 \le \gamma \le 1$. This is used during the CV phase for *AIRBAG_HYBRID.

Parameters for defining ambient environment.

Card 2	1	2	3	4	5	6	7	8
Variable	ATMOST	ATMOSP		GC	CC	TNKVOL	TNKFINP	
Type	F	F		F	F	F	F	
Default	0.	0.		none	1.0	0.0	0.0	
Remarks	2	2				10	10	

VARIABLE	DESCRIPTION
ATMOST	Atmospheric ambient temperature. See Remark 2.
ATMOSP	Atmospheric ambient pressure. See Remark 2.
GC	Universal molar gas constant.
CC	Conversion constant. If EQ: .0 Set to 1.0.
TNKVOL	Tank volume from the inflator tank test – or – Inflator canister volume. See remark 10. Option 1: (LCVEL = 0) This is defined as Tank volume (must also define TNKFINP). Inlet gas velocity is estimated by LS-DYNA method (testing). Option 2: (LCVEL = 0) This is defined as estimated inflator canister volume (must NOT define TNKFINP). Inlet gas velocity is estimated automatically by the Lian-Bhalsod-Olovsson method. Option 3: (LCVEL .NE. 0) This must be left blank.
TNKFINP	Tank final pressure from the inflator tank test data. Only define this parameter for option 1 of TNKVOL definition above. See Remark 10.

Parameters for coupling, see keyword *CONSTRAINED_LAGRANGE_IN_SOLID.

Card 3	1	2	3	4	5	6	7	8
Variable	NQUAD	CTYPE	PFAC	FRIC	FRCMIN	NORMTYP	ILEAK	PLEAK
Type	I	I	F	F	F	I	I	F
Default	4	4	0.1	0.0	0.3	0	2	0.1
Remarks	13	13	14					

VARIABLE	DESCRIPTION
NQUAD	Number of (quadrature) coupling points for coupling Lagrangian slave parts to ALE master solid parts. If NQUAD=n, then nXn coupling points will be parametrically distributed over the surface of each Lagrangian slave segment (default=4). See Remark 13.
CTYPE	Coupling type (default=4, see Remark 13): EQ.4: (default) penalty coupling with DIREC=2 implied. EQ.6: penalty coupling in which DIREC is automatically set to DIREC=1 for the unfolded region and DIREC=2 for folded region.
PFAC	Penalty factor. PFAC is a scale factor for scaling the estimated stiffness of the interacting (coupling) system. It is used to compute the coupling forces to be distributed on the slave and master parts. If positive real: Fraction of estimated critical stiffness default=0.1). If negative integer, -n: Refers to load curve ID n. The curve defines the relative coupling pressure (y-axis) as a function of the tolerable fluid penetration distance (x-axis).
FRIC	Coupling coefficient of friction.
FRCMIN	Minimum fluid volume fraction in an ALE element to activate coupling (default is 0.3).
NORMTYP	Penalty coupling spring direction (DIREC 1 and 2): EQ.0: normal vectors are interpolated from nodal normals (default) EQ.1: normal vectors are interpolated from segment normals.

VARIABLE	DESCRIPTION
ILEAK	Leakage control flag. Default=2 (with energy compensation).
PLEAK	Leakage control penalty factor (default=0.1)

Parameters for airbag venting hole

Card 4

Variable	1 IVSETID	2 IVTYPE	3 IBLOCK	4 VNTCOF	5	6	7	8
Type	I	I	I	F				
Default	0	0	0	0.0				
Remarks	4		5	6				

VARIABLE	DESCRIPTION
IVSETID	Set ID defining the venting hole surface(s). See Remark 4.
IVTYPE	Set type of IVSETID: EQ.0: Part Set (default). EQ.1: Part ID. EQ.2: Segment Set.
IBLOCK	Flag for considering blockage effects for porosity and vents (see Remark 5): EQ.0: no (blockage is NOT considered, default). EQ.1: yes (blockage is considered).
VNTCOF	Vent Coefficient for scaling the flow. See Remark 6.

Parameters for ALE mesh automatic definition and its transformation.

Card 5	1	2	3	4	5	6	7	8
Variable	NX/IDA	NY/IDG	NZ	MOVERN	ZOOM			
Type	I	I	I	I	I			
Default	None	None	None	0	0			
Remarks	7	7	7	8	9			

VARIABLE	DESCRIPTION
NX/IDAIR	Option 1: NX is defined (as the number of ALE elements to be generated in the x direction). This must goes together with option 1 for NY and NZ. Option 2: IDAIR is defined as Part ID of the initial air mesh. See remark 7.
NY/IDGAS	Option 1: NY is defined (as the number of ALE elements to be generated in the y direction). This must goes together with option 1 for NX and NZ. Option 2: IDGAS is defined as Part ID of the initial gas mesh. See remark 7.
NZ	Option 1: NZ is defined (as the number of ALE elements to be generated in the z direction). This must goes together with option 1 for NX and NY. Option 2: Leave blank. See remark 7.
MOVERN	ALE mesh automatic motion option (see Remark 8): EQ.0: ALE mesh is fixed in space. GT.0: Node group id. See *ALE_REFERENCE_SYSTEM_NODE ALE mesh can be moved with PRTYP=5, mesh motion follows a coordinate system defined by 3 reference nodes.
ZOOM	ALE mesh automatic expansion option (see Remark 9): EQ.0: do not expand ALE mesh EQ.1: Expand/contract ALE mesh by keeping all airbag parts contained within the ALE mesh (equivalent to PRTYP=9).

Define card 5a and 5b if NZ > 0

Card 5a	1	2	3	4	5	6	7	8
Variable	X0	Y0	Z0	X1	Y1	Z1	IPAIR	IPGAS
Type	F	F	F	F	F	F	I	I
Default	None	None	None	None	None	None	None	None

Card 5b

Variable	X2	Y2	Z2	Z3	Y3	Z3		
Type	F	F	F	F	F	F		
Default	None	None	None	None	None	None		

VARIABLE	DESCRIPTION
X0, Y0, Z0	Coordinates of origin for ALE mesh generation (node0).
X1, Y1, Z1	Coordinates of point 1 for ALE mesh generation (node1). node0=>node1 = x
X2, Y2, Z2	Coordinates of point 2 for ALE mesh generation (node2). node0=>node2 = y
X3, Y3, Z3	Coordinates of point 3 for ALE mesh generation(node3). node0=>node3 = z

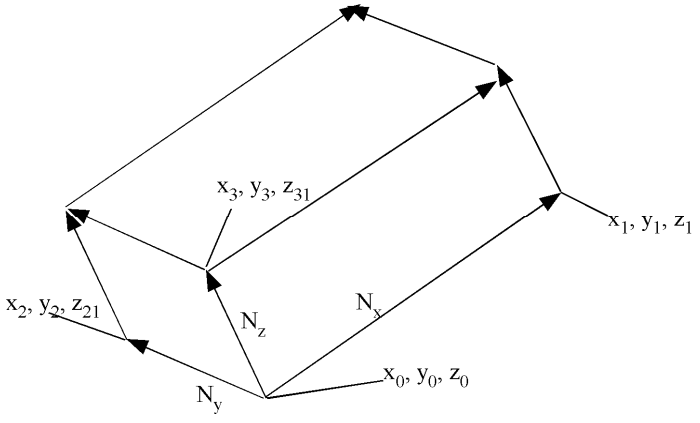


Figure 1.4.

Card 6	1	2	3	4	5	6	7	8
Variable	SWTIME		HG	NAIR	NGAS	NORIF	LCVEL	LCT
Type	F		F	I	I	I	I	I
Default	0.		0.	0	0	0	0	0
Remarks	3						10	11

VARIABLE **DESCRIPTION**

SWTIME — Time to switch from ALE method to control volume (CV) method. Once switched, a method similar to that used by the *AIRBAG_HYBRID card is used.
- EQ.0.0: switch to CV method at time=0.0.
- EQ.blank: switch time is set at 1.0E16 (ALE method is used).
- EQ.t: switch from ALE to CV method at time=t.

HG — Hourglass control for ALE fluid mesh(es).

NAIR — Number of Air components. For example, this equals 2 in case air contains 80% of N2 and 20% of O2. If air is defined as 1 single gas then NAIR=1.

NGAS — Number of inflator Gas components.

NORIF — Number of point sources or orifices (defined below)
- EQ.n: will require n lines of card 9 definitions below, one for each point source.

LCVEL — Load curve ID for inlet velocity (see also TNKVOL & TNKFINP of card 2 above). This is the same estimated velocity curve used in *SECTION_POINT_SOURCE_MIXTURE card.

LCT — Load curve ID for inlet gas temperature (see *AIRBAG_HYBRID).

Define NAIR cards below for air component

Card 7	1	2	3	4	5	6	7	8
Variable				MWAIR	INITM	AIRA	AIRB	AIRC
Type				F	F	F	F	F
Default				0	0	0	0.	0.
Remarks						12	12	12

VARIABLE	DESCRIPTION
MWAIR	Molecular weight of air component
INITA	Initial Mass Fraction of Air component(s)
AIRA	First Coefficient of molar heat capacity at constant pressure (e.g., J/mole/K, remark 12).
AIRB	Second Coefficient of molar heat capacity at constant pressure (e.g., J/mole/K^2, remark 12).
AIRC	Third Coefficient of molar heat capacity at constant pressure (e.g., J/mole/K^3, remark 12).

Define NGAS cards below for the GAS components

Cards 8…	1	2	3	4	5	6	7	8
Variable	LCMF			MWGAS		GASA	GASB	GASC
Type	I			F		F	F	F
Default	none			0		0	0.	0.
Remarks	11					12	12	12

VARIABLE	DESCRIPTION
LCMF	Load curve ID for mass flow rate (see *AIRBAG_HYBRID, e.g., kg/s).
MWGAS	Molecular weight of inflator gas components.
GASA	First Coefficient of molar heat capacity at constant pressure (e.g., J/mole/K, remark 12).
GASB	Second Coefficient of molar heat capacity at constant pressure (e.g., J/mole/K^2, remark 12).
GASC	Third Coefficient of molar heat capacity at constant pressure (e.g., J/mole/K^3, remark 12).

Define NORIF cards below for each point source

Cards 9…	1	2	3	4	5	6	7	8
Variable	NODEID	VECID	ORIFARE					
Type	I	I	I					
Default	0	0	0					

VARIABLE	DESCRIPTION
NODEID	The node ID defining the point source.
VECID	The vector ID defining the direction of flow at the point source.
ORIFARE	The orifice area at the point source.

Remarks:

1. This set ID typically contains the Lagrangian segments of the 3 parts that are coupled to the inflator gas: airbag, airbag canister (compartment), inflator. As in all control-volume, orientation of elements representing bag and canister should point outward. During the ALE phase the segment normal will be reversed automatically for fluid-structure coupling. *However, the orientation of inflator element normal vectors should point to its center.* See Figure 1.5.

2. Atmospheric density for the ambient gas (air) can be computed from $\rho_{amb} = P_{amb}/(R*T_{amb})$

3. Since ALL ALE related activities will be turned off after the switch from ALE method to control-volume method, no other ALE coupling will exist beyond t=SWTIME.

4. Vent definition will be used for ALE venting. Upon switching area of the segments will be used for venting as a23 in *AIRBAG_HYBRID.

5. Fabric porosity for ALE and *AIRBAG_HYBRID can be defined on MAT_FABRIC. Define FLC and FAC on *MAT_FABRIC. FVOPT 7 and 8 will be used for both ALE and *AIRBAG_HYBRID. IBLOCK=0 will use FVOPT=7 and IBLOCK=1 will use FVOPT=8.

6. VCOF will be used to scale the vent area for ALE venting and this coefficient will be used as vent coefficient c23 for *AIRBAG_HYBRID upon switching.

7. If Nz, Ny and Nz are defined (option 1), card 5a and card 5b should be defined to let LS-DYNA generate the mesh for ALE. Alternatively if Nz is 0 (option 2), then Nx=IDAIR and Ny=IDGAS. In the later case the user need to supply the ALE mesh whose PID=IDAIR.

8. If the airbag moves with the vehicle, set MOVERN=GROUPID, this GROUPID is defined using *ALE_REFERENCE_SYSTEM_NODE. The 3 nodes defined in ALE_REFERENCE_SYSTEM_NODE will be used to transform the ALE mesh. The point sources will also follow this motion. This simulates PRTYP=5 in the *ALE_REFERENCE_SYSTEM_GROUP card.

9. Automatic expansion/contraction of the ALE mesh to follow the airbag expansion can be turned on by setting zoom=1. This feature is particularly useful for fully folded airbags requiring very fine ale mesh initially. As the airbag inflates the ale mesh will be automatically scaled such that the airbag will be contained within the ALE mesh. This simulates PRTYP=9 in the *ALE_REFERENCE_SYSTEM_GROUP card.

10. There are 3 methods for defining the inlet gas velocity:
Option 1: define LCVEL = 0 ➔ TNKVOL = Tank volume, and TNKFINP = Tank final pressure from tank test data. Inlet gas velocity is estimated by LSDYNA method (testing).
Option 2: define LCVEL = 0 ➔ TNKVOL = inflator can volume, and TNKFINP = blank. Inlet gas velocity is estimated automatically by Lian-Bhalsod-Olovsson method.
Option 3: define LCVEL = n ➔ TNKVOL =0, and TNKFINP = 0. Inlet gas velocity is defined by user via a load curve ID = n.

11. LCT and LCIDM should have the same number of sampling points.

12. The per-mass-unit, temperature-dependent, constant-pressure heat capacity is

$$C_p(T) = \frac{[A + B*T + C*T^2]}{MW} \sim \frac{J}{kg*K} \qquad B \sim J/(mole*K^2)$$

$$A = \tilde{C}_{P0} \sim J/(mole*K) \qquad C \sim J/(mole*K^3)$$

The units shown are only for demonstration of the equation.

13. Sometimes CTYPE=6 may be used for complex folded airbag. NQUAD=2 may be used as a starting value and increase as necessary depending on the relative mesh resolutions of the Lagrangian and ALE meshes.

14. Use a load curve for PFAC whenever possible. It tends to be more robust.

*AIRBAG_ALE →
- AIR →
  - *PART (AMMG2)
  - *SECTION_SOLID
  - *MAT_GAS_MIXTURE
- GAS →
  - *PART (AMMG1)
  - *SECTION_POINT_SOURCE_MIXTURE
  - *MAT_GAS_MIXTURE
- Couplings → *CONSTRAINED_LAGRANGE_IN_SOLID
- ALE Mesh motion → *ALE_REFERENCE_SYSTEM_GROUP
- CV → *AIRBAG_HYBRID
- VENT → Venting Definitions

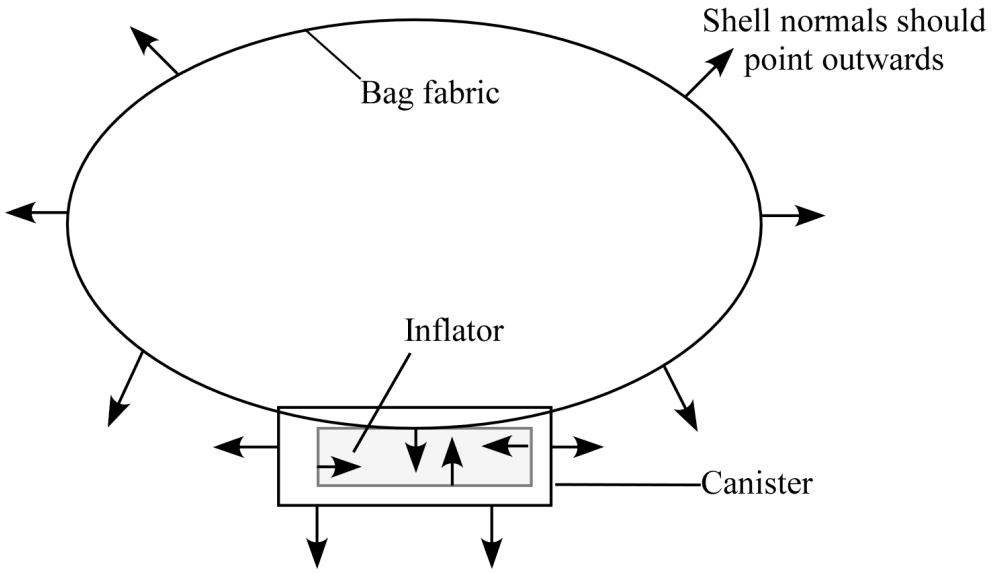


Figure 1.5.

Example 1:

```
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|....8
*AIRBAG_ALE
$#1     SID   SIDTYPE      NONE      NONE      NONE      NONE       MWD      SPSF
        123         1         0         0         0         0       0.0       0.0
$#2 ATMOST    ATMOSP      NONE        GC        CC    TNKVOL     TNKFP
    298.15 1.0132E-4         0     8.314       0.0       0.0       0.0
$#3  NQUAD     CTYPE      PFAC      FRIC    FRCMIN  NORMTYPE     ILEAK     PLEAK
          4         4     -1000       0.0       0.3         0         2       0.1
$#4 VSETID  IVSETTYP    IBLOCK  VENTCOEF
          1         2         0      1.00
$#5NXIDAIR   NYIDGAS        NZ    MOVERN      ZOOM
     50000     50003         0         0         0
$#6 SWTIME      NONE        HG      NAIR      NGAS     NORIF     LCVEL       LCT
   1000.00     0.000     1.e-4         1         1         8      2002      2001
$#7 AIR          NONE      NONE     MWAIR     INITM      AIRA      AIRB      AIRC
          0         0         0   0.02897      1.00    29.100   0.00000   0.00000
$#8 GASLCM       NONE      NONE     MWGAS      NONE      GASA      GASB      GASC
       2003         0         0    0.0235         0    28.000   0.00000   0.00000
$#9 NODEID    VECTID  ORIFAREA
    100019         1 13.500000
    100020         2 13.500000
    100021         3 13.500000
    100022         4 13.500000
    100023         5 13.500000
    100024         6 13.500000
    100017         7 13.500000
    100018         8 13.500000
$ PFAC CURVE = penalty factor curve.
*DEFINE_CURVE
$     lcid      sidr       sfa       sfo      offa      offo    dattyp
      1000         0       0.0       2.0       0.0       0.0
$                 a1                  o1
                 0.0         0.00000000
           1.0000000        4.013000e-04
*SET_SEGMENT_TITLE
vent segments (defined in IVSETID)
         1       0.0       0.0       0.0       0.0
      1735      1736       661      1697       0.0       0.0       0.0       0.0
      1735      2337      1993      1736       0.0       0.0       0.0       0.0
      1735      1969      1988      2337       0.0       0.0       0.0       0.0
      1735      1697       656      1969       0.0       0.0       0.0       0.0
*DEFINE_VECTOR
$#     vid        xt        yt        zt        xh        yh        zh
         1       0.0       0.0-16.250000 21.213200 21.213200-16.250000
         2       0.0       0.0-16.250000 30.000000-1.000e-06-16.250000
         3       0.0       0.0-16.250000 21.213200-21.213200-16.250000
         4       0.0       0.0-16.250000-1.000e-06-30.000000-16.250000
         5       0.0       0.0-16.250000-21.213200-21.213200-16.250000
         6       0.0       0.0-16.250000-30.0000001.0000e-06-16.250000
         7       0.0       0.0-16.250000-21.213200 21.213200-16.250000
         8       0.0       0.0-16.2500001.0000e-06 30.000000-16.250000
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|....8
```

In this example, pre-existing background air mesh with part ID 50000 and gas mesh with part ID 50003 are used. Thus NZ = 0. There is no mesh motion nor expansion allowed. An inlet gas velocity curve is provided.

Example 2:

```
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|....8
$ SIDTYP: 0=SGSID; 1=PSID
*AIRBAG_ALE
$#1    SID   SIDTYPE      NONE      NONE      NONE      NONE       MWD      SPSF
         1         1         0        0.        0.        0.        0.        0.
$#2 ATMOST    ATMOSP      NONE        GC        CC    TNKVOL     TNKFP
       298.   101325.       0.0     8.314        1.    6.0E-5         0
$#3  NQUAD     CTYPE      PFAC      FRIC    FRCMIN  NORMTYPE     ILEAK     PLEAK
         2         6      -321       0.0       0.3         1         2       0.1
$#4 VSETID  IVSETTYP    IBLOCK  VENTCOEF
         0         0         0         0
$#5NXIDAIR   NYIDGAS        NZ    MOVERN      ZOOM
        11        11         9
$5b     x0        y0        z0        x1        y1        z1  NOT-USED  NOT-USED
      -0.3      -0.3    -0.135       0.3      -0.3    -0.135
$5c     x2        y2        z2        x3        y3        z3  NOT-USED  NOT-USED
      -0.3       0.3    -0.135      -0.3      -0.3      0.39
$#6 SWTIME      NONE        HG      NAIR      NGAS     NORIF     LCVEL       LCT
   0.04000     0.005     1.e-4         2         1         1         0         2
$#7 AIR         NONE      NONE     MWAIR     INITM      AIRA      AIRB      AIRC
                                   0.028      0.80    27.296   0.00523
                                   0.032      0.20    25.723   0.01298
$#8 GASLCM      NONE      NONE     MWGAS      NONE      GASA      GASB      GASC
         1                        0.0249               29.680   0.00880
$#9 NODEID    VECTID  ORIFAREA
      9272         1   1.00e-4
$ Lagrangian shell structure to be coupled to the inflator gas
*SET_PART_LIST
         1       0.0       0.0       0.0       0.0
         1         2         3
*DEFINE_VECTOR
$0.100000E+01,  10.000000000
$      vid        xt        yt        zt        xh        yh        zh
         1       0.0       0.0       0.0       0.0       0.0  0.100000
$ bag penetration ~ 1 mm  <=====>  P_coup ~ 500000 pascal ==> ~ 5 atm
*DEFINE_CURVE
$     lcid      sidr       sfa       sfo      offa      offo    dattyp
       321         0       0.0       0.0       0.0       0.0
$                a1                  o1
                0.0                 0.0
         0.00100000       5.0000000e+05
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|....8
```

In this example, LS-DYNA automatically creates the background ALE mesh with:
NX = 11 elements in the x direction.
NY = 11 elements in the y direction.
NZ = 9 elements in the z direction.

*AIRBAG_INTERACTION

Purpose: To define two connected airbags which vent into each other.

Define one card for each airbag interaction definition

Card	1	2	3	4	5	6	7	8
Variable	AB1	AB2	AREA	SF	PID	LCID	IFLOW	
Type	I	I	F	F	I	I	I	
Default	none	none	none	none	0	0	0	

VARIABLE	DESCRIPTION
AB1	First airbag ID, as defined on *AIRBAG card.
AB2	Second airbag ID, as defined on *AIRBAG card.
AREA	Orifice area between connected bags. LT.0.0: |AREA| is the load curve ID defining the orifice area as a function of absolute pressure. EQ.0.0: AREA is taken as the surface area of the part ID defined below.
SF	Shape factor. LT.0.0: |SF| is the load curve ID defining vent orifice coefficient as a function of relative time.
PID	Optional part ID of the partition between the interacting control volumes. AREA is based on this part ID.
LCID	Load curve ID defining mass flow rate versus pressure difference, see *DEFINE_CURVE. If LCID is defined AREA, SF and PID are ignored.
IFLOW	Flow direction LT.0: One way flow from AB1 to AB2 only. EQ.0: Two way flow between AB1 and AB2. GT.0: One way flow from AB2 to AB1 only.

Remarks:

Mass flow rate and temperature load curves for the secondary chambers must be defined as null curves, for example, in the DEFINE_CURVE definitions give two points (0.0,0.0) and (10000.,0.0).

All input options are valid for the following airbag types:

*AIRBAG_SIMPLE_AIRBAG_MODEL

*AIRBAG_WANG_NEFSKE

*AIRBAG_WANG_NEFSKE_JETTING

*AIRBAG_WANG_NEFSKE_MULTIPLE_JETTING

*AIRBAG_HYBRID

*AIRBAG_HYBRID_JETTING

The LCID defining mass flow rate vs. pressure difference may additionally be used with:

*AIRBAG_LOAD_CURVE

*AIRBAG_LINEAR_FLUID

If the AREA, SF, and PID defined method is used to define the interaction then the airbags must contain the same gas, i.e. C_p, C_V and g must be the same. The flow between bags is governed by formulas which are similar to those of Wang-Nefske.

*AIRBAG_PARTICLE

Purpose: To define an airbag using the particle method. Please note that the options of HCONV, IDRP, SIDH, STYPEH, H, IMOM are only available in R4.

Card Format

Card 1

Variable	SID1	STYPE1	SID2	STYPE2	BLOCK	HCONV	FRIC	IRPD
Type	I	I	I	I	I	F	F	I
Default	none	0	0	0	0	0.0	0.0	0

Card 2

Variable	NP	UNIT	VISFLG	TATM	PATM	NVENT	TEND	TSW
Type	F	I	I	F	F	I	F	F
Default	200,000	0	0	393K	1 atm	0	1.0E+10	1.0E+10

Card 3

Variable	IAIR	NGAS	NORIF	NID1	NID2	NID3		
Type	I	I	I	I	I	I		
Default	0	none	none	0	0	0		

Optional Cards if HCONV>1 – Define HCONV cards one for each heat convection part or part set.

Optional	1	2	3	4	5	6	7	8
Variable	SIDH	STYPEH	H	PFRIC				
Type	I	I	F	F				
Default	none	none	none	none				

Optional Cards if NVENT>1 – Define NVENT cards one for vent hole.

Optional	1	2	3	4	5	6	7	8
Variable	SID3	STYPE3	C23	LCTC23	LCPC23	ENH_V	PPOP	
Type	I	I	I	I	I	I	F	
Default	0	none	none	0	0	0	0.0	

Optional Cards if IAIR=1

Optional	1	2	3	4	5	6	7	8
Variable	PAIR	TAIR	XMAIR	AAIR	BAIR	CAIR		
Type	F	F	F	F	F	F		
Default	PATM	TATM	none	none	0.0	0.0		

NGAS Cards (i=1,2,...,n)

	1	2	3	4	5	6	7	8
Variable	LCMi	LCTi	XMi	Ai	Bi	Ci	INFGi	
Type	I	I	F	F	F	F	I	
Default	none	none	none	none	0.0	0.0	1	

NORIF Cards (i=1,2,...,n)

	1	2	3	4	5	6	7	8
Variable	NIDi	ANi	VDi	CAi	INFOi	IMOM		
Type	I	F	I	F	I	I		
Default	none	none	none	30 Deg	1	0		

VARIABLE	DESCRIPTION
SID1	Part or part set ID defining the complete airbag.
STYPE1	Set type: EQ.0: Part EQ.1: Part set
SID2	Part or part set ID defining the internal parts of the airbag.
STYPE2	Set type: EQ.0: Part EQ.1: Part set EQ.2: Number of parts to read
BLOCK	Blocking, see remark 4. EQ.0: Off EQ.1: On
HCONV	Number of heat convection parts or part sets, see remark 5.
FRIC	Friction factor. (Default =0.0) See remark 3.

VARIABLE	DESCRIPTION
IRDP	Dynamic scaling of particle radius (Develoment source only) EQ.0: Off EQ.1: On
NP	Number of particles. (Default =200,000.)
UNIT	Unit system: EQ.0: kg-mm-ms-K EQ.1: SI EQ.2: tonne-mm-ms
VISFLG	Visible particles EQ.0: Particles are invisible (Smaller D3PLOT files, memory) EQ.1: Particle are visible.
TATM	Atmospheric temperature (Default =293K).
PATM	Atmospheric pressure (Default=1 ATM).
NVENT	Number of vent hole parts or part sets
TEND	Time when all (NP) particles have entered bag (Default=1.0E+10).
TSW	Time for switch to control volume calculation (Default=1.0E+10).
IAIR	Initial gas inside bag considered: EQ.0: No EQ.1: Yes
NGAS	Number of gas components
NORIF	Number of orifices
NID1-NID3	Three nodes defining a moving coordinate system for the direction of flow through the gas inlet nozzles (Default=fixed system)
SIDH	Part or part set ID defining heat convection.
STYPEH	Set type: EQ.0: Part EQ.1: Part set
H	Heat convection coefficient (W//Km^2). See *AIRBAG_HYBRID developments (Resp. P.O. Marklund).
SID3	Part or part set ID defining vent holes.

VARIABLE	DESCRIPTION
STYPE3	Set type: EQ.0: Part EQ.1: Part set
C23	Vent hole coefficient, a parameter of Wang-Nefske leakage. (Default=1.0). See remark below.
LCTC23	Load curve defining vent hole coefficient as a function of time. See remarks 1 and 2 below.
LCPC23	Load curve defining vent hole coefficient as a function of pressure. See remarks 1 and 2 below.
ENH_V	Enhance venting option (Default=0.0).
PPOP	Pressure difference between interior and ambient pressure to open the vent holes. Once the vents are open, they will stay open.
PAIR	Initial pressure inside bag (Default PAIR=PATM).
TAIR	Initial temperature inside bag (Default, TAIR=TATM).
XMAIR	Molar mass of gas initially inside bag.
AAIR-CAIR	Constant, linear, and quadratic heat capacity parameters.
LCMi	Mass flow rate curve for gas component i..
LCTi	Temperature curve for gas component i.
XMi	Molar mass of gas component i.
Ai-Ci	Constant, linear, and quadratic heat capacity parameters for gas component i.
INFGi	Inflator ID for this gas component (Default=1).
NIDi	Node ID defining the location of nozzle i.
ANi	Area of nozzle i (Default-all nozzles are assigned the same area).
VDi	Vector ID (Initial direction of gas inflow at nozzle i).
CAi	Cone angle in degrees. (Default=30 degrees). *This option is obsolete in the R4 release.*

VARIABLE	DESCRIPTION
INFOi	Inflator ID for this orifice (Default=1).
IMOM	Inflator reaction force (Development source only). EQ.0: Off EQ.1: On

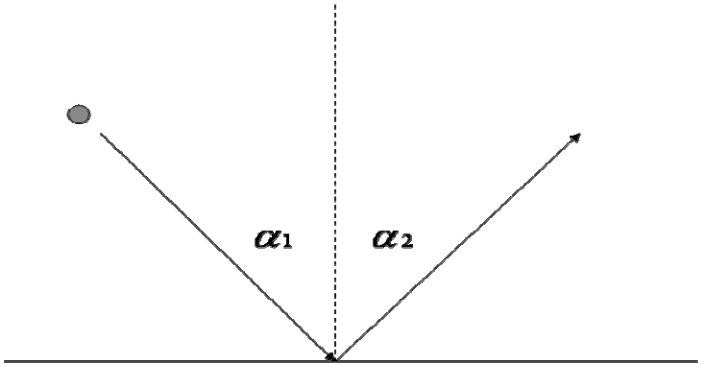
Remarks

1. Total vent hole coefficient = C23•LCTC23•LCPC23.

2. If not specified, a constant value of 1.0 is assumed.

3. Friction factor to simulate the surface roughness

Frictionless $\alpha_1 = \alpha_2$



Friction :

$0 \le F_r \le 1$: **Surface Roughness Factor, Rebouncing angle**

$F_r = 0; \alpha = \alpha_1 + \alpha_2$

$F_r = 1; \alpha = 0$

4. Total fabric porosity coefficient.
   BLOCK=0: coefficient = FLC•FAC
   BLOCK=1: coefficient = FLC•FAC•(1-blockage)•ELA(blockage)

5. dE/dt = A • H • (Tbag – Tatm)
   A is part area
   H is user defined heat convection coefficient
   Tbag is weighted average temperature of the particles impacting the part
   Tatm is ambient temperature

***AIRBAG_REFERENCE_GEOMETRY_**{*OPTION*}_(*OPTION*}

Available options include:

<BLANK>

BIRTH

RDT

The reference geometry becomes active at time BIRTH. Until this time the input geometry is used to inflate the airbag. Until the birth time is reached the actual geometry is used to determine the time step size even if RDT is active.

If RDT is active the time step size will be based on the reference geometry once the solution time exceeds the birth time. This option is useful for shrunken bags where the bag does not carry compressive loads and the elements can freely expand before stresses develop. If this option is not specified, the time step size will be based on the current configuration and will increase as the area of the elements increase. The default may be much more expensive but possibly more stable.

Purpose: If the reference configuration of the airbag is taken as the folded configuration, the geometrical accuracy of the deployed bag will be affected by both the stretching and the compression of elements during the folding process. Such element distortions are very difficult to avoid in a folded bag. By reading in a reference configuration such as the final unstretched configuration of a deployed bag, any distortions in the initial geometry of the folded bag will have no effect on the final geometry of the inflated bag. This is because the stresses depend only on the deformation gradient matrix:

$$F_{ij} = \frac{\partial x_i}{\partial X_j}$$

where the choice of X_j may coincide with the folded or unfold configurations. It is this unfolded configuration which may be specified here.

Note that a reference geometry which is smaller than the initial airbag geometry will not induce initial tensile stresses.

If a liner is included and the parameter LNRC set to 1 in *MAT_FABRIC, compression is disabled in the liner until the reference geometry is reached, i.e., the fabric element becomes tensile.

Define the follow card if and only if the option BIRTH is specified in the keyword.

Card	1	2	3	4	5	6	7	8
Variable	BIRTH							
Type	F							
Default	0.0							

Card Format (I8,3E16.0) The next "*" keyword card terminates this input.

Card 2,...	1	2	3	4	5	6	7	8	9	10
Variable	NID	X		Y		Z				
Type	I	F		F		F				
Default	none	0.		0.		0.				
Remarks										

VARIABLE	DESCRIPTION
BIRTH	Time at which the reference geometry activates (default=0.0)
NID	Node ID for which a reference configuration is defined. Nodes defined in this section must also appear under the *NODE input. It is only necessary to define the reference coordinates of nodal points, if their coordinates are different than those defined in the *NODE section.
X	x coordinate
Y	y coordinate
Z	z coordinate

*AIRBAG_SHELL_REFERENCE_GEOMETRY

Purpose: Usually, the input in this section is not needed; however, sometimes it is convenient to use disjoint pre-cut airbag parts to define the reference geometries. If the reference geometry is based only on nodal input, this is not possible since in the assembled airbag the boundary nodes are merged between parts. By including the shell connectivity with the reference geometry, the reference geometry can be based on the pre-cut airbag parts instead of the assembled airbag. The elements, which are defined in this section, must have identical element ID's as those defined in the *ELEMENT_SHELL input, but the nodal ID's, which may be unique, are only used for the reference geometry. These nodes are defined in the *NODE section, but can also be additionally defined above under *AIRBAG_REFERENCE_GEOMETRY. The element orientation and n1-n4 ordering must be identical to the *ELEMENT_SHELL input.

Card Format (6I8)

Card 1	1	2	3	4	5	6	7	8	9	10
Variable	EID	PID	N1	N2	N3	N4				
Type	I	I	I	I	I	I				
Default	none	none	none	none	none	none				
Remarks			3	3	3	3				

VARIABLE	DESCRIPTION
EID	Element ID
PID	Optional part ID, see *PART, the part ID is not used in this section.
N1	Nodal point 1
N2	Nodal point 2
N3	Nodal point 3
N4	Nodal point 4

*ALE

The keyword *ALE provides a way of defining input data pertaining to the Arbitrary-Lagrangian-Eulerian capability. The keyword cards in this section are defined in alphabetical order:

***ALE_FSI_PROJECTION**

***ALE_FSI_SWITCH_MMG_**{*OPTION*}

***ALE_MULTI-MATERIAL_GROUP**

***ALE_REFERENCE_SYSTEM_CURVE**

***ALE_REFERENCE_SYSTEM_GROUP**

***ALE_REFERENCE_SYSTEM_NODE**

***ALE_REFERENCE_SYSTEM_SWITCH**

***ALE_SMOOTHING**

***ALE_TANK_TEST**

***ALE_UP_SWITCH**

For other input information related to the ALE capability, see keywords:

*ALE_TANK_TEST
*BOUNDARY_AMBIENT_EOS
*CONSTRAINED_EULER_IN_EULER
*CONSTRAINED_LAGRANGE_IN_SOLID
*CONTROL_ALE
*DATABASE_FSI
*INITIAL_VOID
*INITIAL_VOLUME_FRACTION
*INITIAL_VOLUME_FRACTION_GEOMETRY
*SECTION_SOLID
*SECTION_POINT_SOURCE (for gas only)
*SECTION_POINT_SOURCE_MIXTURE
*SET_MULTI-MATERIAL_GROUP_LIST
*CONSTRAINED_EULER_IN_EULER

SINGLE GASEOUS MATERIAL	MULTIPLE GASEOUS MATERIAL
*EOS_LINEAR_POLYNOMIAL *EOS_IDEAL_GAS *MAT_NULL	*MAT_GAS_MIXTURE *INTIAL_GAS_MIXTURE

*ALE_FSI_PROJECTION

Purpose: This card provides a coupling method for simulating the interaction between a Lagrangian material set (structure) and ALE material set (fluid). The nearest ALE nodes are projected onto the Lagrangian structure surface at each time step. This method does not conserve energy, as mass and momentum are transferred via constrained based approach.

Card 1

Variable	LAGSID	ALESID	LSIDTYP	ASIDTYP	SMMGID	ICORREC	INORM	
Type	I	I	I	I	I	I	I	
Default	0	0	0	0	0	0	0	

Card 2

Variable	BIRTH	DEATH						
Type	F	F						
Default	0.0	1.E+10						

VARIABLE	DESCRIPTION
LAGSID	A set ID defining the Lagrangian part(s) for this coupling (structures).
ALESID	A set ID defining the ALE part(s) for this coupling (fluids).
LSIDTYP	Lagrangian set ID type EQ.0: Part set ID (PSID), EQ.1: Part ID (PID).
ASIDTYP	ALE set ID type EQ.0: Part set ID (PSID), EQ.1: Part ID (PID).
SMMGID	A set ID referring to a group of one or more ALE-Multi-Material-Group (AMMG) IDs which represents the ALE materials interacting with the Lagrangian structure. This SMMGID is a set ID defined by *SET_MULTI-MATERIAL_GROUP_LIST.

VARIABLE	DESCRIPTION
ICORREC	Advection error correction method (See Remark 1). EQ.1: ALE mass is conserved. Leaked mass is moved, EQ.2: ALE mass is almost conserved, EQ.3: No correction performed (default). ALE mass is conserved. Some leakage may occur. This may be the best solution.
INORM	Type of coupling. EQ.0: Couple in all directions, EQ.1: Couple in compression and tension (free sliding), EQ.2: Couple in compression only (free sliding). This choice requires ICORREC=3.
BIRTH	Start time for coupling.
DEATH	End time for coupling.

Remarks:

1. As the ALE nodes are projected onto the closest Lagrangian surface, there may be some advection errors introduced. These errors may result in a small element mass fraction being present on the "wrong" side of the coupled Lagrangian surface. There are 3 possible scenarios:

   a. Mass on the wrong side of the Lagrangian structure may be moved to the right side. This may cause P oscillations. No leakage will occur.

   b. Mass on the wrong side is deleted. Mass on the right side is scaled up to compensate for the lost mass. No leakage will occur.

   c. Mass on the wrong side is allowed (no correction performed). Some leakage may occur. This may be the most robust and simplest approach.

Example:

Model Summary:
H1 = AMMG1 = background air mesh.
H2 = AMMG2 = fluid inside container S3.
S3 = cylinder containing AMMG2.
S4 = dummy target cylinder for impact.
The gas inside S3 is AMMG2. S3 is given an initial velocity and it will impact S4.

```
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|....8
*ALE_MULTI-MATERIAL_GROUP
         1         1
         2         1
*SET_MULTI-MATERIAL_GROUP_LIST
        22
         2
*ALE_FSI_PROJECTION
$   LAGSID    ALESID   LSIDTYP   ASIDTYP    SMMGID   ICORREC     INORM
         3         1         1         1        22         3         2
$    BIRTH     DEATH
       0.0      20.0
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|....8
```

***ALE_FSI_SWITCH_MMG_**{*OPTION*}

Purpose: This card is used to allow the switching of an ALE multi-material-group ID (AMMGID) of a fluid as that fluid passes across a monitoring surface. This monitoring surface may be a Lagrangian shell structure, or a segment set. It does not have to be included in the slave set of the coupling card (CLIS).

Available options include:

<BLANK>

ID

An ID number (up to 8 digits) may be defined for this switch command in the first 10-character space.

or

TITLE

A title for the card may be input between the 11th and 80th character on the title-ID line. The optional title line precedes all other cards for this command.

The user can explicitly define a title for this coupling.

Title Card Format
The following card is read if and only if the ID or TITLE option is specified.

Optional

Variable	ID	TITLE
Type	I10	A70

Card 1	1	2	3	4	5	6	7	8
Variable	SID	STYPE	NQUAD	XOFF	BTIME	DTIME	NFREQ	NFOLD
Type	I	I	I	F	F	F	I	I
Default	none	0	1	0.0	0.0	1.0E20	1	0

Card 2	1	2	3	4	5	6	7	8
Variable	FR_MMG	TO_MMG	XLEN					
Type	I	I	F					
Default	none	none	0.0					

VARIABLE	DESCRIPTION
SID	A set ID defining a monitoring surface over which an ALE fluid flows across, and its ALE multi-material-group-ID (AMMGID) is switched. The monitoring surface may be a Lagrangian shell structure, or a segment set. This surface, if Lagrangian, does not have to be included in the coupling definition (see remark 4).
STYPE	Set ID type of the above SID. EQ.0: Part set ID (PSID) (default). EQ.1: Part ID (PID). EQ.2: Segment set ID (SGSID).
NQUAD	The number of flow-sensor points to be distributed over each monitoring surface/segment. There should be enough sensor points distributed to monitor the flow in each ALE element intersected by this monitoring surface (default=1, see remark 3).
XOFF	An offset distance away from the monitoring surface, beyond which the AMMGID switching occurs. The direction of XOFF is defined by the normal vector of the monitoring segment. This offset distance, in general, should be at least 2 ALE element widths away from, and beyond the monitoring interface (default=0.0).
BTIME	Start time for the AMMGID switch to be activated (default=0.0).
DTIME	Ending time for the AMMGID switch (default=1.0E20).
NFREQ	Number of computational cycles between ALE switch check (default=1).
NFOLD	Flag for checking folding logic (default=0=off). If NFOLD=1=on, then LS-DYNA will check if the monitoring segment is in the fold, applicable to airbag. If the monitoring segment is still located within a folded (shell) region, then no switching is allowed yet until it has unfolded.

VARIABLE	DESCRIPTION
FR_MMG	This is the AMMG-SID before the switch. The AMMG-SID corresponds to the SID defined under the *SET_MULTI-MATERIAL_GROUP_LIST (SMMGL) card. This SID points to one or more AMMGs. See Remark 1.
TO_MMG	This is the AMMG-SID after the switch. The AMMG-SID corresponds to the SID defined under the *SET_MULTI-MATERIAL_GROUP_ LIST card. This SID points to one or more AMMGs. See Remark 1.
XLEN	This is an absolute distance for distributing the flow sensor points over the ALE elements. To make sure that at least 1 sensor point, defined on each Lagrangian segment, is present in each ALE element to track the flow of an AMMG, XLEN may be estimated as roughly half the length of the smallest ALE element in the mesh. See Remark 3.

Remarks:

1. There is a correspondence between the FR_MMG and TO_MMG. Consider an example where:

   a. The FR_MMG SID points to a SID=12 (the SID of its SMMGL card is 12, and this SID contains AMMG 1 and AMMG 2)

   b. The TO_MMG points to a SID=34 (the SID of the SMMGL card is 34, and this SID contains AMMG 3 and AMMG 4)

   Then, AMMG 1, if switched, will become AMMG 3, and AMMG 2, if switched, will become AMMG 4.

2. The ID option must be activated if the parameter SWID is used in the *DATABAS_FSI card. Then the accumulated mass of an AMMG that goes through a tracking surface, and being switched, will be reported via the parameter "PLEAK" in the "dbfsi" ASCII output file (or equivalently the "POROSITY" parameter inside LS-Prepost ASCII plotting option).

3. When both NQUAD and XLEN are defined, whichever gives smaller sensor-point interval distance will be used. XLEN may give better control as in the case of a null shell acting as the monitoring surface. As this null shell is stretched, NQUAD distribution of sensor-points may not be adequate, but XLEN would be.

4. The monitoring surface does not have to be included in the slave set of the coupling card. However, at least one coupling card must be present in the model. The monitoring segment set can be made up of Lagrangian or ALE nodes.

Example:

Consider a simple airbag model with 3 part IDs:
H25 = AMMG1 = Inflator gas injected into the airbag.
H24 = AMMG2 = Air outside the airbag = background mesh
H26 = AMMG3 = Dummy AMMG of inflator gas after it passes through a vent hole.
S9 = A Lagrangian shell part representing a vent hole.
S1 = A Lagrangian shell part representing the top half of an airbag.
S2 = A Lagrangian shell part representing the bottom half of an airbag.

The inflator gas inside the airbag is distinguished from the inflator gas that has passed through the monitoring surface (vent hole) to the outside of the airbag by assigning different ALE multi-material group set ID to each. The dummy fluid part (H26) should have the same material and EOS model IDs as the before-switched fluid (H25).

Fr_MMG=1 ==> points to AMMGID=1 ==> points to H25 (inflator gas if inside)
To_MMG=2 ==> points to AMMGID=3 ==> points to H26 (inflator gas if outside)

```
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|....8
*ALE_MULTI-MATERIAL_GROUP
        25         1
        24         1
        26         1
*DATABASE_FSI
$     TOUT                                  [STYPE: 0=PSID ; 1=PID ; 2=SGSID]
    0.1000
$ DBFSI_ID       SID     STYPE  AMMGSWID  LDCONVID
         1         1         1
         2         2         1
         3         9         1     90000
*SET_MULTI-MATERIAL_GROUP_LIST
       125
         1
*SET_MULTI-MATERIAL_GROUP_LIST
       126
         3
*ALE_FSI_SWITCH_MMG_ID
     90000
$       SID   SIDTYPE     NQUAD      XOFF     BTIME     DTIME     NFREQ      FOLD
         9         1         3     -20.0       5.0       0.0         1         1
$   Fr_MMG    To_MMG     XCLEN
       125       126        5.
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|....8
```

NOTE:

1. The *DATABASE_FSI card tracks 3 surface entities: (a) top half of an airbag, (b) bottom half of an airbag, and (c) the vent hole monitoring surface where the AMMGID of the inflator gas is switched.

2. The amount of mass passing through the vent hole during the switch is output to a parameter called “pleak” in a “dbfsi” ASCII file. See *DATABASE_FSI.

3. The *ALE_FSI_SWITCH_MMG_ID card track any flow across S9 and switch the AMMGSID from 125 (AMMG 1) to 126 (AMMG 3).

*ALE_MULTI-MATERIAL_GROUP

Purpose: This command defines the appropriate ALE material groupings for interface reconstruction when many ALE Multi-Material Groups (AMMG) are present in a model. This card is required when ELFORM=11 in the *SECTION_SOLID card. This is the ALE Multi-Material element formulation requiring at least 2 ALE materials to be present in a model. Each data line represents 1 ALE multi-material group (AMMG), with the first line referring to group 1, second line group 2, etc. Each AMMG represents one unique "fluid" which may undergo interaction with any Lagrangian structure in the model.

Card	1	2	3	4	5	6	7	8
Variable	SID	IDTYPE						
Type	I	I						
Default	none	0						
Remarks	1							

VARIABLE	DESCRIPTION
SID	Set ID.
IDTYPE	Set type: EQ.0: Part set, EQ.1: Part.

Remarks:

1. When ELFORM=12 in the *SECTION_SOLID card (single material and void), this card should not be used. In one model, ELFORM=12 cannot be used together with ELFORM=11. If possible, it is recommended that ELFORM=11 be used as it is the most robust and versatile formulation for treating multi-material ALE parts.

2. Each AMMG is given an ID (AMMGID), and consists of one or more PART ID's. The interface of each AMMGID is reconstructed as it evolves dynamically. Each AMMGID is represented by one material contour color in LS-PREPOST.

3. The maximum number of AMMGIDs allowed has been increased to 20. However, there may be 2, at most 3, AMMGs inside an ALE element at anytime. If there are more than 3 AMMGs inside any 1 ALE element, the ALE mesh needs refinement. Better accuracy is obtained with 2 AMMGs in mixed elements.

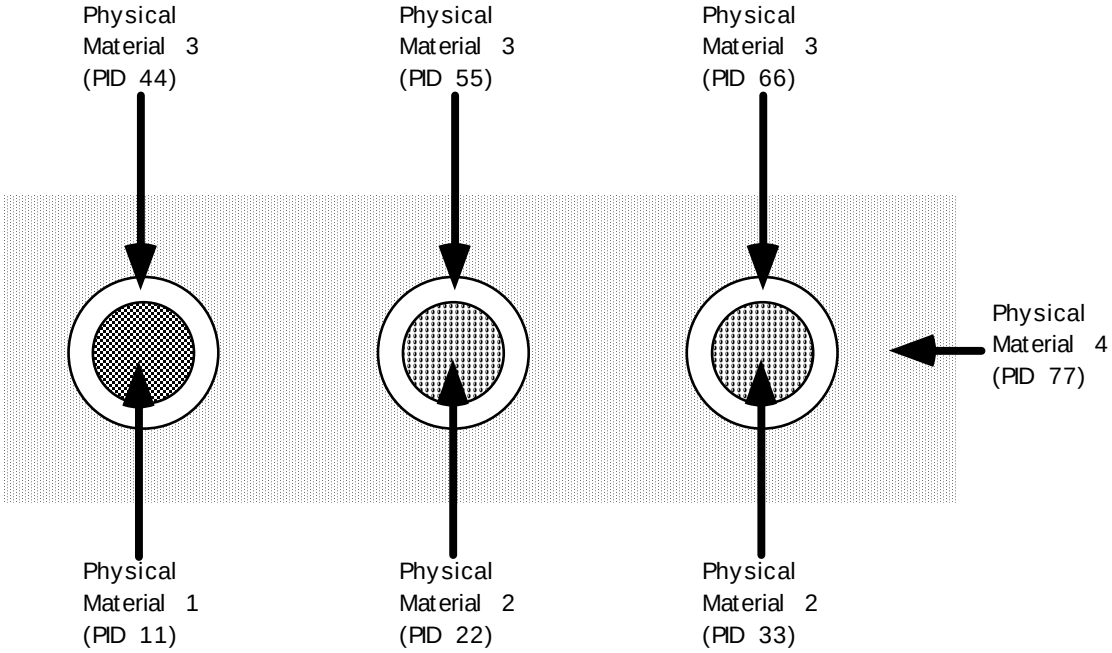
4. To plot these AMMGIDs in LS-PREPOST:
   [FCOMP] ⇒ [MISC] ⇒ [VOLUME FRACTION OF AMMGID #] ⇒ [APPLY]
   (Note: Contour definitions maybe different for gas mixture application)

5. It is very important to distinguish among the
   (a) Physical materials,
   (b) PART IDs, and
   (c) AMMGIDs.

   A *PART may be any mesh component. In ALE formulation, it is simply a geometric entity and a time=0 concept. This means a *PART may be a mesh region that can be filled with one or more AMMGIDs at time zero, via a volume filling command (*INITIAL_VOLUME_FRACTION_GEOMETRY). An AMMGID represents a physical material group which is treated as one material entity (represented by 1 material color contour in LS-PREPOST plotting). AMMGID is used in dealing with multiple ALE or Eulerian materials. For example, it can be used to specify a master ALE group in a coupling card.

Example 1:

Consider a purely Eulerian model containing 3 containers containing 2 different physical materials (fluids 1 and 2). All surrounded by the background material (maybe air). The containers are made of the same material, say, metal. Assume that these containers explode and spill the fluids. We want to track the flow and possibly mixing of the various materials. Note that all 7 parts have ELFORM=11 in their *SECTION_SOLID cards. So we have total of 7 PIDs, but only 4 different physical materials.



Approach 1: If we want to track only the interfaces of the **physical** materials.

```
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|....8
*SET_PART
         1
        11
*SET_PART
         2
        22         33
*SET_PART
         3
        44         55         66
*SET_PART
         4
        77
*ALE_MULTI-MATERIAL_GROUP
         1         0    ⇐ 1st line = 1st AMMG ⇒ AMMGID=1
         2         0    ⇐ 2nd line = 2nd AMMG ⇒ AMMGID=2
         3         0    ⇐ 3rd line = 3rd AMMG ⇒ AMMGID=3
         4         0    ⇐ 4th line = 4th AMMG ⇒ AMMGID=4
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|....8
```

With this approach, we define only 4 AMMGs (NALEGP=4). So in LS-PREPOST, when plotting the material-group (history variable) contours, we will see 4 colors, one for each material group. One implication is that when the fluids from part 22 and part 33 flow into the same element, they will coalesce and no boundary distinction between them is maintained subsequently. While this may be acceptable for fluids at similar thermodynamic states, this may not be intuitive for solids. For example, if the solid container materials from parts 44, 55 and 66 flow into one element, they will coalesce "like a single fluid", and no interfaces among them are tracked. If this is undesirable, an alternate approach may be taken. It is presented next.

Approach 2: If we want to reconstruct as many interfaces as necessary, in this case, we follow the interface of each part.

```
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|....8
*ALE_MULTI-MATERIAL_GROUP
         1         1    ⇐ 1st line = 1st AMMG ⇒ AMMGID=1
         2         1    ⇐ 2nd line = 2nd AMMG ⇒ AMMGID=2
         3         1    ⇐ 3rd line = 3rd AMMG ⇒ AMMGID=3
         4         1    ⇐ 4th line = 4th AMMG ⇒ AMMGID=4
         5         1    ⇐ 5th line = 5th AMMG ⇒ AMMGID=5
         6         1    ⇐ 6th line = 6th AMMG ⇒ AMMGID=6
         7         1    ⇐ 7th line = 7th AMMG ⇒ AMMGID=7
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|....8
```

There are 7 AMMGs in this case (NALEGP=7). This will involve more computational cost for the additional tracking. Realistically, accuracy will be significantly reduced if there are more than 3 or 4 materials in any one element. In that case, higher mesh resolution may be required.

Example 2:

OIL	WATER	AIR
GROUP 1	GROUP 2	GROUP 3
PART ID'S 1 AND 2	PART ID 3	PART ID'S 5, 6, AND 7

The above example defines a mixture of three groups of materials (or “fluids”), oil, water and air, that is, the number of ALE multi-material groups (AMMGs) NALEGP=3.

The first group contains two parts (materials), part ID's 1 and 2.

The second group contains one part (material), part ID 3.

The third group contains three parts (materials), part ID's 5, 6 and 7.

*ALE_REFERENCE_SYSTEM_CURVE

Purpose: This command defines a motion and/or a deformation prescribed for a geometric entity (where a geometric entity may be any part, part set, node set, or segment set). The motion or deformation may be completely defined by 12 parameters (shown in the equation below). These 12 parameters are defined in terms of 12 load curves. This command is required only when PRTYPE=3 in the *ALE_REFERENCE_SYSTEM_GROUP (ARSG) command.

Card 1

Variable	ID							
Type	I							
Default	none							

Card 2

Variable	LCID1	LCID2	LCID3	LCID4	LCID5	LCID6	LCID7	LCID8
Type	I	I	I	I	I	I	I	I
Default	none	none	none	none	none	none	none	none

Card 3

Variable	LCID9	LCID10	LCID11	LCID12				
Type	I	I	I	I				
Default	none	none	none	none				

VARIABLE	DESCRIPTION
ID	Curve group ID.
LCID1...LCID12	Load curve ID's.

Remarks:

1. The velocity of a node at coordinate (x, y, z) is defined as:

$$\begin{Bmatrix} \dot{x} \\ \dot{y} \\ \dot{z} \end{Bmatrix} = \begin{Bmatrix} f_1 \\ f_5 \\ f_9 \end{Bmatrix} + \begin{bmatrix} f_2 & f_3 & f_4 \\ f_6 & f_7 & f_8 \\ f_{10} & f_{11} & f_{12} \end{bmatrix} \begin{Bmatrix} x \\ y \\ z \end{Bmatrix}$$

$f_1(t)$ is the value of load curve LCID1 at time t etc. Note that $f_1(t)$, $f_5(t)$, $f_9(t)$ correspond to the translation components in global x, y, and z direction, respectively. $f_2(t)$, $f_7(t)$, and $f_{12}(t)$ correspond to the expansion or contraction component. The remaining functions give rotation contribution.

Example 1:

Consider a motion that consists of translation in the x and y direction only. Thus only $f_1(t)$ and $f_5(t)$ are required. Hence only 2 load curve ID's need be defined:

```
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|...8
*ALE_REFERENCE_SYSTEM_GROUP
$       SID     STYPE      PRTYP       PRID     BCTRAN      BCEXP      BCROT     ICOORD
          1         0          3         11          0          7          0
$        XC        YC         ZC    EXPLIM
          0         0          0          0
*ALE_REFERENCE_SYSTEM_CURVE
$ CURVESID
         11
$     LCID1      LCID2      LCID3      LCID4      LCID5      LCID6      LCID7     LCID8
        111          0          0          0        222          0          0         0
$     LCID9     LCID10     LCID11     LCID12
          0          0          0          0
*DEFINE_CURVE
$      lcid      sidr       sfa        sfo       offa       offo     dattyp
        111
$                  a1                    o1
                 0.00                   5.0
                 0.15                   4.0
*DEFINE_CURVE
$      lcid      sidr       sfa        sfo       offa       offo     dattyp
        222
$                  a1                    o1
                 0.00                  -1.0
                 0.15                  -5.0
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|....8
```

*ALE_REFERENCE_SYSTEM_GROUP

Purpose: This card is used to associate a geometric entity to a reference system type. A geometric entity may be any part, part set, node set, or segment set of a model (or a collection of meshes). A reference system type refers to the possible transformation allowed for a geometric entity (or mesh). This command defines the type of reference system or transformation that a geometric entity undergoes. In other words, it prescribes how certain mesh can translate, rotate, expand, contract, or be fixed in space, etc.

Card 1

Variable	SID	STYPE	PRTYPE	PRID	BCTRAN	BCEXP	BCROT	ICR/NID
Type	I	I	I	I	I	I	I	I
Default	none	0	0	0	0	0	0	0

Card 2

Variable	XC	YC	ZC	EXPLIM	EFAC		FRCPAD	IEXPND
Type	F	F	F	F	F		F	I
Default	0.0	0.0	0.0	inf.	0.0		0.1	0

Card 3 is optional

Card 3

Variable	IPIDXCL	IPIDTYP						
Type	I	I						
Default	0	0						

VARIABLE	DESCRIPTION
SID	Set ID.
STYPE	Set type: EQ.0: part set, EQ.1: part, EQ.2: node set, EQ.3: segment set.
PRTYPE	Reference system type (See Remark 1 below) EQ.0: Eulerian, EQ.1: Lagrangian, EQ.2: Normal ALE mesh smoothing, EQ.3: Prescribed motion following load curves, see *ALE_REFERENCE_ SYSTEM_CURVE, EQ.4: Automatic mesh motion following mass weighted average velocity in ALE mesh, EQ.5: Automatic mesh motion following a local coordinate system defined by three user defined nodes, see *ALE_REFERENCE_SYSTEM_NODE, EQ.6: Switching in time between different reference system types, see *ALE_REFERENCE_SYSTEM_SWITCH, EQ.7: Automatic mesh expansion in order to enclose up to twelve user defined nodes, see *ALE_REFERENCE_SYSTEM_NODE. EQ.8: Mesh smoothing option for shock waves, where the element grid contracts in the vicinity of the shock front. This may be referred to as the Delayed-ALE option. It controls how much the mesh is to be moved during the remap step. This option requires the definition of the 5th parameter in the 2nd card, EFAC; see below for definition. EQ.9: Allowing the ALE mesh(es) to: -Translate and/or rotate to follow a local Lagrangian reference coordinate system (whose *ALE_REFERENCE_SYSTEM_NODE card ID is defined by the BCTRAN parameter) -Expand or contract to enclose a Lagrangian part-set ID defined by the PRID parameter. -Has a Lagrangian node ID be defined by the ICR/NID parameter to be the center of the ALE mesh expansion.
PRID	A parameter giving additional information depending on the reference system (PRTYPE) choice: PRTYPE= 3: PRID defines a load curve group ID specifying an *ALE_REFERENCE_SYSTEM_CURVE card for mesh translation. This defines up to 12 curves which prescribe the motion of the system.

VARIABLE	DESCRIPTION
	PRTYPE= 5: PRID defines a node group ID specifying an *ALE_REFERENCE_SYSTEM_NODE card, via which, three nodes forming a local coordinate system are defined.
	PRTYPE= 6: PRID defines a switch list ID specifying an *ALE_REFERENCE_SYSTEM_SWITCH card. This defines the switch times and the reference system choices for each time interval between the switches.
	PRTYPE= 7: PRID defines a node group ID specifying an *ALE_REFERENCE_SYSTEM_NODE card. Up to 12 nodes in space forming a region to be enveloped by the ALE mesh are defined.
	PRTYPE= 9: PRID defines a Lagrangian part set ID (PSID) defining the Lagrangian part(s) whose range of motion is to be enveloped by the ALE mesh(es). This is useful for airbag modeling.
BCTRAN	For PRTYPE 4 & 5: BCTRAN is a translational constraint (remark 3). EQ.0: no constraints, EQ.1: constrained x translation, EQ.2: constrained y translation, EQ.3: constrained z translation, EQ.4: constrained x and y translation, EQ.5: constrained y and z translation, EQ.6: constrained z and x translation, EQ.7: constrained x, y, and z translation. For PRTYPE= 9: BCTRAN defines a node group ID defined by *ALE_REFERENCE_SYSTEM_NODE card prescribing a local coordinate system (3 node IDs) whose motion is to be followed by the ALE mesh(es).
BCEXP	For PRTYPE= 4 & 7: BCTRAN is an expansion constraint (remark 3). EQ.0: no constraints, EQ.1: constrained x expansion, EQ.2: constrained y expansion, EQ.3: constrained z expansion, EQ.4: constrained x and y expansion, EQ.5: constrained y and z expansion, EQ.6: constrained z and x expansion, EQ.7: constrained x, y, and z expansion.

VARIABLE	DESCRIPTION
BCROT	For PRTYPE= 4: BCROT is a rotational constraint (remark 3). EQ.0: no constraints, EQ.1: constrained x rotation, EQ.2: constrained y rotation, EQ.3: constrained z rotation, EQ.4: constrained x and y rotation, EQ.5: constrained y and z rotation, EQ.6: constrained z and x rotation, EQ.7: constrained x, y, and z rotation.
ICR/NID	A flag defining the center of mesh expansion and/or rotation (remark 3). PRTYPE=4: ICR is a center of mesh expansion and rotation flag, EQ.0: The center is at center of gravity of the ALE mesh. EQ.1: The center is at (XC, YC, ZC), just a point in space (it does not have to be a defined node) PRTYPE=9: NID (node ID) is a Lagrangian NID. LSDYNA uses this node as an anchored center of ALE mesh expansion (remark 2).
XC,YC,ZC	Center of mesh expansion if PRTYPE= 4. This supplements the ICR parameter above.
EXPLIM	Limit ratio for mesh expansion and contraction. Each Cartesian direction is treated separately. The distance between the nodes is not allowed to increase by more than a factor EXPLIM, or decrease to less than a factor 1/EXPLIM. This flag applies only for PRTYPE=4.
EFAC	Initial mesh remapping factor for PRTYPE=8 only, ranging between 0.0 and 1.0. When EFAC approaches 1.0, the remapping approaches pure Eulerian behavior. The smaller the value of EFAC, the closer the mesh will initially follow the material flow in the vicinity of a shock front, i.e. approaching Lagrangian behavior. Thus, a very small value might lead to severe mesh distortions because the mesh must deform severely to follow the material flow initially. Eventually over time, the mesh smoothing behavior will approach an Eulerian system.
FRCPAD	For PRTYPE=9: This is an ALE mesh padding fraction ranging from 0.01 to 0.2. If the characteristic Lagrange mesh dimension (dL_L) exceeds (1-2* FRCPAD) times the characteristic ALE mesh dimension (dL_A), then the ALE mesh is expanded so that $dL_A = dL_L$ /1-2* FRCPAD). This provides extra few layers of ALE elements beyond the maximum Lagrangian range of motion. EQ.0.01: $dL_A = dL_L / 0.98 = dL_L * 1.020408$ EQ.0.20: $dL_A = dL_L / 0.60 = dL_L * 1.666667$

VARIABLE	DESCRIPTION
IEXPND	If PRTYPE=9: This is an ALE mesh expansion control flag. EQ.0: Both mesh expansion and contraction are allowed. EQ.1: Only mesh expansion is allowed.
IPIDXCL	An ALE set ID to be excluded from the expansion and/or contraction only. Translation and rotation are allowed. For example, this may be used to prevent the ALE mesh (or part) at the inflator gas inlet region from expanding too much. High ALE mesh resolution is usually required to resolve the high speed flow of the gas into the airbag via point sources (remark 2).
IPIDTYPE	Set ID type of IPIDXCL: 0 = PSID; 1 = PID

Remarks:

1. Some PRTYP may require a supplemental definition defined via corresponding PRID. For example, PRTYP=3 requires a *ALE_REFERENCE_ SYSTEM_CURVE card. If PRID=n, then in the corresponding *ALE_REFERENCE_SYSTEM_CURVE card, ID=n. Similar association applies for any PRTYP (i.e. 3, 5, 6, or 7) which requires a definition for its corresponding PRID parameter.

2. For PRTYPE=9: ICR/NID can be useful to keep a high density ALE mesh centered on the region of greatest interest, (such as the inflator orifices region in an airbag model). For example, in the case of nonsymmetrical airbag deployment, assuming that the ALE mesh is initially finer near the inlet orifices, and gradually coarsened away from it. Defining an "anchor node" at the center of the orifice location will keep the fine ALE mesh region centered on the orifice region. So that this fine ALE mesh region will not be shifted away (from the point sources) during expansion and translation. The ALE mesh can move and expand outward to envelop the Lagrangian airbag in such a way that the inlet is well resolved throughout the deployment.

3. The table below shows the applicability of the various choices of PRTYPE. Simple deductions from the functional definitions of the PRTYPE choices will clarify the applications of the various constraints. For example, when PRTYP=3, nodal motion of the ALE mesh is completely controlled by the 12 curves. Therefore, no constraints are needed.

PRTYPE	ICR/NID	BCTRAN	BCROT	BCEXP
3	NO	NO	NO	NO
4	YES (ICR)	YES	YES	YES
5	NO	YES	NO	NO
6	NO	NO	NO	NO
7	NO	NO	NO	YES
8	NO	NO	NO	NO
9	YES (NID)	NO	NO	NO

Example 1:

Consider a bird-strike model containing 2 ALE parts: a bird is surrounded by air (or void). A part-set ID 1 is defined containing both parts. To allow for the meshes of these 2 parts to move with their combined mass-weighted-average velocity, PRTYPE=4 is used. Note that BCEXP=7 indicating mesh expansion is constrained in all global directions.

```
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|...8
*ALE_REFERENCE_SYSTEM_GROUP
$       SID     STYPE      PRTYP       PRID     BCTRAN      BCEXP      BCROT     ICOORD
          1         0          4          0          0          7          0
$        XC        YC         ZC     EXPLIM
          0         0          0          0
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|...8
```

Example 2:

Consider a bouncing ball model containing 2 ALE parts: a solid ball (PID 1) is surrounded by air or void (PID 2). A part-set ID 1 is defined containing both parts. To allow for the meshes of these 2 parts to move with 2 reference system types: (a) first, they move with their combined mass-weighted-average velocity between 0.0 and 0.01 second; and subsequently (between 0.01 and 10.0 seconds) their reference system is switched to (b) an Eulerian system (thus the mesh is fixed in space), a reference system "SWITCH" is required. This is done by setting PRTYPE=6. This PRTYPE requires a corresponding *ALE_REFERENCE_SYSTEM_SWITCH card. Note that PRID=11 in the *ALE_REFERENCE_SYSTEM_GROUP card corresponds to the SWITCHID=11 in *ALE_REFERENCE_SYSTEM_SWITCH card.

```
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|....8
*ALE_REFERENCE_SYSTEM_GROUP
$       SID     STYPE     PRTYP       PRID     BCTRAN      BCEXP      BCROT     ICOORD
          1         0         6         11          0          7          7
$        XC        YC        ZC    EXPLIM   EULFACT SMOOTHVMX
          0         0         0         0       0.0
*ALE_REFERENCE_SYSTEM_SWITCH
$ SWITCHID
         11
$        t1        t2        t3        t4        t5        t6        t7
      0.01      10.0
$     TYPE1     TYPE2     TYPE3     TYPE4     TYPE5     TYPE6     TYPE7     TYPE8
          4         0
$       ID1       ID2       ID3       ID4       ID5       ID6       ID7       ID8
          0         0         0         0         0         0         0         0
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|....8
```

*ALE_REFERENCE_SYSTEM_NODE

Purpose: This command defines a group of nodes that control the motion of an ALE mesh. It is used only when PRTYPE=5 or 7 in a corresponding *ALE_ REFERENCE_SYSTEM_GROUP card.

Card 1

Variable	ID							
Type	I							
Default	none							

Card 2

Variable	NID1	NID2	NID3	NID4	NID5	NID6	NID7	NID8
Type	I	I	I	I	I	I	I	I
Default	none	none	none	none	none	none	none	none

Card 3

Variable	NID9	NID10	NID11	NID12				
Type	I	I	I	I				
Default	none	none	none	none				

VARIABLE	DESCRIPTION
ID	Node group ID for PRTYPE 5 or 7, see *ALE_REFERENCE_SYSTEM_GROUP.
NID1...NID12	User specified nodes.

Remarks:

1. For PRTYPE=5 the ALE mesh is forced to follow the motion of a coordinate system, which is defined by three nodes (NID1, NID2, NID3). These nodes are located at x_1, x_2 and x_3, respectively. The axes of the coordinate system, x', y' and z', are defined as:

$$x' = (x_2 - x_1)/|x_2 - x_1|$$
$$z' = x' \times (x_3 - x_1)/|x' \times (x_3 - x_1)|$$
$$y' = z' \times x'$$

   Note that $x_1 \rightarrow x_2$ is the local x' axis, $x_1 \rightarrow x_3$ is the local y' axis and x' crosses y' gives the local z' axis. These 3 nodes are used to locate the reference system at any time. Therefore, their positions relative to each other should be as close to an orthogonal system as possible for better transformation accuracy of the ALE mesh.

2. For PRTYPE=7, the ALE mesh is forced to move and expand, so as to enclose up to twelve user defined nodes (NID1...NID12). This is a rarely used option.

Example 1:

Consider modeling sloshing of water inside a rigid tank. Assuming there are 2 ALE parts, the water (PID 1) and air or void (PID 2) contained inside a rigid (Lagrangian) tank (PID 3). The outer boundary nodes of both ALE parts are merged with the inner tank nodes. A part-set ID 1 is defined containing both ALE parts (PIDs 1 and 2). To allow for the meshes of the 2 ALE parts to move with the rigid Lagrangian tank, PRTYPE=5 is used. The motion of the ALE parts then follows 3 reference nodes on the rigid tank. These 3 reference nodes must be defined by a corresponding *ALE_REFERENCE_SYSTEM_NODE card. In this case the reference nodes have the nodal IDs of 5, 6 and 7. Note that PRID=12 in the

*ALE_REFERENCE_SYSTEM_GROUP card corresponds to the SID=12 in the *ALE_REFERENCE_SYSTEM_NODE card.

```
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|...8
*ALE_REFERENCE_SYSTEM_GROUP
$       SID     STYPE      PRTYP       PRID     BCTRAN      BCEXP      BCROT     ICOORD
          1         0          5         12
$        XC        YC         ZC    EXPLIM
          0         0          0          0
*ALE_REFERENCE_SYSTEM_NODE
$      NSID
          12
$        N1        N2         N3         N4         N5         N6         N7         N8
          5         6          7
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|...8
```

*ALE_REFERENCE_SYSTEM_SWITCH

Purpose: The PRTYPE parameter in the *ALE_REFERENCE_SYSTEM_GROUP (ARSG) card allows many choices of the reference system types for any ALE geometric entity. This command allows for the time-dependent switches between these different types of reference systems, i.e., switching to multiple PRTYPEs at different times during the simulation. This command is required only when PRTYPE=6 in ARSG card. Please see example 2 in the ARSG section.

Card 1

Variable	ID							
Type	I							
Default	none							

Card 2

Variable	T1	T2	T3	T4	T5	T6	T7	
Type	F	F	F	F	F	F	F	
Default	0.0	0.0	0.0	0.0	0.0	0.0	0.0	

Card 3

Variable	TYPE1	TYPE2	TYPE3	TYPE4	TYPE5	TYPE6	TYPE7	TYPE8
Type	I	I	I	I	I	I	I	I
Default	0	0	0	0	0	0	0	0

Card 4	1	2	3	4	5	6	7	8
Variable	ID1	ID2	ID3	ID4	ID5	ID6	ID7	ID8
Type	I	I	I	I	I	I	I	I
Default	none	none	none	none	None	none	none	none

VARIABLE	DESCRIPTION
ID	Switch list ID, see *ALE_REFERENCE_SYSTEM_GROUP,
T1...T7	Times for switching reference system type. By default, the reference system TYPE1 occurs between time=0 and time=T1, and TYPE2 occurs between time=T1 and time=T2, etc.
TYPE1...TYPE8	Reference system types (also see PRTYPE under ARSG): EQ.0: Eulerian, EQ.1: Lagrangian, EQ.2: Normal ALE mesh smoothing, EQ.3: Prescribed motion following load curves, see *ALE_REFERENCE_SYSTEM_CURVE, EQ.4: Automatic mesh motion following mass weighted average velocity in ALE mesh, EQ.5: Automatic mesh motion following a local coordinate system defined by three user defined nodes, see *ALE_REFERENCE_SYSEM_NODE,
ID1...ID8	The corresponding PRID parameters supporting each PRTYPE used during the simulation.

Remarks:

1. The beginning time is assumed to be t=0, and the starting PRTYPE is TYPE1. So at T1, the 1st switching time, PRTYPE is switched from TYPE1 to TYPE2, and so forth. This option can be complex in nature so it is seldom applied.

*ALE_SMOOTHING

Purpose: This smoothing constraint keeps a node at its initial parametric location along a line between two other nodes. This constraint is active during each mesh smoothing operation.

Card 1	1	2	3	4	5	6	7	8
Variable	SNID	MNID1	MNID2	IPRE	XCO	YCO	ZCO	
Type	I	I	I	I	F	F	F	
Default	none	none	none	0	0.0	0.0	0.0	

VARIABLE	DESCRIPTION
SNID	Slave node ID, see Figure 2.1.
MNID1	First master node ID.
MNID2	Second master node ID.
IPRE	EQ.0: smoothing constraints are performed after mesh relaxation, EQ.1: smoothing constraints are performed before mesh relaxation.
XCO	x-coordinate of constraint vector
YCO	y-coordinate of constraint vector
ZCO	z-coordinate of constraint vector

Remarks:

1. Arbitrary Lagrangian Eulerian meshes are defined via the choice of the element type and the *CONTROL_ALE card. This can only be used with solid elements.

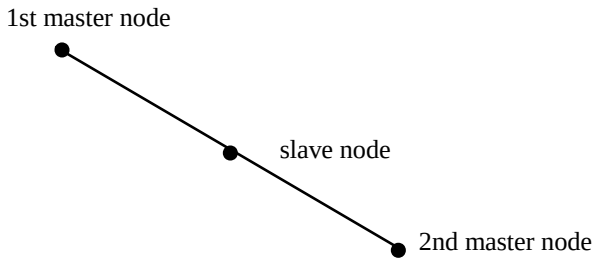


Figure 2.1 This simple constraint, which ensures that a slave node remains on a straight line between two master nodes, is sometimes necessary during ALE smoothing.

*ALE_TANK_TEST

Purpose: This command allows for the airbag information input $\left(\dot{m}(t), \bar{T}_{gas}(t)\right)$ of the control volume (*AIRBAG_) approach to be used as input for the ALE/Eulerian fluid-structure interaction model of the airbag. It complements and must be used together with the*SECTION_POINT_SOURCE command. Please see *SECTION_POINT_SOURCE for additional information.

Card 1

Variable	MDOTLC	TANKV	PAMB	PFINAL	MACHL	VELMAX	AORIF	
Type	I	I	I	I	F	F	F	
Default	0	0.0	0.0	0.0	0.0	0.0	0.0	

Card 2

Variable	AMGIDG	AMGIDA	NUMPNT					
Type	I	I	I					
Default	0	0	50					

VARIABLE	DESCRIPTION
MDOTLC	LCID for mass flow rate as a function of time. This may be obtained directly from the control-volume type input data.
TANKV	Volume of the tank used in a tank test from which the tank pressure is measured, and $\dot{m}(t)$ and $\bar{T}_{gas}(t)$ are computed from this tank pressure data.
PAMB	The pressure inside the tank before jetting (usually 1bar).
PFINAL	The final equilibrated pressure inside the tank from the tank test.
MACHL	A limiting MACH number for the gas at the throat (MACH=1 preferred).

VARIABLE	DESCRIPTION
VELMAX	Maximum allowable gas velocity across the inflator orifice (not preferred).
AORIF	Total inflator orifice area (optional, only needed if the *SECTION_ POINT_SOURCE card is not used).
AMGIDG	The ALE multi-material group ID (AMMGID) of the gas.
AMGIDA	The ALE multi-material group ID (AMMGID) of the air.
NUMPNT	The number of points in $\dot{m}(t)$ and $\bar{T}_{gas}(t)$ curves. If NUMPNT=0, defaults to 50 points.

Remarks:

1. In an airbag inflator tank test, the tank pressure data is measured. This pressure is used to derive $\dot{m}(t)$ and the estimated $\bar{T}_{gas}(t)$, usually via a lumped-parameter method, a system of conservation equations and EOS. These 2 curves are used as the direct input for the control volume method in LS-DYNA via the *AIRBAG_ cards. Typically, $\bar{T}_{gas}(t)$ is the stagnation temperature of the incoming inflator gas. In an ALE or Eulerian fluid-structure interaction analysis, the gas velocity, $vel(t)$, and density, $\rho(t)$, at the inlet must be computed. Since only $\dot{m}(t)$ is known, additional assumptions about the inlet condition must be made to compute both $vel(t)$ and $\rho(t)$ curves from the information available. If this computation is done outside of LS-DYNA, then $\dot{m}(t)$ and $\bar{T}_{gas}(t)$ are used to compute 3 curves which are then used as the input for the ALE model: $\bar{T}_{gas\_corrected}(t)$, $vel(t)$ and $\rho(t)$. This *ALE_TANK_TEST card allows for this inlet condition conversion to be done inside LS-DYNA. Thus, with this card together with the *SECTION_POINT_SOURCE card, LS-DYNA can take in directly the control volume input ($\dot{m}(t)$ and $\bar{T}_{gas}(t)$) and performs an ALE or Eulerian fluid-structure interaction analysis. The users do not have to do the conversion themselves.

If the *ALE_TANK_TEST card is present:

2. The definitions of the relative volume, $v_r(t)$ and $vel(t)$ curves in the *SECTION_POINT_SOURCE card will be ignored. They are computed internally inside LS-DYNA.

3. The $\dot{m}(t)$ curve will be read in on *ALE_TANK_TEST card.

4. The $\bar{T}_{gas}(t)$ curve (stagnation temperature) will be read in on *SECTION_POINT_SOURCE card (not $\bar{T}_{gas\_corrected}(t)$). A fine distinction between the two temperatures may be made. $\bar{T}_{gas}(t)$ is derived directly from the tank pressure data

based on a lump-parameter approach. $\bar{T}_{gas\_corrected}(t)$ is computed from $\dot{m}(t)$ and $\bar{T}_{gas}(t)$ with additional isentropic and sonic flow assumption for the maximum velocity at an orifice ($\bar{T}_{gas\_corrected}(t)$ is the static temperature). These assumptions are necessary since in $\dot{m}(t) = \rho(t) * vel(t) * A$, we only know $\dot{m}(t)$ (1 known) but we need $\rho(t)$ and $vel(t)$ (2 unknowns).

5. The inflator area is computed from the *SECTION_POINT_SOURCE card that has the AMMGID of the inflator gas in the *ALE_TANK_TEST card. If the *BOUNDARY_AMBIENT_EOS card is used instead of the *SECTION_POINT_SOURCE card, then the area may be input in this *ALE_TANK_TEST card.

6. The reference density of the propellant "gas", ρ_0, is computed internally and automatically used for the calculation. The ρ_0 value from the *MAT_NULL card is ignored.

Example:

Consider a tank test model consists of the inflator gas (PID 1) and the air inside the tank (PID 2). The following information from the control volume model is available:

- $\dot{m}(t)$ (LCID 1 is from control volume model input).
- $\bar{T}_{gas}(t)$ (LCID 2 is from control volume model input).
- Volume of the tank used in the inflator tank test.
- Final equilibrated pressure inside the tank.
- Ambient pressure in the air.

Also available are:

- The nodal IDs of the nodes defining the orifice holes through which the gas flows into the tank.
- The area associated with each hole (the node is assumed to be at the center of this area).
- The vector associated with each hole defining the direction of flow.

In the input below LCID 1 and 2 are $\dot{m}(t)$ and $\bar{T}_{gas}(t)$, respectively. LCID 4 and 5 will be ignored when the *ALE_TANK_TEST card is present. If it is not present, all 3 curves in the *SECTION_POINT_SOURCE card will be used. When the *SECTION_POINT_SOURCE card is present, the element formulation is equivalent to an ELFORM=11.

```
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|...8
*PART
inflator gas
$      PID     SECID       MID     EOSID      HGID      GRAV    ADPOPT      TMID
         1         1         1         0         0         0         0         0
*PART
air inside the tank
$      PID     SECID       MID     EOSID      HGID      GRAV    ADPOPT      TMID
         2         2         2         0         0         0         0         0
*SECTION_SOLID
$    SECID    ELFORM       AET
         2        11         0
*ALE_MULTI-MATERIAL_GROUP
$      SID   SIDTYPE
         1         1
         2         1
*SECTION_POINT_SOURCE
$    SECID     LCIDT  LCIDVOLR   LCIDVEL      <= 3 curves in tempvolrvel.k file
         1         2         4         5
$   NODEID    VECTID      AREA
     24485         3    15.066
       ...
     24557         3    15.066
*ALE_TANK_TEST
$   MDOTLC     TANKV      PAMB    PFINAL     MACHL    VELMAX     AORIF
         1     6.0E7    1.0E-4  5.288E-4       1.0       0.0
$   AMGIDG    AMGIDA    NUMPNT
         1         2        80
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|...8
```

*ALE_UP_SWITCH

Purpose: For the simulation of airbag inflation process, this card allows the switching from an ALE computation to a control volume (CV) or uniform pressure (UP) method at a user-defined switch time.

Card 1

Variable	UPID	SWTIME						
Type	I	F						
Default	0	1.0e+16						
Remarks	1							

Card 2

Variable	FSI_ID1	FSI_ID2	FSI_ID3	FSI_ID4	FSI_ID5	FSI_ID6	FSI_ID7	FSI_ID8
Type	I	I	I	I	I	I	I	I
Default	0	0	0	0	0	0	0	0
Remarks								

Optional Card 3 Format – input only if UPID=0 or not defined.

Card 3	1	2	3	4	5	6	7	8
Variable	SID	SIDTYPE	MMGAIR	MMGGAS				
Type	I	I	I	I				
Default	0	0	0	0				
Remarks								

VARIABLE	DESCRIPTION
UPID	An ID defines a corresponding *AIRBAG_HYBRID_ID card for use in an ALE-method-switching-to-CV-method simulation. The simulation starts with ALE computational method, then switches to a CV (or UP) method at some given time. EQ.0: (or blank) The code will construct an equivalent *AIRBAG_HYBRID_ID card automatically internally, (default). The 3rd optional line is then a required input. NE.0: An ID points to a corresponding *AIRBAG_HYBRID_ID card which must be defined for use after the switch. If UPID is defined, do not define the 3rd optional card.
SWTIME	The time at which the computation does a switch from an ALE-method-to-CV-method.
FSI_ID1→FSI_ID8	Coupling IDs for one or more ALE fluid-structure-interaction (FSI) *CONSTRAINED_LAGRANGE_IN_SOLID_ID cards. These couplings are deleted during the 2nd, CV computational phase.
SID	A set ID defines the Lagrangian parts which make up the airbag.
SIDTYPE	Set ID type for the above SETID (following the conventions in *AIRBAG_HYBRID card). EQ.0: SID is a segment set ID (SGSID). NE.0: SID is a part set ID (PSID).
MMGAIR	The AMMG (ALE multi-material group) ID of surrounding air.
MMGGAS	The AMMG ID of inflator gas injected into the airbag.

Remarks:

1. If UPID is zero or blank, optional card 3 must be defined. LSDYNA will construct an equivalent *AIRBAG_HYBRID_ID card automatically.

Example 1:

Consider an airbag model with a 2-phase simulation: an ALE calculation being switched to a CV method. During the CV phase, the simulation is defined by an *AIRBAG_HYBRID_ID card.

```
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|....8
*ALE_UP_SWITCH
$    UP_ID   SW_time
    100000    2.0000
$ FSI_ID_1  FSI_ID_2  FSI_ID_3  FSI_ID_4  FSI_ID_5  FSI_ID_6  FSI_ID_7  FSI_ID_8
         1         2
$--------------------------------------------------------------------------------
*AIRBAG_HYBRID_ID
$       ID
    100000
$      SID    SIDTYP      RBID      VSCA      PSCA      VINI       MWD      SPSF
         2         1         0       1.0       1.0       0.0       0.0       0.0
$ 2   ATMT      ATMP      ATMD        GC        CC
      293. 1.0130e-4  1.200E-9    8.3143        1.
$      C23     LCC23       A23     LCA23      CP23     LCP23      AP23    LCAP23

$      OPT     PVENT      NGAS
                             4
$bac LCIDM     LCIDT   NOTUSED        MW     INITM         A         B         C
      1001      1002           0.0288691       1.0     28.98
$    FMASS

$air LCIDM     LCIDT   NOTUSED        MW     INITM         A         B         C
      1600      1603          28.97E-3       0.0     26.38  8.178e-3 -1.612e-6
$    FMASS

$pyroLCIDM     LCIDT   NOTUSED        MW     INITM         A         B         C
      1601      1603          43.45E-3       0.0     32.87  2.127e-2 -5.193E-6
$    FMASS

$sto_LCIDM     LCIDT   NOTUSED        MW     INITM         A         B         C
      1602      1603          39.49E-3       0.0     22.41  2.865e-3 -6.995e-7
$    FMASS
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|....8
```

Example 2:

Consider the same airbag model with the same 2-phase simulation. However, all the *AIRBAG_HYBRID_ID card definitions are extracted automatically from the ALE model. There is no need to define the *AIRBAG_HYBRID_ID card. The 3rd optional card is required.

```
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|....8
*ALE_UP_SWITCH
$    UP_ID   SW_time
$   100000    2.0000
         0    2.0000
$ FSI_ID_1  FSI_ID_2  FSI_ID_3  FSI_ID_4  FSI_ID_5  FSI_ID_6  FSI_ID_7  FSI_ID_8
         1         2
$    SETID    SETYPE   MMG_AIR   MMG_GAS
         2         1         2         1
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|....8
```

*BOUNDARY

The keyword *BOUNDARY provides a way of defining imposed motions on boundary nodes. The keyword control cards in this section are defined in alphabetical order:

***BOUNDARY_ACOUSTIC_COUPLING**

***BOUNDARY_ALE_MAPPING**

***BOUNDARY_AMBIENT_EOS**

BOUNDARY_CONVECTION_OPTION*

***BOUNDARY_CYCLIC**

***BOUNDARY_ELEMENT_METHOD_ACOUSTIC**

BOUNDARY_ELEMENT_METHOD_OPTION*

BOUNDARY_FLUX_OPTION*

***BOUNDARY_FREE_FIELD_GROUND_MOTION** *OPTION1*}_{*OPTION2*}

***BOUNDARY_MCOL**

***BOUNDARY_NON_REFLECTING**

***BOUNDARY_NON_REFLECTING_2D**

BOUNDARY_PORE_FLUID_OPTION*

***BOUNDARY_PRESCRIBED_ACCELEROMETER_RIGID**

***BOUNDARY_PRESCRIBED_FINAL_GEOMETRY**

***BOUNDARY_PRESCRIBED_MOTION_**{*OPTION1*}_{*OPTION2*}

BOUNDARY_PRESCRIBED_ORIENTATION_RIGID_OPTION*

BOUNDARY_PRESSURE_OUTFLOW_OPTION*

BOUNDARY_PWP_OPTION*

BOUNDARY_RADIATION_OPTION*

***BOUNDARY_SLIDING_PLANE**

***BOUNDARY_SPC_**{*OPTION1*}_{*OPTION2*}_{*OPTION3*}

***BOUNDARY_SPH_FLOW**

***BOUNDARY_SPH_SYMMETRY_PLANE**

***BOUNDARY_SYMMETRY_FAILURE**

BOUNDARY_TEMPERATURE_OPTION*

***BOUNDARY_THERMAL_WELD**

***BOUNDARY_USA_SURFACE**

***BOUNDARY_ACOUSTIC_COUPLING**

Purpose: Define a segment set for acoustic coupling. The segments should define the surface of a shell or solid (structural) element. This option allows for acoustic elements (type 8 solid elements) to couple on either one side of a shell or solid element structure or both sides of a shell structure. The nodal points of the shell segments and those on either side of the segments must be coincident. If the fluid exists on just one side of the segment and if the nodes are merged, no input is necessary and input data in this section is not needed. Two-sided coupling will not work if the interface nodes are merged out.

Card	1	2	3	4	5	6	7	8
Variable	SSID							
Type	I							
Default	none							

VARIABLE	DESCRIPTION
SSID	Segment set ID, see *SET_SEGMENT

Remarks:

For the stability of the acoustic-structure coupling, the following condition must be satisfied:

$$\frac{2\rho_a D}{\rho_s t_s} < 5$$

where ρ_a is the density of the acoustic medium, D is the total thickness of the acoustic elements adjacent to the structural element, ρ_s is the density, and t_s is the thickness of the structural shell element.

*BOUNDARY_ALE_MAPPING

Purpose: This card maps ALE data histories from a previous run to a region of elements. Data are read or written in a mapping file called by the prompt "map=" on the command line (see remark 4 and 5). To map data at the initial time (not the histories) to all the ALE domain (not just a region of elements) see *INITIAL_ALE_MAPPING.

Card Format

Card 1

Variable	ID	TYP	AMMSID	IVOLTYP	BIRTH	DEATH	DTOUT	INI
Type	I	I	I	I	F	F	F	I
Default	none	none	none	none	0.0	1e20	time step	0

Card 2

Variable	THICK	RADIUS	X1	Y1	Z1	X2	Y2	Z2
Type	F	F	F	F	F	F	F	F
Default	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0

Card 3

Variable	XO	YO	ZO	VECID				
Type	F	F	F	I				
Default	0.0	0.0	0.0	None				

VARIABLE	DESCRIPTION
ID	Part ID or part set ID or element set ID

TYP	Type of "ID" (see remark 1): EQ.0: part set ID. EQ.1: part ID. EQ.2: shell set ID. EQ.3: solid set ID.
AMMSID	Set ID of ALE multi-material groups defined in *SET_MULTI-MATERIAL_GROUP. See remark 1.
IVOLTYP	Type of volume containing the selected elements for the mapping: EQ.1: Spherical surface with thickness (THICK). EQ.2: Box. EQ.3: Cylindrical surface with thickness (THICK) EQ.4: All the elements defined by ID. If ivoltyp>0, data from the mapping file are read for the elements of this volume. If ivoltyp<0, data from the elements of this volume are written in the mapping file.
BIRTH	Birth time to write or read the mapping file. If a mapping file is written, the next run reading this file will begin at time BIRTH if this parameter for this next run is not larger.
DEATH	Death time to write or read the mapping file. If a mapping file is written, the next run will stop to read this file at time DEATH if this parameter for this next run is not smaller.
DTOUT	Time interval between outputs in the mapping file. This parameter is only used to write in the mapping file.
INI	Flag to initialize all the ALE domain of the next run: EQ.0: No initialization EQ.1: Initialization. *INITIAL_ALE_MAPPING will have to be in the input deck of the next run to read the data from the mapping file. The initial time of the next run will be BIRTH.
THICK	Thickness for the element selection using surfaces.
RADIUS	Radius for abs(ivoltyp)=1 and abs(ivoltyp)=2.
X1	Geometric parameter defined by ivoltyp: If abs(ivoltyp)=1, X1 is the X-coordinate of the sphere center. If abs(ivoltyp)=2, X1 is the X-coordinate of the minimum coordinate of the box. If abs(ivoltyp)=3, X1 is the X-coordinate of a point on the cylinder axis. If abs(ivoltyp)=4, X1 is ignored.
Y1	Geometric parameter defined by ivoltyp:

If abs(ivoltyp)=1, Y1 is the Y-coordinate of the sphere center.
If abs(ivoltyp)=2, Y1 is the Y-coordinate of the minimum coordinate of the box.
If abs(ivoltyp)=3, Y1 is the Y-coordinate of a point on the cylinder axis.
If abs(ivoltyp)=4, Y1 is ignored.

Z1 Geometric parameter defined by ivoltyp:
If abs(ivoltyp)=1, Z1 is the Z-coordinate of the sphere center.
If abs(ivoltyp)=2, Z1 is the Z-coordinate of the minimum coordinate of the box.
If abs(ivoltyp)=3, Z1 is the Z-coordinate of a point on the cylinder axis.
If abs(ivoltyp)=4, Z1 is ignored.

X2 Geometric parameter defined by ivoltyp:
If abs(ivoltyp)=1, X2 is ignored
If abs(ivoltyp)=2, X2 is the X-coordinate of the maximum coordinate of the box.
If abs(ivoltyp)=3, X2 is the X-coordinate of a vector parallel to the cylinder axis.
If abs(ivoltyp)=4, X2 is ignored.

Y2 Geometric parameter defined by ivoltyp:
If abs(ivoltyp)=1, Y2 is ignored
If abs(ivoltyp)=2, Y2 is the Y-coordinate of the maximum coordinate of the box.
If abs(ivoltyp)=3, Y2 is the Y-coordinate of a vector parallel to the cylinder axis.
If abs(ivoltyp)=4, Y2 is ignored.

Z2 Geometric parameter defined by ivoltyp:
If abs(ivoltyp)=1, Z2 is ignored
If abs(ivoltyp)=2, Z2 is the Z-coordinate of the maximum coordinate of the box.
If abs(ivoltyp)=3, Z2 is the Z-coordinate of a vector parallel to the cylinder axis.
If abs(ivoltyp)=4, X2 is ignored.

XO Origin position in global X-direction. See remark 2.

YO Origin position in global Y-direction. See remark 2.

ZO Origin position in global Z-direction. See remark 2.

VECID ID of the symmetric axis defined by *DEFINE_VECTOR. See remark 3.

Remarks:

1. The routines of this card need to know which mesh will be initialized with the mapping data and more specifically which multi-material groups. The first 2 parameters (ID and TYP) defines the mesh and the third one (AMMSID) calls the *SET_MULTI-MATERIAL_GROUP_LIST card. This card will define a list of material groups in the current run. The rank in this list should match the rank of the multi-material groups from the previous run (as a reminder the ranks of multi-material groups are defined by *ALE_MULTI-MATERIAL_GROUP). Let's take an example. For instance if the previous model has 3 groups, the current one has 5 groups and the following mapping is wanted:
   - The 1st group from the previous run is mapped on the 3rd one in the current run,
   - The 2nd group from the previous run is mapped on the 5th one in the current run,
   - The 3rd group from the previous run is mapped on the 4th one in the current run,

   The *SET_MULTI-MATERIAL_GROUP_LIST card should be set as follows:

   ```
   *SET_MULTI-MATERIAL_GROUP_LIST
   300
   3,5,4
   ```

2. The data can be mapped in different parts of the mesh by defining the origin of the coordinate system (XO,YO,ZO).

3. For a mapping between a 2D axisymmetric model and a 3D run the symmetric axis orientation in the 3D model needs to be defined.

4. To make one mapping: only the prompt "map=" is necessary. If IVOLTYP is positive, the mapping file will be created and ALE data histories will be written in this file. If IVOLTYP is negative the mapping file will be read and ALE data histories will be used to interpolate the ALE variables of the selected elements.

5. To make several successive mapping: the prompt "map1=" is necessary. If IVOLTYP is positive and the prompt "map1=" is in the command line, the ALE data are written to the mapping file given by "map1=". If IVOLTYP is negative and the prompt "map=" is in the command line, ALE data are read from the mapping file given by "map=".

***BOUNDARY_AMBIENT_EOS**

Purpose: This command defines the IDs of 2 load curves: (1) internal energy per unit reference specific volume (or temperature if using *EOS_IDEAL_GAS) and (2) relative volume. These 2 curves completely prescribe the thermodynamic state as a function of time for any ALE or Eulerian part with an "ambient" type element formulation (please see Remark 4).

Card	1	2	3	4	5	6	7	8
Variable	PID	LCID1	LCID2					
Type	I	I	I					
Default	none	none	none					

VARIABLE	DESCRIPTION
PID	The ambient Part ID for which the thermodynamic state is being defined.
LCID1	A load curve ID for internal energy per unit reference specific volume (please read the beginning of the EOS section for details). If *EOS_IDEAL_GAS is being used, this ID then refers to a temperature load curve ID.
LCID2	Load curve ID for relative volume, $v_r = \left(\frac{v}{v_0} = \frac{\rho_0}{\rho} \right)$. (Please read the beginning of the EOS section for details).

Remarks:

1. The term "ambient" refers to a medium that has predetermined thermodynamic state throughout the simulation. All "ambient" parts/elements will have its thermodynamic state reset back to this predetermined state every cycle. If this state is defined via the *EOS_ card, then this predetermined thermodynamic state is constant throughout the simulation. If it is defined via this card, *BOUNDARY_AMBIENT_EOS, then its thermodynamic state will vary according to these defined load curves. "Ambient" part is sometimes also referred to as "reservoir" part as it may be used to simulate semi-infinite region.

2. In general, a thermodynamic state of a non-reacting and no-phase-change material may be defined by 2 thermodynamic variables. By defining (a) an internal energy per unit reference specific volume load curve (or a temperature load curve if using

*EOS_IDEAL_GAS) and (b) a relative volume load curve, the pressure as a function of time for this ambient part ID can be computed directly via the equation of state (*EOS_).

3. A reference specific volume, $v_0 = \frac{1}{\rho_0}$, is the inverse of a reference density, ρ_0. The reference density is defined as the density at which the material is under a reference or nominal state. Please refer to the *EOS section for additional explanation on this.

4. The internal energy per unit reference specific volume may be defined as $e_{ipv0} = \frac{C_v T}{v_0}$.

   The specific internal energy (or internal energy per unit mass) is defined as $C_v T$.

5. This card is only to be used with "ambient" element type as defined by the parameters under the *SECTION_SOLID card:
   - ELFORM = 7, or
   - ELFORM = 11 and AET=4, or
   - ELFORM = 12 and AET=4.

Example:

Consider an ambient ALE part ID 1 which has its internal energy per unit reference specific volume in a load curve ID 2 and relative volume load curve ID 3:

```
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|...8
*BOUNDARY_AMBIENT_EOS
$       PID  e/T_LCID rvol_LCID
          1         2         3
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|...8
```

BOUNDARY_CONVECTION_*OPTION

Available options include:

SEGMENT

SET

Purpose: Define convection boundary conditions for a thermal or coupled thermal/structural analysis. Two cards are defined for each option.

For the **SET** option define the following card:

(Card 1 of 2)

Card 1	1	2	3	4	5	6	7	8
Variable	SSID							
Type	I							
Default	none							

For the **SEGMENT** option define the following card:

(Card 1 of 2)

Card 1	1	2	3	4	5	6	7	8
Variable	N1	N2	N3	N4				
Type	I	I	I	I				
Default	none	none	none	none				

Define the following card for both options:

(Card 2 of 2)

Card 2	1	2	3	4	5	6	7	8
Variable	HLCID	HMULT	TLCID	TMULT				
Type	I	F	I	F	I			
Default	none	0.	none	0.	0			

VARIABLE	DESCRIPTION
SSID	Segment set ID, see *SET_SEGMENT.
N1,N2...	Node ID's defining segment.
HLCID	Load curve ID for heat transfer coefficient, *h:* GT.0: function versus time, EQ.0: use constant multiplier value, HMULT, LT.0: function versus temperature.
HMULT	Curve multiplier for *h.*
TLCID	Load curve ID for T_∞ versus time, see *DEFINE_CURVE: EQ.0: use constant multiplier value, TMULT.
TMULT	Curve multiplier for T_∞
LOC	Application of surface for thermal shell elements, see parameter, TSHELL, in the *CONTROL_SHELL input: EQ.-1: lower surface of thermal shell element EQ. 1: upper surface of thermal shell element

.

Remarks:

A convection boundary condition is calculated using $\dot{q}'' = (T - T_\infty)$ where

h heat transfer coefficient

$(T - T_\infty)$ temperature potential

Three alternatives are possible for the heat transfer coefficient which can be a function of time, a function of temperature, or constant. Also, the temperature of the boundary T_∞ can be either constant or a function of time. For both curves, multipliers can be used to scale the values.

*BOUNDARY_CYCLIC

Purpose: Define nodes in boundary planes for cyclic symmetry.

These boundary conditions can be used to model a segment of an object that has rotational symmetry such as an impeller, i.e., Figure 3.1. The segment boundary, denoted as a side 1 and side 2, may be curved or planar. In this section, a paired list of points are defined on the sides that are to be joined.

Card	1	2	3	4	5	6	7	8
Variable	XC	YC	ZC	NSID1	NSID2	IGLOBAL	ISORT	
Type	F	F	F	I	I	I	I	
Default	none	none	none	none	none	0	2	

VARIABLE	DESCRIPTION
XC	x-component axis vector of axis of rotation
YC	y-component axis vector of axis of rotation
ZC	z-component axis vector of axis of rotation
NSID1	Node set ID for first boundary plane (side 1, see Figure 3.1).
NSID2	Node set ID for second boundary plane (side 2, see Figure 3.1). Each boundary node in this boundary plane is constrained to its corresponding node in the first node set. Node sets NSID1 and NSID2 must contain the same number of nodal points. Care has to be taken that the nodes in both node sets have a location which, if given in cylindrical coordinates, all differ by the same angle.
IGLOBAL	Flag for repeating symmetry: EQ.0: Axi-symmetric cyclic symmetry (default) EQ.1: Repeating symmetry in planes normal to global X EQ.2: Repeating symmetry in planes normal to global Y EQ.3: Repeating symmetry in planes normal to global Z
ISORT	Flag for automatic sorting of boundary nodes: EQ.0: No automatic sorting (default) EQ.1: Automatic sorting of nodes.

Remarks:

1. Each node set should lie on a plane, and these should generally be boundaries of the model.

2. Previous versions of LS-DYNA, prior to version 970, it was assumed that the nodes are correctly ordered within each set, i.e. the n^{th} node in NSID1 is equivalent to the n^{th} node in NSID2. If the ISORT flag is active, the nodes in NSID2 are automatically sorted to achieve equivalence, so the nodes can be picked by the quickest available method. However, for axi-symmetric cyclic symmetry (IGLOBAL=0), it is assumed that the axis passes through the origin, i.e., only globally defined axes of rotation are possible.

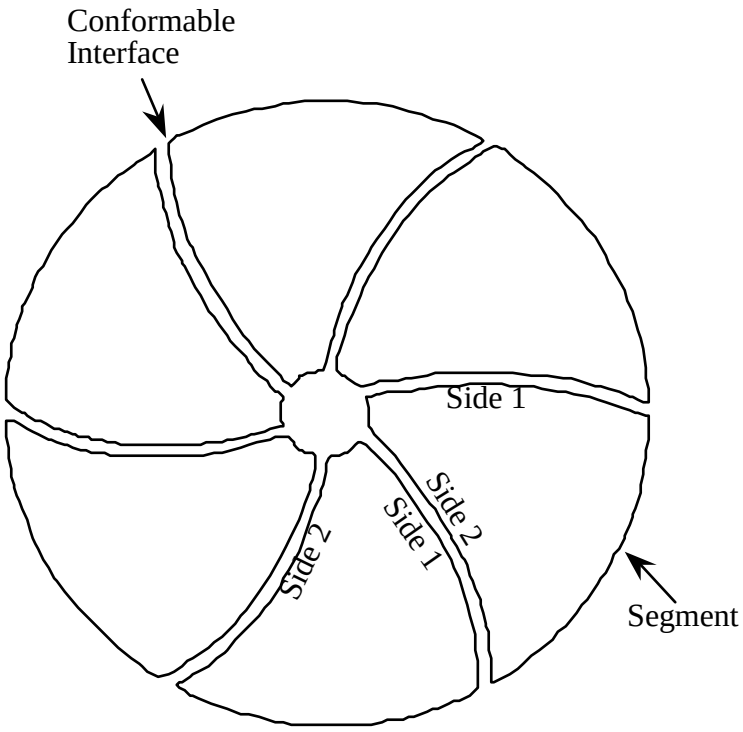


Figure 3.1 With axi-symmetric cyclic symmetry only one segment is modeled.

*BOUNDARY_ELEMENT_METHOD_ACOUSTIC

Purpose: Use boundary element method in frequency domain for acoustic problems. This Keyword is only used if the option BEM = **filename** in the LSDYNA run command is used:

LS-DYNA I=**inf** BEM=**filename**

Card 1

Variable	RO	C	FMIN	FMAX	NFREQ	DTOUT	TSTART	PREF
Type	F	F	F	F	I	F	F	F
Default	none	none	none	none	0	0	0	0
Remark							1	2

Card 2

Variable	NSIDEXT	TYPEXT	NSIDINT	TYPINT	FFTWIN	TRSLT	IPFILE	IUNITS
Type	I	I	I	I	I	I	I	I
Default	0	0	0	0	0	0	0	0
Remark					3	4		5

Card 3

Variable	METHOD	MAXIT	TOLITR	NDD	TOLLR	TOLFCT	IBDIM	NPG
Type	I	I	F	I	F	F	I	I
Default		100	10^{-4}	1	10^{-6}	10^{-6}	1000	2
Remark	6			7				

Card 4	1	2	3	4	5	6	7	8
Variable	SSID	SSTYPE	NORM	BEMTYP	RESTRT	IEDGE		
Type	I	I	I	I	I	I		
Default	0	0	0	0	0	0		
Remark					8	9		

VARIABLE	DESCRIPTION
RO	Fluid density.
C	Sound speed of the fluid.
FMIN	Minimum value of output frequencies.
FMAX	Maximum value of output frequencies.
NFREQ	Number of output frequencies.
DTOUT	Time interval between writing velocity or acceleration, and pressure at boundary elements in the binary file, to be proceeded at the end of LS-DYNA simulation
TSTART	Start time for recording velocity or acceleration in LS-DYNA simulation.
PREF	Reference pressure to be used to output pressure in dB, in the file Press_dB. If PREF=0, the Press_dB file will not be generated. A file called Press_Pa is generated and contains the pressure at the output nodes (see Card 2)
NSIDEXT	Node or segment set ID of output exterior field points.
TYPEXT	Output exterior field point type. EQ.1: node set ID. EQ.2: segment set ID.
NSIDINT	Node or segment set ID of output interior field points.

VARIABLE	DESCRIPTION
TYPINT	Output interior field point type. EQ.1: node set ID. EQ.2: segment set ID.
FFTWIN	FFT windows (Default=0). EQ.0: rectangular window EQ.1: Hanning window EQ.2: Hamming window EQ.3: Blackman window EQ.4: raised cosine window
TRSLT	Request time domain results: EQ.0: no time domain results are requested; EQ.1: time domain results are requested.
IPFILE	Flag for output files (default=0). EQ.0: Press_Pa (magnitude of pressure vs. frequency), Press_dB (sound pressure level vs. frequency) and bepres (ASCII database file for LS-Prepost) are provided. EQ.1: Press_Pa_real (the real part of the pressure vs. frequency) and Press_Pa_imag (the imaginary part of the pressure vs. frequency) are provided, in addition to Press_Pa, Press_dB and bepres.
IUNITS	Flag for unit changes EQ.0: do not apply unit change EQ.1: MKS units are used, no change needed; EQ.2: units ($lbf \times s^2/in$, inch, s, lbf, psi, etc.) are used, changed to MKS in BEM Acoustic computation; EQ.3: units (kg, mm, ms, kN, GPa, etc.) are used, changed to MKS in BEM acoustic computation;
METHOD	Method used in acoustic analysis EQ.0: Rayleigh method (very fast) EQ.1: Kirchhoff method coupled to FEM for acoustics (*MAT_ ACOUSTIC) (see Remark 4) EQ.2: variational Indirect BEM EQ.3: collocation BEM EQ.4: collocation BEM with Burton-Miller formulation for exterior problems (no irregular frequency phenomenon)
MAXIT	Maximum number of iterations for iterative solver (default =100) if METHOD ≥ 2.
TOLITR	Tolerance for the iterative solver (default=1.E-4)

VARIABLE	DESCRIPTION
NDD	Number of domain decomposition, used for memory saving. For large problems, the boundary mesh is decomposed into NDD domains for less memory allocation. This option is only used if METHOD≥2.
TOLLR	Tolerance for low rank approximation of dense matrix (default=1.E-6).
TOLFCT	Tolerance in factorization of the low rank matrix (default=1.E-6).
IBDIM	Inner iteration limit in GMRES iterative solver (default=1000).
NPG	Number of Gauss integration points (default=2).
SSID	Part, part set ID, or segment set ID of boundary elements.
SSTYPE	Boundary element type: EQ.0: part Set ID EQ.1: part ID EQ.2: segment set ID.
NORM	NORM should be set such that the normal vectors face toward the fluid. EQ.0: normal vectors are not inverted (default). EQ.1: normal vectors are inverted.
BEMTYP	Type of input boundary values in BEM analysis. EQ.0: boundary velocity will be processed in BEM analysis EQ.1: boundary acceleration will be processed in BEM analysis EQ.-n: velocity is given in frequency domain, through load curve n. An amplitude versus. frequency load curve (with curve ID n) needs to be defined.
RESTRT	This flag is used to save an LS-DYNA analysis if the binary output file in the (bem=filename) option has not been changed (default = 0). EQ.0: LS-DYNA analysis is processed and generates a new binary file. EQ.1: LS-DYNA analysis is not processed. The binary file from previous run is used.
IEDGE	Free edge and multi-connection constraints option (default = 0). EQ.0: Free edge and multi-connection constraints not considered. EQ.1: Free edge and multi-connection constraints considered. EQ.2: Only free edge constraints are considered. EQ.3: Only multi-connection constraints are considered.

Remarks:

1. TSTART indicates the time at which velocity or acceleration and pressure are stored in the binary file.
2. This reference pressure is required for the computation of the pressure in dB. Usually, in International Unit System the reference pressure is 20 micro_Pascal.
3. Velocity or acceleration (pressure) is provided by LS-DYNA analysis. They are written in a binary file (bem= filename). The boundary element method is processed after the LS-DYNA analysis. An FFT algorithm is used to transform time domain data into frequency domain in order to use the boundary element method for acoustics. In order to overcome the FFT leakage problem due to the truncation of the temporal response, several windows are proposed. Windowing is used to have a periodic velocity, acceleration and pressure in order to use the FFT.
4. If time domain results are requested, FMIN is changed to 0 in the code.
5. We change units used in BEM computation to international units (kg, m, s, N, Pa) so that the reference pressure will not be too small. For example, it can be 20.E-15 GPa if one uses the units (kg, mm, ms, kN, GPa) and this may pose potential truncation error in the computation, especially in single precision version.
6. Rayleigh method represents an approximation for the BEM. It is very fast since there is no linear system to solve. It can only be used for external radiation problem. Kirchhoff method is a coupling between BEM and FEM for acoustics (*MAT_ACOUSTIC) with Non Reflecting Boundary condition, see *BOUNDARY_NON_REFLECTING. In this case, at least one fluid layer with non reflecting boundary condition is to be merged to the vibrating structure. This additional fluid is given in *MAT_ACOUSTIC by the same density and sound speed as used in this keyword. Both methods constitute a good approximation of BEM for external problems.
7. BEM formulation for large and medium size problems (more than 2000 boundary elements) is memory and time consuming. In this case, user may run LS-DYNA using the memory option. In order to save memory, domain decomposition can be used.
8. The binary file generated by a previous run can be used for the next run by using the restart option. The restart option allows the user to use the binary file generated from a previous calculation in order to run BEM. In this case, the frequency range can be changed. User should keep the same time parameters as in previous calculation.
9. IEDGE option only applies to METHOD=2, the Variational Indirect BEM.

BOUNDARY_ELEMENT_METHOD_OPTION*

Available options include:

CONTROL

FLOW

NEIGHBOR

SYMMETRY

WAKE

Purpose: Define input parameters for boundary element method analysis of incompressible fluid dynamics or fluid-structure interaction problems.

The boundary element method (BEM) can be used to compute the steady state or transient fluid flow about a rigid or deformable body. The theory which underlies the method (see the LS-DYNA Theory Manual) is restricted to inviscid, incompressible, attached fluid flow. The method should not be used to analyze flows where shocks or cavitation are present.

In practice the method can be successfully applied to a wider class of fluid flow problems than the assumption of inviscid, incompressible, attached flow would imply. Many flows of practical engineering significance have large Reynolds numbers (above 1 million). For these flows the effects of fluid viscosity are small if the flow remains attached, and the assumption of zero viscosity may not be a significant limitation. Flow separation does not necessarily invalidate the analysis. If well-defined separation lines exist on the body, then wakes can be attached to these separation lines and reasonable results can be obtained. The Prandtl-Glauert rule can be used to correct for non-zero Mach numbers in a gas, so the effects of aerodynamic compressibility can be correctly modeled (as long as no shocks are present).

The BOUNDARY_ELEMENT_METHOD_FLOW card turns on the analysis, and is mandatory.

*BOUNDARY_ELEMENT_METHOD_CONTROL

Purpose: Control the execution time of the boundary element method calculation. The **CONTROL** option is used to control the execution time of the boundary element method calculation, and the use of this option is strongly recommended. The BEM calculations can easily dominate the total execution time of a LS-DYNA run unless the parameters on this card (especially DTBEM and/or IUPBEM) are used appropriately.

DTBEM is used to increase the time increment between calls to the BEM routines. This can usually be done with little loss in accuracy since the characteristic times of the structural dynamics and the fluid flow can differ by several orders of magnitude. The characteristic time of the structural dynamics in LS-DYNA is given by the size of the smallest structural element divided by the speed of sound of its material. For a typical problem this characteristic time might be equal to 1 microsecond. Since the fluid in the boundary element method is assumed to be incompressible (infinite speed of sound), the characteristic time of the fluid flow is given by the streamwise length of the smallest surface in the flow divided by the fluid velocity. For a typical problem this characteristic time might be equal to 10 milliseconds. For this example DTBEM might be set to 1 millisecond with little loss of accuracy. Thus, for this example, the boundary element method would be called only once for every 1000 LS-DYNA iterations, saving an enormous amount of computer time.

IUPBEM is used to increase the number of times the BEM routines are called before the matrix of influence coefficients is recomputed and factored (these are time-consuming procedures). If the motion of the body is entirely rigid body motion there is no need to ever recompute and factor the matrix of influence coefficients after initialization, and the execution time of the BEM can be significantly reduced by setting IUPBEM to a very large number. For situations where the structural deformations are modest an intermediate value (e.g., 10) for IUPBEM can be used.

Define one card.

Card	1	2	3	4	5	6	7	8
Variable	LWAKE	DTBEM	IUPBEM	FARBEM				
Type	I	F	I	F				
Default	50	0.	100	2.0				
Remark	1			2				

VARIABLE	DESCRIPTION
LWAKE	Number of elements in the wake of lifting surfaces. Wakes must be defined for all lifting surfaces.
DTBEM	Time increment between calls to the boundary element method. The fluid pressures computed during the previous call to the BEM will continue to be used for subsequent LS-DYNA iterations until a time increment of DTBEM has elapsed.
IUPBEM	The number of times the BEM routines are called before the matrix of influence coefficients is recomputed and refactored.
FARBEM	Nondimensional boundary between near-field and far-field calculation of influence coefficients.

Remarks:

1. Wakes convect with the free-stream velocity. The number of elements in the wake should be set to provide a total wake length equal to 5-10 times the characteristic streamwise length of the lifting surface to which the wake is attached. Note that each wake element has a streamwise length equal to the magnitude of the free stream velocity multiplied by the time increment between calls to the boundary element method routines. This time increment is controlled by DTBEM.

2. The most accurate results will be obtained with FARBEM set to 5 or more, while values as low as 2 will provide slightly reduced accuracy with a 50% reduction in the time required to compute the matrix of influence coefficients.

*BOUNDARY_ELEMENT_METHOD_FLOW

Purpose: Turn on the boundary element method calculation, specify the set of shells which define the surface of the bodies of interest, and specify the onset flow.

The *BOUNDARY_ELEMENT_METHOD_FLOW card turns on the BEM calculation. This card also identifies the shell elements which define the surfaces of the bodies of interest, and the properties of the onset fluid flow. The onset flow can be zero for bodies which move through a fluid which is initially at rest.

Define one card.

Card	1	2	3	4	5	6	7	8
Variable	SSID	VX	VY	VZ	RO	PSTATIC	MACH	
Type	I	F	F	F	F	F	F	
Default	none	none	none	none	none	0.	0.	
Remark	1					2	3	

VARIABLE	DESCRIPTION
SSID	Shell set ID for the set of shell elements which define the surface of the bodies of interest (see *SET_SHELL). The nodes of these shells should be ordered so that the shell normals point into the fluid.
VX, VY, VZ	x, y, and z components of the free-stream fluid velocity.
RO	Fluid density.
PSTATIC	Fluid static pressure.
MACH	Free-stream Mach number.

Remarks:

1. It is recommended that the shell segments in the SSID set use the NULL material (see *MAT_NULL). This will provide for the display of fluid pressures in the post-processor. For triangular shells the 4th node number should be the same as the 3rd node number. For fluid-structure interaction problems it is recommended that the boundary element shells use the same nodes and be coincident with the structural shell elements (or the outer face of solid elements) which define the surface of the body. This approach guarantees that the boundary element segments will move with the surface of the body as it deforms.

2. A pressure of PSTATIC is applied uniformly to all segments in the segment set. If the body of interest is hollow, then PSTATIC should be set to the free-stream static pressure minus the pressure on the inside of the body.

3. The effects of subsonic compressibility on gas flows can be included using a non-zero value for MACH. The pressures which arise from the fluid flow are increased using the Prandtl-Glauert compressibility correction. MACH should be set to zero for water or other liquid flows.

*BOUNDARY_ELEMENT_METHOD_NEIGHBOR

Purpose: Define the neighboring elements for a given boundary element segment.

The pressure at the surface of a body is determined by the gradient of the doublet distribution on the surface (see the LS-DYNA Theory Manual). The "Neighbor Array" is used to specify how the gradient is computed for each boundary element segment. Ordinarily, the Neighbor Array is set up automatically by LS-DYNA, and no user input is required. The NEIGHBOR option is provided for those circumstances when the user desires to define this array manually.

For the **NEIGHBOR** option define the following cards:

Cards 1, 2, 3, ... (The next "*" card terminates the input.)

Card	1	2	3	4	5	6	7	8
Variable	NELEM	NABOR1	NABOR2	NABOR3	NABOR4			
Type	I	I	I	I	I			
Default	none	none	none	none	none			

VARIABLE	DESCRIPTION
NELEM	Element number.
NABOR1	Neighbor for side 1 of NELEM.
NABOR2	Neighbor for side 2 of NELEM.
NABOR3	Neighbor for side 3 of NELEM.
NABOR4	Neighbor for side 4 of NELEM.

Remarks:

Each boundary element has 4 sides (Figure 3.2). Side 1 connects the 1st and 2nd nodes, side 2 connects the 2nd and 3rd nodes, etc. The 4th side is null for triangular elements.

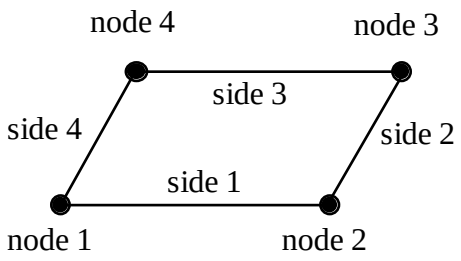


Figure 3.2 Each segment has 4 sides.

For most elements the specification of neighbors is straightforward. For the typical case a quadrilateral element is surrounded by 4 other elements, and the neighbor array is as shown in Figure 3.3.

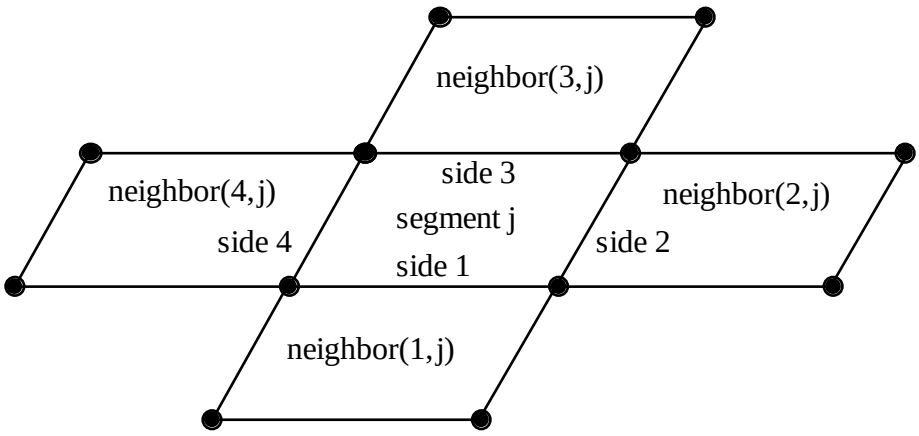


Figure 3.3 Typical neighbor specification.

There are several situations for which the user may desire to directly specify the neighbor array for certain elements. For example, boundary element wakes result in discontinuous doublet distributions, and neighbors which cross a wake should not be used. Figure 3.4 illustrates a situation where a wake is attached to side 2 of segment j. For this situation two options exist. If neighbor(2,j) is set to zero, then a linear computation of the gradient in the side 2 to side 4 direction will be made using the difference between the doublet strengths on segment j and segment neighbor(4,j). This is the default setup used by LS-DYNA when no user input is provided. By specifying neighbor(2,j) as a negative number a more accurate quadratic curve fit will be used to compute the gradient. The curve fit will use segment j, segment neighbor(4,j), and segment -neighbor(2,j); which is located on the opposite side of segment neighbor(4,j) as segment j.

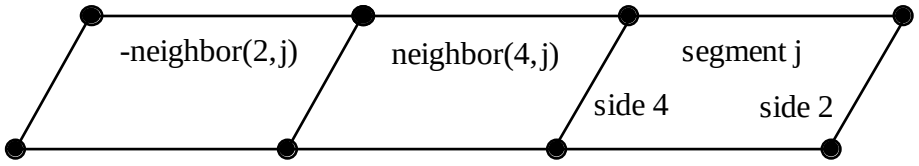


Figure 3.4 If neighbor(2,j) is a negative number it is assumed to lie on the opposite side of neighbor(4,j) as segment j.

Another possibility is that no neighbors at all are available in the side 2 to side 4 direction. In this case both neighbor(2,j) and neighbor(4,j) can be set to zero, and the gradient in that direction will be assumed to be zero. This option should be used with caution, as the resulting fluid pressures will not be accurate for three-dimensional flows. However, this option is occasionally useful where quasi-two dimensional results are desired. All of the above options apply to the side 1 to side 3 direction in the obvious ways.

For triangular boundary elements side 4 is null. Gradients in the side 2 to side 4 direction can be computed as described above by setting neighbor(4,j) to zero for a linear derivative computation (this is the default setup used by LS-DYNA when no user input is provided) or to a negative number to use the segment on the other side of neighbor(2,j) and a quadratic curve fit. There may also be another triangular segment which can be used as neighbor(4,j) (see Figure 3.5).

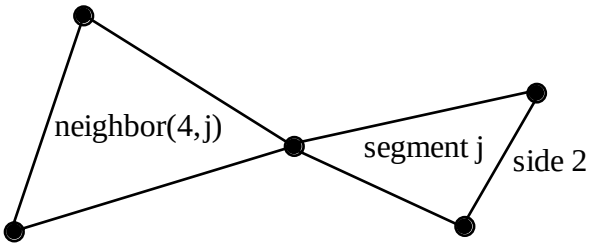


Figure 3.5 Sometimes another triangular boundary element segment can be used as neighbor (4,j).

The rules for computing the doublet gradient in the side 2 to side 4 direction can be summarized as follows (the side 1 to side 3 case is similar):

Table 3.1 Surface pressure computation for element j.

NABOR2	NABOR4	Doublet Gradient Computation
GT.0	GT.0	quadratic fit using elements j, NABOR2, and NABOR4
LT.0	GT.0	quadratic fit using elements j, -NABOR2, and NABOR4. -NABOR2 is assumed to lie on the opposite side of NABOR4 as segment j (see Fig. 3.4)
GT.0	LT.0	quadratic fit using elements j, NABOR2, and -NABOR4. -NABOR4 is assumed to lie on the opposite side of NABOR2 as segment j
EQ.0	GT.0	linear fit using elements j and NABOR4
GT.0	EQ.0	linear fit using elements j and NABOR2
EQ.0	EQ.0	zero gradient

*BOUNDARY_ELEMENT_METHOD_SYMMETRY

Purpose: To define a plane of symmetry for the boundary element method. The SYMMETRY option can be used to reduce the time and memory required for symmetric configurations. For these configurations the reduction in the number of boundary elements by a factor of 2 will reduce the memory used by the boundary element method by a factor of 4, and will reduce the computer time required to factor the matrix of influence coefficients by a factor of 8. Only 1 plane of symmetry can be defined.

For the **SYMMETRY** option define the following card:

Define one card.

Card	1	2	3	4	5	6	7	8
Variable	BEMSYM							
Type	I							
Default	0							
Remark								

VARIABLE	DESCRIPTION
BEMSYM	Defines symmetry plane for boundary element method. EQ.0: no symmetry plane is defined EQ.1: x = 0 is a symmetry plane EQ.2: y = 0 is a symmetry plane EQ.3: z = 0 is a symmetry plane

***BOUNDARY_ELEMENT_METHOD_WAKE**

Purpose: To attach wakes to the trailing edges of lifting surfaces. Wakes should be attached to boundary elements at the trailing edge of a lifting surface (such as a wing, propeller blade, rudder, or diving plane). Wakes should also be attached to known separation lines when detached flow is known to exist (such as the sharp leading edge of a delta wing at high angles of attack). Wakes are required for the correct computation of surface pressures for these situations. As described above, two segments on opposite sides of a wake should never be used as neighbors.

For the **WAKE** option define the following cards:

Cards 1, 2, 3, ... (The next "*" card terminates the input.)

Card	1	2	3	4	5	6	7	8
Variable	NELEM	NSIDE						
Type	I	I						
Default	none	none						
Remark	1							

VARIABLE	DESCRIPTION
NELEM	Element number to which a wake is attached.
NSIDE	The side of NELEM to which the wake is attached (see Fig. 3.2). This should be the "downstream" side of NELEM.

Remarks:

1. Normally two elements meet at a trailing edge (one on the "upper" surface and one on the "lower" surface). The wake can be attached to either element, but not to both.

BOUNDARY_FLUX_OPTION*

Available options include:

SEGMENT

SET

Purpose: Define flux boundary conditions for a thermal or coupled thermal/structural analysis. Two or more cards are defined for each option. History variables can be associated with the boundary condition which will invoke a call to a user defined boundary flux subroutine for computing the flux.

For the **SET** option define the following card:

(Card 1 of 2)

Card 1	1	2	3	4	5	6	7	8
Variable	SSID							
Type	I							
Default	none							

For the **SEGMENT** option define the following card:

(Card 1 of 2)

Card 1	1	2	3	4	5	6	7	8
Variable	N1	N2	N3	N4				
Type	I	I	I	I				
Default	none	none	none	none				

Define the following card for both options:

(Card 2)

Card 2	1	2	3	4	5	6	7	8
Variable	LCID	MLC1	MLC2	MLC3	MLC4	LOC	NHISV	
Type	I	F	F	F	F	I	I	
Default	none	0.	0.	0.	0.	0	0	

Define as many cards as necessary to initialize NHISV history variables.
(Card 3 …)

Card 3	1	2	3	4	5	6	7	8
Variable	HISV1	HISV2	HISV3	HISV4	HISV5	HISV6	HISV7	HISV8
Type	F	F	F	F	F	F	F	F
Default	0.	0.	0.	0.	0.	0.	0.	0.

VARIABLE	DESCRIPTION
SSID	Segment set ID, see *SET_SEGMENT
N1,N2...	Node ID's defining segment
LCID	Load curve ID for heat flux, see *DEFINE_CURVE: GT.0: function versus time, EQ.0: use constant multiplier values at nodes, LT.0: function versus temperature.
MLC1	Curve multiplier at node N_1, see Figure 3.2.
MLC2	Curve multiplier at node N_2, see Figure 3.2.
MLC3	Curve multiplier at node N_3, see Figure 3.2.
MLC4	Curve multiplier at node N_4, see Figure 3.2.

VARIABLE	DESCRIPTION
LOC	Application of surface for thermal shell elements, see parameter, TSHELL, in the *CONTROL_SHELL input: EQ.-1: lower surface of thermal shell element EQ.1: upper surface of thermal shell element
NHISV	Number of history variables associated with the flux definition: GT.0: A user defined subroutine will be called to compute the flux. See Remark 1.
HISV1	Initial value of history variable 1
HISV2 . .	Initial value of history variable 2
HISVn	Initial value of history variable n, where n = NHISV

Remarks:

1. Flux can be defined by:

   a) a constant baseline value defined by MLC1, MLC2, MLC3, MLC4

   b) LCID – load curve id

   c) FID – function id

   d) user subroutine

2. If NHISV is a number greater than 0, the user subroutine

   ```
   subroutine usrflux(fl,flp,…)
   ```

   will be called to compute the flux (fl) defined as heat (energy) per time per surface area. For more details see Appendix S.

3. Three definitions for heat flux are possible. Heat flux can be a function of time, a function of temperature, or constant values that are maintained throughout the calculation. With the definition of multipliers at each node of the segment, a bilinear spatial variation can be assumed.

   By convention, heat flow is negative in the direction of the surface outward normal vector. Surface definition is in accordance with the left hand rule. The outward normal vector points to the left as one progresses from node N_1-N_2-N_3-N_4. See Figure 3.6.

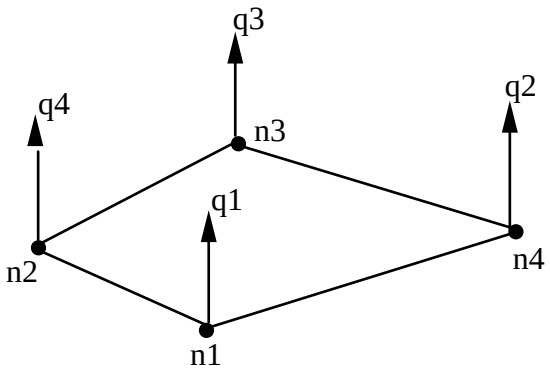


Figure 3.6. Nodal number determines outward normal.

*BOUNDARY_MCOL

Purpose: Define parameters for MCOL coupling. The MCOL Program is a rigid body mechanics program for modeling the dynamics of ships. See Remark 1 for more information.

Card	1	2	3	4	5	6	7	8
Variable	NMCOL	MXSTEP	ENDTMCOL	TSUBC	PRTMCOL			
Type	I	I	F	F	F			
Default	2	none	0.0	0.0	none			
Remarks			2					

Card 2 must be defined for each ship

Card 2	1	2	3	4	5
Variable	RBMCOL	MCOLFILE			
Type	I	A60			
Default		None			
Remarks					

VARIABLE	DESCRIPTION
NMCOL	Number of ships in MCOL coupling.
MXSTEP	Maximum of time step in MCOL calculation. If the number of MCOL time steps exceeds MXSTEP, then LS-DYNA will terminate.
ENDTMCOL	Uncoupling termination time, see Remark 2 below. EQ.0.0: set to LS-DYNA termination time
TSUBC	Time interval for MCOL subcycling. EQ.0.0: no subcycling

VARIABLE	DESCRIPTION
PRTMCOL	Time interval for output of MCOL rigid body data.
RBMCOL	LS-DYNA rigid body material assignment for the ship.
MCOLFILE	Filename containing MCOL input parameters for the ship.

Remarks:

1. The basis for MCOL is a convolution integral approach for simulating the equations of motion. A mass and inertia tensor are required as input for each ship. The masses are then augmented to include the effects of the mass of the surrounding water. A separate program determines the various terms of the damping/buoyancy force formulas which are also input to MCOL. The coupling is accomplished in a simple manner: at each time step LS-DYNA computes the resultant forces and moments on the MCOL rigid bodies and passes them to MCOL. MCOL then updates the positions of the ships and returns the new rigid body locations to LS-DYNA. A more detailed theoretical and practical description of MCOL can be found in a separate report (to appear).

2. After the end of the LS-DYNA / MCOL calculation, the analysis can be pursued using MCOL alone. ENDTMCOL is the termination time for this analysis. If ENDTMCOL is lower than the LS-DYNA termination time, the uncoupled analysis will not be activated.

3. The MCOL output is set to the files MCOLOUT (ship position) and MCOLENERGY (energy breakdown). In LS-PREPOST, MCOLOUT can be plotted through the rigid body time history option and MCOLENERGY.

*BOUNDARY_NON_REFLECTING

Purpose: Define a non-reflecting boundary. This option applies to continuum domains modeled with solid elements, as indefinite domains are usually not modeled. For geomechanical problems this option is important for limiting the size of the models.

Card 1	1	2	3	4	5	6	7	8
Variable	SSID	AD	AS					
Type	I	F	F					
Default	none	0.0	0.0					
Remarks	1, 2	3	3					

VARIABLE	DESCRIPTION
SSID	Segment set ID, see *SET_SEGMENT.
AD	Default activation flag for dilatational waves. (on.EQ.0.0, off.NE.0.0)
AS	Default activation flag for shear waves. (on.EQ.0.0, off.NE.0.0)

Remarks:

1. Non-reflecting boundaries defined with this keyword are only used with three-dimensional solid elements. Boundaries are defined as a collection of segments, and segments are equivalent to element faces on the boundary. Segments are defined by listing the corner nodes in either a clockwise or counterclockwise order.

2. Non-reflecting boundaries are used on the exterior boundaries of an analysis model of an infinite domain, such as a half-space to prevent artificial stress wave reflections generated at the model boundaries form reentering the model and contaminating the results. Internally, LS-DYNA computes an impedance matching function for all non-reflecting boundary segments based on an assumption of linear material behavior. Thus, the finite element mesh should be constructed so that all significant nonlinear behavior is contained within the discrete analysis model.

3. With the two optional switches, the influence of reflecting waves can be studied.

4. During the dynamic relaxation phase (optional), nodes on non-reflecting segments are constrained in the normal direction. Nodal forces associated with these constraints are then applied as external loads and held constraint in the transient phase while the constraints are replaced with the impedance matching functions. In this manner, soil can be quasi-statically prestressed during the dynamic relaxation phase and dynamic loads (with non-reflecting boundaries) subsequently applied in the transient phase.

*BOUNDARY_NON_REFLECTING_2D

Purpose: Define a non-reflecting boundary. This option applies to continuum domains modeled with two-dimensional solid elements in the xy plane, as indefinite domains are usually not modeled. For geomechanical problems, this option is important for limiting the size of the models.

Card	1	2	3	4	5	6	7	8
Variable	NSID							
Type	I							
Default	none							
Remarks	1, 2							

VARIABLE	DESCRIPTION
NSID	Node set ID, see *SET_NODE. See Figure 3.7.

Remarks:

1. Non-reflecting boundaries defined with this keyword are only used with two-dimensional solid elements in either plane strain or axisymmetric geometries. Boundaries are defined as a sequential string of nodes moving counterclockwise around the boundary.

2. Non-reflecting boundaries are used on the exterior boundaries of an analysis model of an infinite domain, such as a half-space to prevent artificial stress wave reflections generated at the model boundaries form reentering the model and contaminating the results. Internally, LS-DYNA computes an impedance matching function for all non-reflecting boundary segments based on an assumption of linear material behavior. Thus, the finite element mesh should be constructed so that all significant nonlinear behavior in contained within the discrete analysis model.

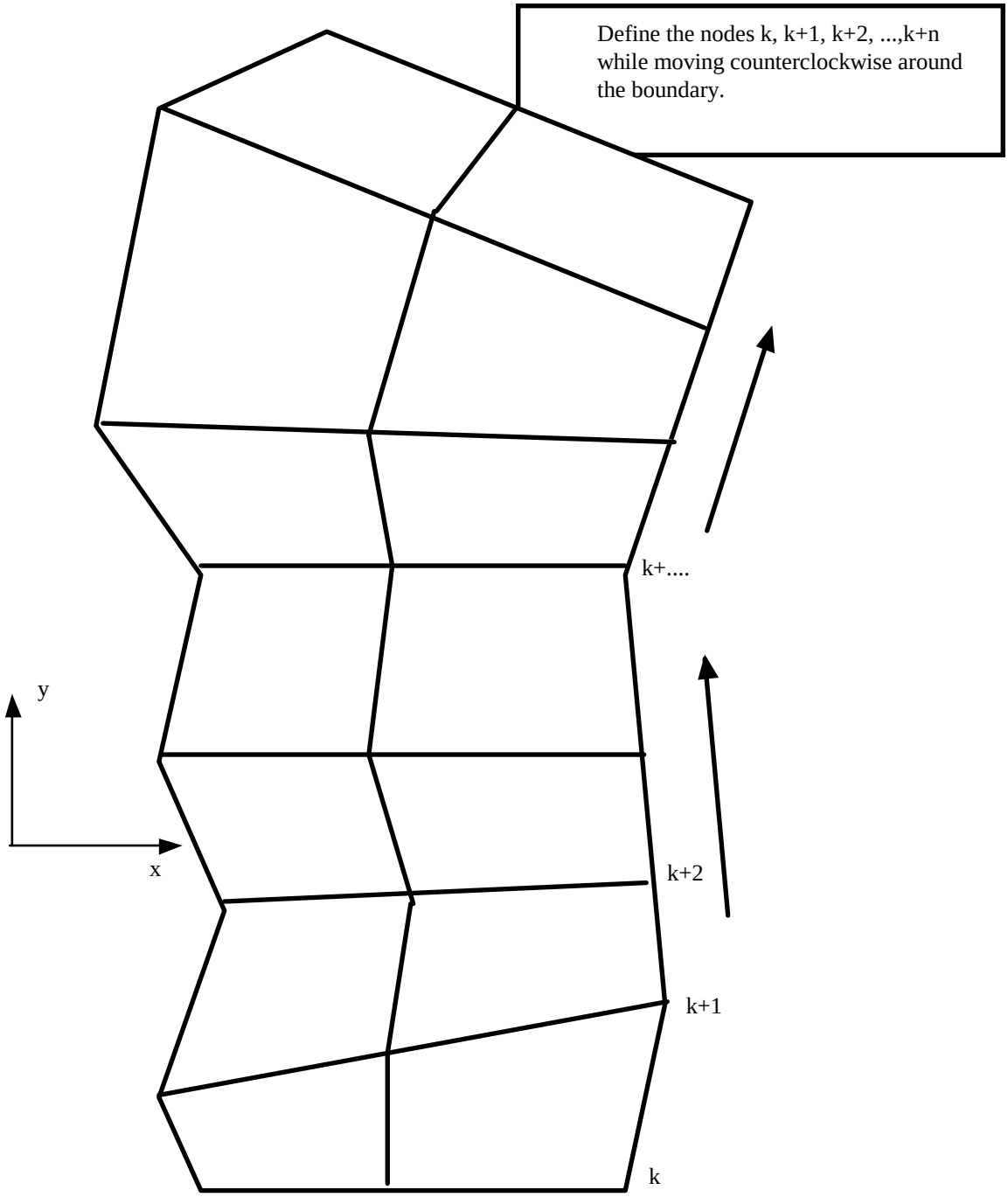


Figure 3.7. When defining a transmitting boundary in 2D define the node numbers in the node set in consecutive order while moving counterclockwise around the boundary.

BOUNDARY_PORE_FLUID_OPTION*

Available options include:

PART
SET

Purpose: Define parts that contain pore fluid. Defaults are given on *CONTROL_PORE_FLUID.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	*type*ID	WTABLE	PF_RHO	ATYPE	PF_BULK	ACURVE	WTCUR	SUCLIM
Type	I	F	F	I	F	I	I	F
Default	none	*	*	*	*	0	0	0.

* Defaults are taken from *CONTROL_PORE_FLUID

VARIABLE	DESCRIPTION
PID, PSID	Part ID (PID) or Part set ID, see *PART and *SET_PART. All elements within the part must lie below the water table.
WTABLE	Z-coordinate at which pore pressure = 0 (water table)
PF_RHO	Density of pore water in soil skeleton: EQ.0: Default density specified on *CONTROL_PORE_FLUID card is used.
ATYPE	Analysis type for Parts: EQ.0: Default to value specified on *CONTROL_PORE_FLUID EQ 1: Undrained analysis EQ 2: Drained analysis EQ 3:Time dependent consolidation (coupled) EQ 4:Consolidate to steady state (uncoupled) EQ 5:Drained in dynamic relaxation, undrained in transient

VARIABLE	DESCRIPTION
PF_BULK	Bulk modulus of pore fluid: EQ.0: Default to value specified on *CONTROL_PORE_FLUID
ACURVE	Curve of analysis type vs time (see notes below)
WTCUR	Curve of water table (z-coordinate) vs time
SUCLIM	Suction limit (defined in head, i.e. length units). Must not be negative. See notes.

This card must be present for all parts having pore water.

The density on this card is used only to calculate pressure head. To ensure the correct gravity loading, the density of the soil material should be increased to include the mass associated with the pore water.

The y-axis values of the curve of analysis type vs time can only be 1 or 2 (undrained or drained) at present. In future it will be extended to include time-dependent. During dynamic relaxation, the analysis type will be taken from the first value on the curve

The default for SUCLIM is zero, meaning that the pore fluid cannot generate suction. To allow unlimited suction, set this parameter to a large positive number.

*BOUNDARY_PRESCRIBED_ACCELEROMETER_RIGID

Purpose: Prescribe the motion of a rigid body based on experimental data obtained from accelerometers affixed to the rigid body.

Note: This feature is available only in release R3 and higher of Version 971.

Card 1	1	2	3	4	5	6	7	8
Variable	PID	SOLV						
Type	I	I						
Default	none	1						

Card 2, 3, 4, etc. Define one card for each accelerometer affixed to the rigid body. Input is terminated when a "*" card is found. A minimum of three accelerometers are required (see Remarks below).

Card	1	2	3	4	5	6	7	8
Variable	NID	CID	LCIDX	LCIDY	LCIDZ			
Type	I	I	I	I	I			
Default	none	none	none	none	none			

VARIABLE	DESCRIPTION
PID	Part ID for rigid body whose motion is prescribed.
SOLV	Solver type: EQ.1: Gaussian elimination (default), EQ.2: linear regression
NID	Node ID corresponding to the location of the accelerometer.
CID	Coordinate system ID describing the orientation of the accelerometer's local axes (see *DEFINE_COORDINATE).

VARIABLE	DESCRIPTION
LCIDX	Load curve ID containing the local x-acceleration time history from the accelerometer.
LCIDY	Load curve ID containing the local y-acceleration time history from the accelerometer.
LCIDZ	Load curve ID containing the local z-acceleration time history from the accelerometer.

Remarks:

1. Acceleration time histories from a minimum of three accelerometers each providing output from three channels are required.

2. Local axes of the accelerometers must be orthogonal.

*BOUNDARY_PRESCRIBED_FINAL_GEOMETRY

The final displaced geometry for a subset of nodal points is defined. The nodes of this subset are displaced from their initial positions specified in the *NODE input to the final geometry along a straight line trajectory. A load curve defines a scale factor as a function of time that is bounded between zero and unity corresponding to the initial and final geometry, respectively. A unique load curve can be specified for each node, or a default load curve can apply to all nodes. The external work generated by the displacement field is included in the energy ratio calculation for the GLSTAT file.

Card 1	1	2	3	4	5	6	7	8
Variable	BPFGID	LCIDF	DEATHD					
Type	I	I	F					
Default	0	0	infinity					

Card Format (I8,3E16.0,I8,E16.0) The next "*" keyword card terminates this input.

Card 2,...	1	2	3	4	5	6	7	8	9	10
Variable	NID	X		Y		Z		LCID	DEATH	
Type	I	F		F		F		I	F	
Default	none	0.		0.		0.		LCIDF	infinity	

VARIABLE	DESCRIPTION
BPFGID	ID for this set of imposed boundary conditions
LCIDF	Default load curve ID. This curve varies between zero and unity.
DEATHD	Default death time. At this time the prescribed motion is inactive and the nodal point is allowed to move freely.
NID	Node ID for which the final position is defined. Nodes defined in this section must also appear under the *NODE input. .
X	x-coordinate of final geometry

VARIABLE	DESCRIPTION
Y	y-coordinate of final geometry
Z	z-coordinate of final geometry
LCID	Load curve ID. If zero the default curve ID, LCIDF, is used.
DEATH	Death time. If zero the default value, DEATHD, is used..

BOUNDARY_PRESCRIBED_MOTION_OPTION1*_{*OPTION2*}

Available options for *OPTION1* include:

NODE

SET

RIGID

RIGID_LOCAL

OPTION2 allows an optional ID to be given that applies either to the single node, node set or a rigid body:

ID

If a heading is defined with the ID, then the ID with the heading will be written at the beginning of the ASCII file, BNDOUT.

Purpose: Define an imposed nodal motion (velocity, acceleration, or displacement) on a node or a set of nodes. Also velocities and displacements can be imposed on rigid bodies. If the local option is active the motion is prescribed with respect to the local coordinate system for the rigid body (See variable LCO for keyword *MAT_RIGID). Translational nodal velocity and acceleration specifications for rigid body nodes are allowed and are applied as described at the end of this section. For nodes on rigid bodies use the NODE option. Do not use the NODE option in r-adaptive problems since the node ID's may change during the adaptive step.

The following card is read if and only if the ID option is specified. The second card is required.

Optional 1 2-8

Variable	ID	HEADING
Type	I	A70

Card 1 1 2 3 4 5 6 7 8

Variable	*type*ID	DOF	VAD	LCID	SF	VID	DEATH	BIRTH
Type	I	I	I	I	F	I	F	F
Default	none	None	0	none	1.	0	1.E+28	0.0

Card is required if DOF=9,10,11 on the first card or VAD=4. If DOF<9 and VAD<4, skip this card.

Card 2 1 2 3 4 5 6 7 8

Variable	OFFSET1	OFFSET2	MRB	NODE1	NODE2			
Type	F	F	I	I	I			
Default	0.	0.	0	0	0			

VARIABLE **DESCRIPTION**

ID — Optional PRESCRIBED MOTION set ID to which this node, node set, or rigid body belongs. This ID does not need to be unique.

HEADING — An optional descriptor for the given ID that will be written into the D3HSP file and the BNDOUT file.

*type*ID — Node ID (NID in *NODE), nodal set ID (SID in *SET_NODE) , or part ID (PID in *PART) for a rigid body.

DOF — Applicable degrees-of-freedom:

EQ.1: x-translational degree-of-freedom,
EQ.2: y-translational degree-of-freedom,
EQ.3: z-translational degree-of-freedom,
EQ.4: translational motion in direction given by the VID. Movement on plane normal to the vector is permitted.
EQ.-4: translational motion in direction given by the VID. Movement on plane normal to the vector is not permitted. This option does not apply to rigid bodies.
EQ.5: x-rotational degree-of-freedom,
EQ.6: y-rotational degree-of-freedom,
EQ.7: z-rotational degree-of-freedom,
EQ.8: rotational motion about a vector parallel to vector VID. Rotation about the normal axes is permitted.
EQ.-8: rotational motion about a vector parallel to vector VID. Rotation about the normal axes is not permitted. This option does not apply to rigid bodies.
EQ.9: y/z degrees-of-freedom for node rotating about the x-axis at location (OFFSET1,OFFSET2) in the yz-plane, *point (y,z)*. Radial motion is NOT permitted. Not applicable to rigid bodies.
EQ.-9: y/z degrees-of-freedom for node rotating about the x-axis at location (OFFSET1,OFFSET2) in the yz-plane, *point (y,z)*. Radial motion is permitted. Not applicable to rigid bodies.

VARIABLE	DESCRIPTION
	EQ.10: z/x degrees-of-freedom for node rotating about the y-axis at location (OFFSET1,OFFSET2) in the zx-plane, *point (z,x).* Radial motion is NOT permitted. Not applicable to rigid bodies. EQ.-10: z/x degrees-of-freedom for node rotating about the y-axis at location (OFFSET1,OFFSET2) in the zx-plane, *point (z,x).* Radial motion is permitted. Not applicable to rigid bodies. EQ.11: x/y degrees-of-freedom for node rotating about the z-axis at location (OFFSET1,OFFSET2) in the xy-plane, *point (x,y).* Radial motion is NOT permitted. Not applicable to rigid bodies. EQ.-11: x/y degrees-of-freedom for node rotating about the z-axis at location (OFFSET1,OFFSET2) in the xy-plane, *point (x,y).* Radial motion is permitted. Not applicable to rigid bodies.
VAD	Velocity/Acceleration/Displacement flag: EQ.0: velocity (rigid bodies and nodes), EQ.1: acceleration (rigid bodies and nodes), EQ.2: displacement (rigid bodies and nodes). EQ.3: velocity versus displacement (rigid bodies and nodes) EQ.4: relative displacement (rigid bodies only)
LCID	Load curve ID to describe motion value versus time, see *DEFINE_ CURVE, *DEFINE_CURVE_FUNCTION, or *DEFINE_FUNCTION. See BIRTH below.
SF	Load curve scale factor. (default=1.0)
VID	Vector ID for DOF values of 4 or 8, see *DEFINE_VECTOR.
DEATH	Time imposed motion/constraint is removed: EQ.0.0: default set to 10^{28}
BIRTH	Time imposed motion/constraint is activated starting from the initial abscissa value of the curve or function (*DEFINE_FUNCTION). Warning: BIRTH is ignored if the LCID is defined as a function, i.e., *DEFINE_CURVE_FUNCTION.
OFFSET1	Offset for DOF types 9-11 (y, z, x direction)
OFFSET2	Offset for DOF types 9-11 (z, x, y direction)
MRB	Master rigid body for measuring the relative displacement.
NODE1	Optional orientation node, n1, for relative displacement
NODE2	Optional orientation node, n2, for relative displacement

Remarks:

When DOF=5, 6, 7, or 8, nodal rotational degrees-of-freedom are prescribed in the case of deformable nodes (*OPTION1*=NODE or SET) whereas body rotations are prescribed in the case of a rigid body (*OPTION1*=RIGID). In the case of a rigid body, the axis of prescribed rotation always passes through the body's center of mass. For |DOF|=8, the axis of the prescribed rotation is parallel to vector VID. To prescribe a body rotation of a set of deformable nodes, with the axis of rotation parallel to global axes x, y, or z, use *OPTION1*=SET with |DOF| = 9, 10, or 11, respectively. The load curve scale factor can be used for simple modifications or unit adjustments.

The relative displacement can be measured in either of two ways:

1. Along a straight line between the mass centers of the rigid bodies,

2. Along a vector beginning at node n1 and terminating at node n2.

With option 1, a positive displacement will move the rigid bodies further apart, and, likewise a negative motion will move the rigid bodies closer together. The mass centers of the rigid bodies must not be coincident when this option is used. With option 2 the relative displacement is measured along the vector, and the rigid bodies may be coincident. Note that the motion of the master rigid body is not directly affected by this option, i.e., no forces are generated on the master rigid body.

The activation time, BIRTH, is the time during the solution that the constraint begins to act. Until this time, the prescribed motion card is ignored. The function value of the load curves will be evaluated at the offset time given by the difference of the solution time and BIRTH, i.e., (solution time-BIRTH). Relative displacements that occur prior to reaching BIRTH are ignored. Only relative displacements that occur after BIRTH are prescribed.

When the constrained node is on a rigid body, the translational motion is imposed without altering the angular velocity of the rigid body by calculating the appropriate translational velocity for the center of mass of the rigid body using the equation:

$$v_{cm} = v_{node} - \omega \times (x_{cm} - x_{node})$$

where v_{cm} is the velocity of the center of mass, v_{node} is the specified nodal velocity, ω is the angular velocity of the rigid body, x_{cm} is the current coordinate of the mass center, and x_{node} is the current coordinate of the nodal point. Extreme care must be used when prescribing motion of a rigid body node. Typically, for nodes on a given rigid body, the motion of no more than one node should be prescribed or unexpected results may be obtained.

When the RIGID option is used to prescribe rotation of a rigid body, the axis of rotation will always be shifted such that it passes through the center-of-mass of the rigid body. By using *PART_INERTIA or *CONSTRAINED_NODAL_RIGID_BODY_INERTIA, one can override the internally-calculated location of the center-of-mass.

When the RIGID_LOCAL option is invoked, the orientation of the local coordinate system rotates with time in accordance with rotation of the rigid body.

Angular displacements are applied in an incremental fashion hence it is not possible to correctly prescribe a successive set of rotations about multiple axes. In light of this the command *BOUNDARY_PRESCRIBED_ORIENTATION_RIGID should be used for the purpose of prescribing the general orientation of a rigid body.

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$  *BOUNDARY_PRESCRIBED_MOTION_SET
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  A set of nodes is given a prescribed translational velocity in the
$  x-direction according to a specified vel-time curve (which is scaled).
$
*BOUNDARY_PRESCRIBED_MOTION_SET
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$     nsid       dof       vad      lcid        sf       vid     death
         4         1         0         8       2.0
$
$     nsid = 4    nodal set ID number, requires a *SET_NODE_option
$      dof = 1    motion is in x-translation
$      vad = 0    motion prescribed is velocity
$     lcid = 8    velocity follows load curve 8, requires a *DEFINE_CURVE
$       sf = 2.0  velocity specified by load curve is scaled by 2.0
$      vid        not used in this example
$    death        use default (essentially no death time for the motion)
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$

$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$  *BOUNDARY_PRESCRIBED_MOTION_RIGID
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  A rigid body is given a prescribed rotational displacement about the
$  z-axis according to a specified displacement-time curve.
$
*BOUNDARY_PRESCRIBED_MOTION_RIGID
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$      pid       dof       vad      lcid        sf       vid     death
        84         7         2         9                          14.0
$
$      pid = 84   apply motion to part number 84
$      dof = 7    rotation is prescribed about the z-axis
$      vad = 2    the prescribed motion is displacement (angular)
$     lcid = 9    rotation follows load curve 9, requires a *DEFINE_CURVE
$                     (rotation should be in radians)
$       sf        use default (sf = 1.0)
$      vid        not used in this example
$    death = 14   prescribed motion is removed at 14 ms (assuming time is in ms)
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

BOUNDARY_PRESCRIBED_ORIENTATION_RIGID_*OPTION

Available options include:

DIRCOS

ANGLES

EULERP

VECTOR

Purpose: Prescribe the orientation of rigid body as a function of time.

Note: This feature is available only in release R3 and higher of Version 971.

Card Formats:

Card 1 is common to all orientation methods.
Cards 2 to 3 are unique for each orientation method.

Card 1 - Required for all orientation methods.

Card 1	1	2	3	4	5	6	7	8
Variable	PIDB	PIDA	INTRP					
Type	I	I	I					
Default	none	0	1					

VARIABLE	DESCRIPTION
PIDB	Part ID for rigid body B whose orientation is prescribed. See Remark 1.
PIDA	Part ID for rigid body A. If zero then orientation of PIDB is performed with respect to the global reference frame. See Remark 1.
INTRP	Interpolation method used on time history curves: EQ.1: linear interpolation (default) EQ.2: cubic spline interpolation

Card 2 of 3 - Required for DIRCOS option.

Card 2	1	2	3	4	5	6	7	8
Variable	LCIDC11	LCIDC12	LCIDC13	LCIDC21	LCIDC22	LCIDC23	LCIDC31	LCIDC32
Type	I	I	I	I	I	I	I	I
Default	none	none	None	none	none	none	none	none

Card 3	1	2	3	4	5	6	7	8
Variable	LCIDC33							
Type	I							
Default	none							

VARIABLE	DESCRIPTION
LCIDCij	Load curve ID specifying direction cosine C_{ij} as a function of time. C_{ij} is defined as: $$C_{ij} \triangleq \mathbf{a}_i \bullet \mathbf{b}_j$$ where $\mathbf{a}_i$ (i=1,2,3) are mutually perpendicular unit vectors fixed in PIDA and $\mathbf{b}_j$ (j=1,2,3) are mutually perpendicular unit vectors fixed in PIDB. If PIDA=0 then $\mathbf{a}_j$ (j=1,2,3) are unit vectors aligned, respectively, with the global axes X, Y, and Z. See Remark 2.

Card 2 - Required for ANGLES option.

Card 2	1	2	3	4	5	6	7	8
Variable	LCIDQ1	LCIDQ2	LCIDQ3	ISEQ	ISHFT			
Type	I	I	I	I	I			
Default	none	none	none	none	1			

VARIABLE	DESCRIPTION
LCIDQi	Load curve ID specifying the orientation angle q_i as a function of time.
ISEQ	Specifies the sequence in which the rotations are effected. In this first set of sequences three unique axes are involved. EQ.123: the first rotation is performed about the x axis an amount q_1, the second about the y axis an amount q_2 and the third about the z axis an amount q_3. EQ.231: the first rotation is performed about the y axis an amount q_1, the second about the z axis an amount q_2 and the third about the x axis an amount q_3. EQ.312: the first rotation is performed about the z axis an amount q_1, the second about the x axis an amount q_2 and the third about the y axis an amount q_3. EQ.132: the first rotation is performed about the x axis an amount q_1, the second about the z axis an amount q_2 and the third about the y axis an amount q_3. EQ.213: the first rotation is performed about the y axis an amount q_1, the second about the x axis an amount q_2 and the third about the z axis an amount q_3. EQ.321: the first rotation is performed about the z axis an amount q_1, the second about the y axis an amount q_2 and the third about the x axis an amount q_3. The second set of sequences involve only two unique axes where the first and third are repeated. EQ.121: the first rotation is performed about the x axis an amount q_1, the second about the y axis an amount q_2 and the third about the x axis an amount q_3. EQ.131: the first rotation is performed about the x axis an amount q_1, the second about the z axis an amount q_2 and the third about the x axis an amount q_3.

VARIABLE	DESCRIPTION
	EQ.212: the first rotation is performed about the y axis an amount q_1, the second about the x axis an amount q_2 and the third about the y axis an amount q_3. EQ.232: the first rotation is performed about the y axis an amount q_1, the second about the z axis an amount q_2 and the third about the y axis an amount q_3. EQ.313: the first rotation is performed about the z axis an amount q_1, the second about the x axis an amount q_2 and the third about the z axis an amount q_3. EQ.323: the first rotation is performed about the z axis an amount q_1, the second about the x axis an amount q_2 and the third about the z axis an amount q_3.
ISHFT	Angle shift. EQ.1: Angle curves are unaltered. EQ.2: Shifts angle data in the LCIDQi curves as necessary to eliminate discontinuities. If angles are confined to the range $[-\pi, \pi]$ and the data contains excursions exceeding π then set ISHFT=2.

Remarks:

1. For the ANGLES Option note the following: If PIDA=0 then the successive rotations are performed about the global axes. If PIDA=PIDB then the rotations are performed about local axes fixed in PIDB. Angles are specified in radians.

2. For the DIRCOS Option the load curves LCIDCij must contain the same number of points with corresponding time values on the abscissa.

Card 2 - Required for EULERP option.

Card 2	1	2	3	4	5	6	7	8
Variable	LCIDE1	LCIDE2	LCIDE3	LCIDE4				
Type	I	I	I	I				
Default	none	none	none	none				

VARIABLE **DESCRIPTION**

LCIDEi Load curve ID specifying Euler parameter e_i as a function of time. The Euler parameters are defined as follows.

$$\varepsilon_i \triangleq \boldsymbol{\varepsilon} \bullet \mathbf{a}_i = \boldsymbol{\varepsilon} \bullet \mathbf{b}_i \quad (i = 1,2,3)$$

$$\varepsilon_4 \triangleq cos\left(\frac{\theta}{2}\right)$$

where $\boldsymbol{\varepsilon}$ is the Euler vector, $\mathbf{a}_i$ and $\mathbf{b}_i$ (i=1,2,3) are dextral sets of unit vectors fixed in PIDA and PIDB, respectively, and θ is the angle associated with the rotation of PIDB in PIDA about Euler vector. If PIDA=0 then $\mathbf{a}_j$ (j=1,2,3) are unit vectors aligned, respectively, with the global axes X, Y, and Z.

Card 2 - Required for VECTOR option.

Card 2	1	2	3	4	5	6	7	8
Variable	LCIDV1	LCIDV2	LCIDV3	LCIDS	VALSPIN			
Type	I	I	I	I	F			
Default	none	none	none	0	0.			

VARIABLE	DESCRIPTION
LCIDVi	Load curve ID specifying the vector measure number v_i as a function of time. The vector measure numbers are defined as follows. $$v_i \triangleq \mathbf{v} \bullet \mathbf{n}_i \quad (i = 1,2,3)$$ where **v** is a vector and $\mathbf{n}_i$ (i=1,2,3) are unit vectors aligned, respectively, with the global axes X, Y, and Z.
LCIDS	Load curve ID which specifies the spin speed of PIDB about an axis parallel to the vector. EQ.0: a constant spin speed as defined by VALSPIN is used, EQ.n: spin speed (radians per unit time) is given by load curve n.
VALSPIN	Value for constant the spin speed of PIDB (radians per unit time). This option is bypassed if the load curve number defined above is non-zero.

BOUNDARY_PRESSURE_OUTFLOW_OPTION*

Available options include:

SEGMENT

SET

Purpose: Define pressure outflow boundary conditions. These boundary conditions are attached to solid elements using the Eulerian ambient formulation (refer to ELFORM in *SECTION_SOLID_ALE) and defined to be pressure outflow ambient elements (refer to AET in *SECTION_SOLID_ALE).

For the SET option define the following card

Card 1	1	2	3	4	5	6	7	8
Variable	SSID							
Type	I							
Default	none							

For the SEGMENT option define the following card

Card 1	1	2	3	4	5	6	7	8
Variable	N1	N2	N3	N4				
Type	I	I	I	I				
Default	none	none	none	none				

VARIABLE	DESCRIPTION
SSID	Segment set ID
N1,N2...	Node ID's defining segment

BOUNDARY_PWP_OPTION*

Available options include:

NODE
SET
TABLE

Purpose: Define pressure boundary conditions for pore water, e.g. at soil surface. The TABLE option applies to a whole Part, while the other options apply to specified nodes.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	*type*ID	LC	CMULT	LCDR	TBIRTH	TDEATH		
Type	I	F	F	I	F	F		
Default	none	none	0.0	None	0.0	1.0e20		

Card 2	1	2	3	4	5	6	7	8
Variable	IPHRE	ITOTEX	IDRFLAG	TABLE				
Type	I	I	I	I				
Default	0	0	0	0				

VARIABLE	DESCRIPTION
*type*ID	Node ID (option=NODE) or Node set ID (option=SET) or Part ID (option=TABLE)
LC	Load curve giving pore water pressure head (length units) vs time. =0: constant pressure head assumed equal to CMULT (leave blank for TABLE option)

CMULT	Factor on curve or constant pressure head if LC=0
LCDR	Load curve giving pore water pressure head during dynamic relaxation. =0: during D.R, use first pressure head value on LC (leave blank for TABLE option)
TBIRTH	Time at which boundary condition becomes active
TDEATH	Time at which boundary condition becomes inactive
IPHRE	Flag =1 for phreatic behaviour (water can be removed by the boundary condition but not added, e.g. at a sloping free surface). Not applicable to TABLE option
ITOTEX	Flag for type of pressure boundary condition: (see notes) =0: Total head =1: Excess head =2: Hydraulic head =4: Z-coord where head=0 (piezometric level)
IDRFLAG	Active flag: =0: Active only in transient analysis =1: Active only in dynamic relaxation =2: Active in all analysis phases (leave blank for TABLE option)
TABLE	Table ID for TABLE option only. See notes below.

Notes:

Pressure is given as pressure head, i.e. pressure/ρg.

NODE and SET options do not affect the pore pressure in Drained parts (the pore pressure for these is set on a part basis and overrides any nodal boundary conditions). The TABLE option should be used only with Drained parts.

TABLE option: The table consists of a list of times in ascending order, followed immediately by curves of z-coordinate versus pore pressure head. Each curve represents the pore water pressure head distribution with z-coordinate at the corresponding time. There must be the same number of curves as time values, arranged immediately after the *DEFINE_TABLE and in the correct order to correspond to the time values. Each curve should be arranged in ascending order of z-coordinate – they look upside-down on the page. The z-coordinate is the x-axis of the curve, the pore water pressure head (in length units) is the y-axis.

“Phreatic” means that water can be removed by the boundary condition but not added. The boundary condition enforces that the pressure head be less than or equal to the stated value. This

condition occurs when the free surface of the soil is sloping so that any water emerging from the soil runs away down the slope.

ITOTEX=0: value from curve or table is total head. This may be used with any pore pressure analysis type.

ITOTEX=1: value from curve or table is excess head. Total head will be determined by adding the hydrostatic head. This option cannot be used with drained analysis, which sets excess head to zero.

ITOTEX=2: value from curve or table is hydraulic head, to which excess head may be added due to volume change in the soil if the analysis type is not drained.

ITOTEX=4: the curve value is the z-coordinate of the water surface; pore pressure head at any node in this boundary condition is given by ($z_{surface}$-z_{node}). This option allows a single boundary condition to be used for nodes at any depth, provided that the pressure distribution is hydrostatic below the given surface. This option is not available for the TABLE option.

*BOUNDARY_PWP_NODE or SET overrides pressure head from
*BOUNDARY_PWP_TABLE at nodes where both are present.

BOUNDARY_RADIATION_OPTION1*_{*OPTION2*}

OPTION1 specifies radiation boundary surface definition by a surface set **(SET)** or by a segment list **(SEGMENT).**

OPTION2 indicates the radiation boundary surface is part of an enclosure. OPTION2 specifies the use of view factors **(VF)** or exchange factors **(EF)** for the radiation calculations. The suffix **(READ)** indicates that the view factors should be read from the file **"viewfl"** or exchange factors from the file **"exchfl"**. The suffix **(CALCULATE)** indicates that the view factors or exchange factors should be calculated. The Stefan Boltzmann constant must be defined for radiation in an enclosure. See *CONTROL_THERMAL_SOLVER.

A list of acceptable keywords are:

***BOUNDARY_RADIATION_SEGMENT**

***BOUNDARY_RADIATION_SEGMENT_VF_READ**

***BOUNDARY_RADIATION_SEGMENT_VF_CALCULATE**

***BOUNDARY_RADIATION_SET**

***BOUNDARY_RADIATION_SET_VF_READ**

***BOUNDARY_RADIATION_SET_VF_CALCULATE**

***BOUNDARY_RADIATION_SET_EF_READ**

***BOUNDARY_RADIATION_SET_EF_CALCULATE**

*BOUNDARY_RADIATION_SEGMENT

Include the following 2 cards for each segment. This keyword defines surface segments that transfer energy by radiation to the environment. Setting TYPE=1 on Card 1 below indicates that the segment transfers energy to the environment.

(Card 1 of 2)

Card 1	1	2	3	4	5	6	7	8
Variable	N1	N2	N3	N4	TYPE			
Type	I	I	I	I	I			
Default	none	none	none	none	1			

(Card 2 of 2)

Card 2	1	2	3	4	5	6	7	8
Variable	FLCID	FMULT	TILCID	TIMULT	LOC			
Type	I	F	I	F	I			
Default	none	0.	none	0.	0			

***BOUNDARY_RADIATION_SEGMENT_VF_READ**

***BOUNDARY_RADIATION_SEGMENT_VF_CALCULATE**

Include the following 2 cards for each segment. This keyword defines surface segments that transfer energy by radiation within an enclosure using view factors. Setting TYPE=2 on Card 1 below specifies that the segment belongs to an enclosure. The file "viewfl" must be present for the **READ** option. The file "viewfl" will be created for the **CALCULATE** option. If the file "viewfl" exists when using the **CACULATE** option, LS-DYNA will terminate with an error message to prevent overwriting the file. The format of "viewfl" is defined at the end of this section.

(Card 1 of 2)

Card 1	1	2	3	4	5	6	7	8
Variable	N1	N2	N3	N4	TYPE	BLOCK	NINT	
Type	I	I	I	I	I	I	I	
Default	none	none	none	none	2	0	0	

(Card 2 of 2)

Card 2	1	2	3	4	5	6	7	8
Variable	SELCID	SEMULT						
Type	I	F						
Default	none	0.						

*BOUNDARY_RADIATION_SET

Include the following 2 cards for each set. This keyword defines surface segment sets that transfer energy by radiation to the environment. Setting TYPE=1 on Card 1 below indicates that the segment transfers energy to the environment.

(Card 1 of 2)

Card 1	1	2	3	4	5	6	7	8
Variable	SSID	TYPE						
Type	I	I						
Default	none	1						

(Card 2 of 2)

Card 2	1	2	3	4	5	6	7	8
Variable	RFLCID	RFMULT	TILCID	TIMULT	LOC			
Type	I	F	I	F	I			
Default	none	0.	none	0.	0			

***BOUNDARY_RADIATION_SET_VF_READ**

***BOUNDARY_RADIATION_SET_VF_CALCULATE**

Include the following 2 cards for each set. This keyword defines surface segment sets that transfer energy by radiation within an enclosure using view factors. Setting TYPE=2 on Card 1 below specifies that the segment set belongs to an enclosure. The file "viewfl" must be present for the **READ** option. The file "viewfl" will be created for the **CALCULATE** option. If the file "viewfl" exists when using the **CACULATE** option, LS-DYNA will terminate with an error message to prevent overwriting the file. The format of "viewfl" is defined at the end of this section.

(Card 1 of 2)

Card 1 1 2 3 4 5 6 7 8

Variable	SSID	TYPE	RAD_GRP	FILE_NO	BLOCK	NINT		
Type	I	I	I	I	I	I		
Default	none	2	0	0	0	0		

(Card 2 of 2)

Card 2 1 2 3 4 5 6 7 8

Variable	SELCID	SEMULT						
Type	I	F						
Default	none	0.						

***BOUNDARY_RADIATION_SET_EF_READ**

***BOUNDARY_RADIATION_SET_EF_CALCULATE**

Include the following card for each set. This keyword defines surface segment sets that transfer energy by radiation within an enclosure using exchange factors. The file "exchfl" must be present for the **READ** option. The file "exchfl" will be created for the **CALCULATE** option. If the file "exchfl" exists when using the **CACULATE** option, LS-DYNA will terminate with an error message to prevent overwriting the file. The format of "exchfl" is defined at the end of this section.

Card 1

1	2	3	4	5	6	7	8
SSID	NMAT	NPHT	ERRMAX				
I	I	I	I				
none	none	1	1.0e-02				

VARIABLE	DESCRIPTION
BLOCK	Flag indicating if this surface blocks the view between any other 2 surfaces. EQ.0: no blocking (default) EQ.1: blocking
ERRMAX	ERRMAX is the convergence error tolerance for the surface.
FILE_NO	File number for view factor file. FILE_NO is added to **viewfl_** to form the name of the file containing the view factors. For example if FILE_NO is specified as 22, then the view factors are read from **viewfl_22**. For radiation enclosure group zero FILE_NO is ignored and view factors are read from **viewfl**. The same file may be used for different radiation enclosure group definitions.
LOC	Shell surface flag for thermal shell elements. See the parameter TSHELL on the *CONTROL_SHELL keyword. EQ.-1: lower surface of thermal shell element EQ. 1: upper surface of thermal shell element
N1, N2, N3, N4	Node ID's defining segment

VARIABLE	DESCRIPTION
NINT	Number of integration points for viewfactor calculation EQ.0: LS-DYNA determines the number of integration points based on the segment size and separation distance $1 \le$ NINT ≤ 10: User specified number
NMAT	NMAT specifies the material type for the portion of the boundary specified by SSID. NMAT must be an exchange factor material ID. See the *EF_MATERIAL keyword.
NPHT	The segments specified by SSID will emit NPHT*NPHOTON photons. See the *EF_CONTROL keyword.
RAD_GRP	Radiation enclosure group ID. The segment sets from all radiation enclosure definitions with the same group ID are augmented to form a single enclosure definition. If RAD_GRP is not specified or set to zero, then the segments are placed in group zero. All segments defined by the SEGMENT option are placed in set zero.
FLCID	Load curve ID for radiation factor f, see *DEFINE_CURVE GT.0: function versus time EQ.0: use constant multiplier value, FMULT LT.0: function versus temperature
FMULT	Curve multiplier for f for use in the equation $\dot{q}'' = \sigma\varepsilon F\left(T_2^4 - T_1^4\right) = f\left(T_{surface}^4 - T_\infty^4\right)$
SELCID	Load curve ID for surface emissivity, see *DEFINE_CURVE GT.0: function versus time EQ.0: use constant multiplier value, SEMULT LT.0: function versus temperature
SEMULT	Curve multiplier for surface emissivity, see *DEFINE_CURVE
SSID	SSID specifies the ID for a set of segments that comprise a portion of, or possibly, the entire enclosure. See *SET_SEGMENT.
TILCID	Load curve ID for T_∞ versus time, see *DEFINE_CURVE: EQ.0: use constant multiplier, TIMULT
TIMULT	Curve multiplier for T_∞
TYPE	Radiation type: EQ.1: Radiation to environment EQ.2: Radiation within an enclosure

Remarks:

The exchange factor, *F*, is a characterization of the effect of the system geometry, emissivity and reflectivity on the capability of radiative transport between surfaces. The radiation boundary condition data cards require specification of the product, $f = F\sigma$, and T_∞ for the boundary surface.

1. When using view factors, the file "viewfl" must exist when using the READ option. This file contains the surface-to-surface area*view factor products (i.e., A_iF_{ij}). The A_iF_{ij} products must be stored in this file by row and formatted as 5E16.0.

2. When using exchange factors, the file "exchfl" must exist when using the READ option. This file contains the surface-to-surface emittance*exchange fraction products (i.e., ε_iF_{ij}). The ε_iF_{ij} products must be stored in this file by row and formatted as 5E16.0.

3. Multiple enclosures can be modeled when using view factors. Consider the following example input. The order of segments in the view factor file follows the order the sets are assigned to the boundary radiation definition.

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$  *BOUNDARY_RADIATION_SET
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Make boundary enclosure radiation groups 8 and 9.
$
*BOUNDARY_RADIATION_SET_VF_READ
*       SSID      TYPE   RAD_GRP   FILE_NO
         15          2         9        10
        1.0       1.0
*BOUNDARY_RADIATION_SET_VF_READ
*       SSID      TYPE   RAD_GRP   FILE_NO
         12          2         9        10
        1.0       1.0
*BOUNDARY_RADIATION_SET_VF_READ
*       SSID      TYPE   RAD_GRP   FILE_NO
         13          2         8        21
        1.0       1.0
$
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

Enclosure radiation group 9 is composed of all the segments in segment set 15 followed by those in segment set 12. The view factors are stored in the file viewfl_10. Enclosure radiation group 8 is composed of the segments in segment set 13. The view factors are stored in the file viewfl_21.

***BOUNDARY_SLIDING_PLANE**

Purpose: Define a sliding symmetry plane. This option applies to continuum domains modeled with solid elements.

Card	1	2	3	4	5	6	7	8
Variable	NSID	VX	VY	VZ	COPT			
Type	I	F	F	F	I			
Default	none	0	0	0	0			

VARIABLE	DESCRIPTION
NSID	Nodal set ID, see *SET_NODE
VX	x-component of vector defining normal or vector
VY	y-component of vector defining normal or vector
VZ	z-component of vector defining normal or vector
COPT	Option: EQ.0: node moves on normal plane, EQ.1: node moves only in vector direction.

Remarks:

Any node may be constrained to move on an arbitrarily oriented plane or line depending on the choice of COPT. Each boundary condition card defines a vector originating at (0,0,0) and terminating at the coordinates defined above. Since an arbitrary magnitude is assumed for this vector, the specified coordinates are non-unique and define only a direction. Use of *BOUNDARY_SPC is preferred over *BOUNDARY_SLIDING_PLANE as the boundary conditions imposed via the latter have been seen to break down somewhat in lengthy simulations owing to numerical roundoff.

BOUNDARY_SPC_OPTION1*_{*OPTION2*}_{*OPTION3*}

OPTION1 is required since it specifies whether the SPC applies to a single node or to a set. The two choices are:

NODE

SET

OPTION2 allows optional birth and death times to be assigned the single node or node set:

BIRTH_DEATH

This option requires one additional line of input.

OPTION3 allows an optional ID to be given that applies either to the single node or to the entire set:

ID

If a heading is defined with the ID, then the ID with the heading will be written at the beginning of the ASCII file, SPCFORC.

Purpose: Define nodal single point constraints. Do not use this option in r-adaptive problems since the nodal point ID's change during the adaptive step. If possible use CONSTRAINED_GLOBAL instead.

The following card is read if and only if the ID option is specified. The second card is required.

Optional 1 2-8

Variable	ID	HEADING
Type	I	A70

	1	2	3	4	5	6	7	8
Variable	NID/NSID	CID	DOFX	DOFY	DOFZ	DOFRX	DOFRY	DOFRZ
Type	I	I	I	I	I	I	I	I
Default	none	0	0	0	0	0	0	0

Read this card if the BIRTH_DEATH option is active.

	1	2	3	4	5	6	7	8
Variable	BIRTH	DEATH						
Type	F	F						
Default	0.0	1.E+20						

VARIABLE	DESCRIPTION
ID	Optional SPC set ID to which this node or node set belongs. This ID does not need to be unique
HEADING	An optional SPC descriptor that will be written into the D3HSP file and the SPCFORC file.
NID/NSID	Node ID or nodal set ID, see *SET_NODE.
CID	Coordinate system ID, see *DEFINE_COORDINATE_SYSTEM.
DOFX	Insert 1 for translational constraint in local x-direction.
DOFY	Insert 1 for translational constraint in local y-direction.
DOFZ	Insert 1 for translational constraint in local z-direction.
DOFRX	Insert 1 for rotational constraint about local x-axis.
DOFRY	Insert 1 for rotational constraint about local y-axis.
DOFRZ	Insert 1 for rotational constraint about local z-axis.
BIRTH	Activation time for constraint.
DEATH	Deactivation time for constraint.

Remarks:

Constraints are applied if a value of 1 is given for DOFxx. A value of zero means no constraint. No attempt should be made to apply SPCs to nodes belonging to rigid bodies (see *MAT_RIGID for application of rigid body constraints).

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$   *BOUNDARY_SPC_NODE
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Make boundary constraints for nodes 6 and 542.
$
*BOUNDARY_SPC_NODE
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$      nid       cid      dofx      dofy      dofz     dofrx     dofry     dofrz
         6         0         1         1         1         1         1         1
       542         0         0         1         0         1         0         1
$
$      Node 6 is fixed in all six degrees of freedom (no motion allowed).
$
$      Node 542 has a symmetry condition constraint in the x-z plane,
$        no motion allowed for y translation, and x & z rotation.
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

***BOUNDARY_SPH_FLOW**

Purpose: Define a flow of particles. This option applies to continuum domains modeled with SPH elements.

Card 1 Format

Card 1	1	2	3	4	5	6	7	8
Variable	ID	STYP	DOF	VAD	LCID	SF	DEATH	BIRTH
Type	I	I	I	I	I	F	F	F
Default	none	none	None	0	none	1.	1.E+20	0.0

Card 2 Format

Card 2	1	2	3	4	5	6	7	8
Variable	NODE	VID						
Type	I	I						
Default	none	0						

VARIABLE	DESCRIPTION
NSID, PID	Nodal set ID (NSID), SEE *SET_NODE, or part ID (PID), see *PART.
STYP	Set type: EQ.1: part set ID, see *SET_PART, EQ.2: part ID, see *PART, EQ.3: node set ID, see *NODE_SET,
DOF	Applicable degrees-of-freedom: EQ.1: x-translational degree-of-freedom, EQ.2: y-translational degree-of-freedom, EQ.3: z-translational degree-of-freedom, EQ.4: translational motion in direction given by the VID. Movement on plane normal to the vector is permitted.

VARIABLE	DESCRIPTION
VAD	Velocity/Acceleration/Displacement flag applied to SPH elements before activation: EQ.0: velocity, EQ.1: acceleration, EQ.2: displacement.
LCID	Load curve ID to describe motion value versus time, see *DEFINE_CURVE.
SF	Load curve scale factor. (default=1.0)
DEATH	Time imposed motion/constraint is removed: EQ.0.0: default set to 10^{20}.
BIRTH	Time imposed motion/constraint is activated.
NODE	Node fixed in space which determines the boundary between activated particles and deactivated particles.
VID	Vector ID for DOF value of 4, see *DEFINE_VECTOR

Remarks:

Initially, the user defines the set of particles that are representing the flow of particles during the simulation. At time t=0, all the particles are deactivated which means that no particle approximation is calculated. The boundary of activation is a plane determined by the NODE, and normal to the vector VID. The particles are activated when they reached the boundary. Since they are activated, particle approximation is started.

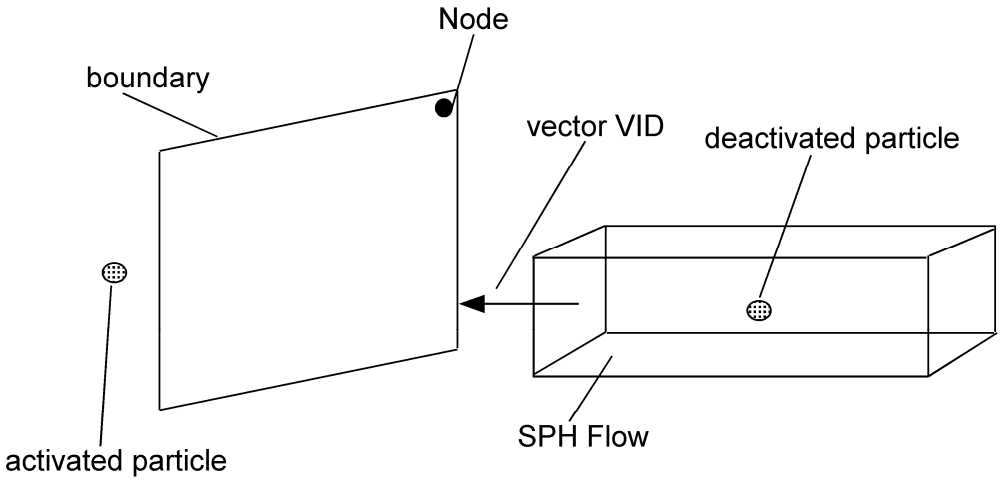


Figure 3.9. Vector **VID** determines the orientation of the SPH flow.

*BOUNDARY_SPH_SYMMETRY_PLANE

Purpose: Define a symmetry plane for SPH. This option applies to continuum domains modeled with SPH elements.

Card	1	2	3	4	5	6	7	8
Variable	VTX	VTY	VTZ	VHX	VHY	VHZ		
Type	F	F	F	F	F	F		
Default	0.	0.	0.	0.	0.	0.		

VARIABLE	DESCRIPTION
VTX	x-coordinate of tail of a normal vector originating on the wall (tail) and terminating in the body (head) (i.e., vector points from the symmetry plane into the body).
VTY	y-coordinate of tail
VTZ	z-coordinate of tail
VHX	x-coordinate of head
VHY	y-coordinate of head
VHZ	z-coordinate of head

Remarks:

1. A plane of symmetry is assumed for all SPH elements defined in the model.

2. The plane of symmetry has to be normal to either the x, y or z direction.

*BOUNDARY_SYMMETRY_FAILURE

Purpose: Define a symmetry plane with a failure criterion. This option applies to continuum domains modeled with solid elements.

Card	1	2	3	4	5	6	7	8
Variable	SSID	FS	VTX	VTY	VTZ	VHX	VHY	VHZ
Type	I	F	F	F	F	F	F	F
Default	none	0.	0.	0.	0.	0.	0.	0.

VARIABLE	DESCRIPTION
SSID	Segment set ID, see *SET_SEGMENT
FS	Tensile failure stress > 0.0. The average stress in the elements surrounding the boundary nodes in a direction perpendicular to the boundary is used.
VTX	x-coordinate of tail of a normal vector originating on the wall (tail) and terminating in the body (head) (i.e., vector points from the symmetry plane into the body).
VTY	y-coordinate of tail
VTZ	z-coordinate of tail
VHX	x-coordinate of head
VHY	y-coordinate of head
VHZ	z-coordinate of head

Remarks:

A plane of symmetry is assumed for the nodes on the boundary at the tail of the vector given above. Only the motion perpendicular to the symmetry plane is constrained. After failure the nodes are set free.

BOUNDARY_TEMPERATURE_*OPTION

Available options include:

NODE

SET

Purpose: Define temperature boundary conditions for a thermal or coupled thermal/structural analysis.

Card	1	2	3	4	5	6	7	8
Variable	NID/SID	LCID	CMULT	LOC	FID			
Type	I	I	F	I				
Default	none	0	0.	0				

VARIABLE	DESCRIPTION
NID/SID	Node ID/Node Set ID, see *SET_NODE_*OPTION*
LCID	Load curve ID for temperature versus time: EQ.0: use the constant multiplier value given below by CMULT.
CMULT	Curve multiplier for temperature
LOC	Application of surface for thermal shell elements, see parameter, TSHELL, in the *CONTROL_SHELL input: EQ.-1: lower surface of thermal shell element EQ.0: middle surface of thermal shell element EQ.1: upper surface of thermal shell element
FID	Function ID

Remarks:

If no load curve ID is given, then a constant boundary temperature is assumed. CMULT is also used to scale the load curve values.

*BOUNDARY_THERMAL_WELD

Purpose: Define a moving heat source to model welding. Only applicable for a coupled thermal-structural simulations in which the weld source or work piece is moving.

Card 1 Format

Card 1	1	2	3	4	5	6	7	8
Variable	PID	PTYP	NID	NFLAG	X0	Y0	Z0	N2ID
Type	I	I	I	I	F	F	F	I
Default	none	1	none	1	none	none	none	none

Card 2 Format

Card 2	1	2	3	4	5	6	7	8
Variable	a	b	c_f	c_r	LCID	Q	F_f	F_r
Type	F	F	F	F	I	F	F	F
Default	none	none	none	none	none	none	none	none

Optional Card 3 Format (define this card only if N2ID = -1 on card 1 above)

Card 3	1	2	3	4	5	6	7	8
Variable	tx	ty	tz					
Type	F	F	F					
Default	none	none	none					

VARIABLE	DESCRIPTION
PID	Part ID or Part Set ID to which weld source is applied
PTYP	PID type: EQ.1: PID defines a single part ID EQ.2: PID defines a part set ID
NID	Node ID giving location of weld source EQ.0: location defined by (X0,Y0,Z0) below
NFLAG	Flag controlling motion of weld source EQ.1: source moves with node NID EQ.2: source is fixed in space at original position of node NID
X0,Y0,Z0	Coordinates of weld source, which remains fixed in space (optional, ignored if NID nonzero above)
N2ID	Second node ID for weld beam aiming direction GT.0: beam is aimed from N2ID to NID, moves with these nodes EQ.-1: beam aiming direction is (tx,ty,tz) input on optional card 3
a	Weld pool width
b	Weld pool depth (in beam aiming direction)
c_f	Weld pool forward direction
c_r	Weld pool rearward direction
LCID	Load curve ID for weld energy input rate vs. time EQ.0: use constant multiplier value Q.
Q	Curve multiplier for weld energy input rate [energy/time, e.g., Watt]
F_f	Forward distribution function
F_r	Rear distribution function (Note: $F_f + F_r = 2.0$)
tx,ty,tz	Weld beam direction vector in global coordinates (N2ID = -1 only)

Remarks:

This boundary condition allows simulation of a moving weld heat source, following the work of Goldak, Chakravarti, and Bibby [1984]. Heat is generated in an ellipsoidal region centered at the weld source, and decaying exponentially with distance according to:

$$q = \frac{6\sqrt{3}FQ}{\pi\sqrt{\pi}abc} e^{\left(\frac{-3x^2}{a^2}\right)} e^{\left(\frac{-3y^2}{b^2}\right)} e^{\left(\frac{-3z^2}{c^2}\right)}$$

where:

$q =$ weld source power density

$(x, y, z) =$ coordinates of point p in weld material

$$F = \begin{cases} \mathrm{F_f} \text{ if point } p \text{ is in front of beam} \\ \mathrm{F_r} \text{ if point } p \text{ is behind beam} \end{cases}$$

$$c = \begin{cases} \mathrm{c_f} \text{ if point } p \text{ is in front of beam} \\ \mathrm{c_r} \text{ if point } p \text{ is behind beam} \end{cases}$$

A local coordinate system is constructed which is centered at the heat source. The relative velocity vector of the heat source defines the "forward" direction, so material points that are approaching the heat source are in "front" of the beam. The beam aiming direction is used to compute the weld pool depth. The weld pool width is measured normal to the relative velocity - aiming direction plane.

***BOUNDARY_USA_SURFACE**

Purpose: Define a surface for coupling with the USA boundary element code [DeRuntz 1993]. The outward normal vectors should point into the fluid media.

Card	1	2	3	4	5	6	7	8
Variable	SSID	WETDRY	NBEAM					
Type	I	I	I					
Default	none	0	0					

VARIABLE	DESCRIPTION
SSID	Segment set ID, see *SET_SEGMENT
WETDRY	Wet surface flag: EQ.0: dry, no coupling, EQ.1: wet, coupled with USA.
NBEAM	The number of nodes touched by USA Surface-of-Revolution (SOR) elements. It is not necessary that the LS-DYNA model has beams where USA has beams (i.e., SOR elements), merely that the LS-DYNA model has nodes to receive the forces that USA will return.

Remarks:

The wet surface of 3 and 4-noded USA General boundary elements is defined in LS-DYNA with a segment set of 4-noded surface segments, where the fourth node can duplicate the third node to form a triangle. The segment normals should be directed into the USA fluid. If USA overlays are going to be used to reduce the size of the DAA matrices, the user should nonetheless define the wet surface here as if no overlay were being used. If Surface-of -Revolution elements (SORs) are being used in USA, then NBEAM should be non-zero on one and only one card in this section.

When running a coupled problem with USA, the procedure involves several steps. First, LS-DYNA is executed to create a LS-DYNA dump file "d3dump" and a linking file "strnam" which contains the nodal grid point data and wet segment connectivity data for the FLUMAS processor, and the dof-equation table and structural mass vector for the AUGMAT processor. "Dyna.pre" is denoted "grdnam" in the FLUMAS manual and "strnam" in the AUGMAT manual. The execution line in the first step is:

LS-DYNA memory=nwds i=inputfilename > outputfilename

where "inputfilename" is the LS-DYNA input file.

In the second step, the DAA fluid mass matrix is created through execution of the USA FLUMAS processor:

FLUMAS -m nwds < flumasinputfilename > flumasoutputfilename

In the third step, the modified augmented DAA equations for the coupled problem are calculated and saved through execution of the USA AUGMAT processor:

AUGMAT -m nwds < augmatinputfilename > augmatoutputfilename

This step is repeated whenever one wishes to change DAA formulations.

In the fourth step the actual coupled time-integration is conducted using the execution line:

LS-DYNA memory=nwds r=d3dump usa=usainputfilename > outputfilename

The input files, flumasinputfilename, augmatinputfilename, and usainputfilename, are prepared in accordance with the USA code documentation.

It is advisable when running coupled problems to check the ASCII output files to ensure that each run completed normally.

*CASE

This keyword option provides a way of running multiple load cases sequentially. Within each case, the input parameters, which include loads, boundary conditions, control cards, contact definitions, initial conditions, etc., can change. If desired, the results from a previous case can be used during initialization. Each case creates unique file names for all output results files by appending "**CID*n***." to the default file name.

Note: To use the *CASE keyword requires a stand alone program that is started by the LS-DYNA executable. To obtain that program contact LSTC or your local distributor.

***CASE_{***OPTION***}**

Available options include:

<**BLANK**>

BEGIN_CID*n*

END_CID*n*

Purpose: Begin a case definition. In options *CASE_BEGIN_ and *CASE_END_, CID*n* is a numeric identification number that may or may not be active for a particular case. All keywords between *CASE_BEGIN_CID*n* and *CASE_END_CID*n* will be included for the case if CID*n* is activated for the case. See the definition of the *CASE keyword below. *CASE_BEGIN/*CASE_END can be nested, overlapped, and disjointed. An example below demonstrates the use of these options. However convenient, the optional keywords, BEGIN_ and END_ are not necessary. All keywords cards that contain an ID that are active for a case are included. Any *non-case* keyword card can be tagged with an ID by adding the string "CID=CID*n*." This keyword will then be active only for those cases having this CID in their active list. There can be more than one space before the CID= identifier. Any keyword without CID= is active for all cases.

Card 1	1	2	3	4	5	6	7	8
Variable	CASEID							
Type	I							
Default	None							

*CASE

Optional: Define if additional command line arguments are required for this case ID.

Cards 2, …	1	2	3	4	5	6	7	8
Variable	COMMANDS							
Type	A							
Default	Not Required							

Define as many Active IDs for this case. Use as many cards as necessary.

Cards …	1	2	3	4	5	6	7	8
Variable	CID1	CID2	…	…	…	…	CID7	…
Type	I	I	I	I	I	I	I	I
Default	none	none	none	none	none	none	none	None

VARIABLE	DESCRIPTION
CASEID	Identification number for CASE.
COMMANDS	Command line arguments.
CID*n*	Active numeric ID for case, CASEID. Any non-case keyword card can be tagged with a case ID by adding the string "CID=CID*n*. This keyword will then be active only for those cases having this CID in their active list. There can be more than one space before the CID= identifier. Any keyword without CID= is active for all cases.

Example:

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$  *CASE
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
*CASE
$
$...>....1...>....2...>....3...>....4...>....5...>....6...>....7
$   CASEID
       101
MEMORY=20M
         1
$
*CASE
$   CASEID
       102
MEMORY=20M  NCYCLE=1845
$
*TITLE  CID=1
THIS IS THE TITLE FOR CASE 101
$
*TITLE  CID=5
THIS IS THE TITLE FOR CASE 102
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
```

Example:

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$  *CASE_BEGIN_5
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
*CASE_BEGIN_5
$...>....1...>....2...>....3...>....4...>....5...>....6...>....7....>
*DATABASE_BINARY_D3THDT
1.e-5
*CASE_BEGIN_3
*DATABASE_NODOUT
1.e-5
*CASE_END_5
*DATABASE_ELOUT
1.e-5
*CASE_END_3
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
```

The last example will generate D3THDT and NODOUT for CID=5, and NODOUT and ELOUT for CID=3. If there are no CASE statements, but there are *CASE_BEGIN/*CASE_END statements, then one case is created for each unique ID on the *CASE_BEGIN lines. Since optional command line arguments may be required, *CASE_BEGIN can optionally be followed by extra command line arguments.

If *CASE statements are used, cases are not generated for each *CASE_BEGIN, but only for each *CASE. If no *CASE keywords appear, it is an error to have CID= on any keyword. If multiple *CASE or *CASE_BEGIN statements appear that have the same ID, their command line arguments and active id lists are merged.

*COMPONENT

The keyword *COMPONENT provides a way of incorporating specialized components and features. The keyword control cards in this section are defined in alphabetical order:

COMPONENT_GEBOD_OPTION*

COMPONENT_GEBOD_JOINT_OPTION*

***COMPONENT_HYBRIDIII**

COMPONENT_HYBRIDIII_JOINT_OPTION*

COMPONENT_GEBOD_OPTION*

Purpose: Generate a rigid body dummy based on dimensions and mass properties from the GEBOD database. The motion of the dummy is governed by equations integrated within LS-DYNA separately from the finite element model. Default joint characteristics (stiffnesses, stop angles, etc.) are set internally and should give reasonable results, however, they may be altered using the *COMPONENT_GEBOD_JOINT command. Contact between the segments of the dummy and the finite element model is defined using the *CONTACT_GEBOD command. The use of a positioning file is essential with this feature, see Appendix N for further details.

OPTION specifies the human subject type. The male and female types represent adults while the child is genderless.

MALE

FEMALE

CHILD

(Card 1 of 2)

Card 1	1	2	3	4	5	6	7	8
Variable	DID	UNITS	SIZE					
Type	I	I	F					
Default	none	none	none					

VARIABLE	DESCRIPTION
DID	Dummy ID. A unique number must be specified.
UNITS	System of units used in the finite element model. EQ.1: lbf*sec^2/in - inch - sec EQ.2: kg - meter - sec EQ.3: kgf*sec^2/mm - mm - sec EQ.4: metric ton - mm - sec EQ.5: kg - mm - msec
SIZE	Size of the dummy. This represents a combined height and weight percentile ranging from 0 to 100 for the male and female types. For the child the number of months of age is input with an admissible range from 24 to 240.

(Card 2 of 2)

Card 2	1	2	3	4	5	6	7	8
Variable	VX	VY	VZ	GX	GY	GZ		
Type	F	F	F	F	F	F		
Default	0.	0.	0.	0.	0.	0.		

VARIABLE	DESCRIPTION
VX,VY,VZ	Initial velocity of the dummy in the global x, y and z directions.
GX,GY,GZ	Global x, y, and z components of gravitational acceleration applied to the dummy.

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$   *COMPONENT_GEBOD_MALE
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  A 50th percentile male dummy with the ID number of 7 is generated in the
$  lbf*sec^2-inch-sec system of units.  The dummy is given an initial velocity of
$  616 in/sec in the negative x direction and gravity acts in the negative z
$  direction with a value 386 in/sec^2.
$
*COMPONENT_GEBOD_MALE
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$      did     units      size
         7         1        50
$       vx        vy        vz        gx        gy        gz
      -616         0         0         0         0      -386
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

COMPONENT_GEBOD_JOINT_OPTION*

Purpose: Alter the joint characteristics of a GEBOD rigid body dummy. Setting a joint parameter value to zero retains the default value set internally. See Appendix N for further details.

The following options are available.

PELVIS

WAIST

LOWER_NECK

UPPER_NECK

LEFT_SHOULDER

RIGHT_SHOULDER

LEFT_ELBOW

RIGHT_ELBOW

LEFT_HIP

RIGHT_HIP

LEFT_KNEE

RIGHT_KNEE

LEFT_ANKLE

RIGHT_ANKLE

Card 1 - Required.

Card 1	1	2	3	4	5	6	7	8
Variable	DID	LC1	LC2	LC3	SCF1	SCF2	SCF3	
Type	F	I	I	I	F	F	F	

VARIABLE	DESCRIPTION
DID	Dummy ID, see *COMPONENT_GEBOD_*OPTION*.
LCi	Load curve ID specifying the loading torque versus rotation (in radians) for the i-th degree of freedom of the joint.
SCFi	Scale factor applied to the load curve of the i-th joint degree of freedom.

Card 2 - Required.

Card 2	1	2	3	4	5	6	7	8
Variable	C1	C2	C3	NEUT1	NEUT2	NEUT3		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
Ci	Linear viscous damping coefficient applied to the i-th DOF of the joint. Units are torque*time/radian, where the units of torque and time depend on the choice of UNITS in card 1 of *COMPONENT_GEBOD_ *OPTION*.
NEUTi	Neutral angle (degrees) of joint's i-th DOF.

Card 3 - Required.

Card 3	1	2	3	4	5	6	7	8
Variable	LOSA1	HISA1	LOSA2	HISA2	LOSA3	HISA3		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
LOSAi	Value of the low stop angle (degrees) for the i-th DOF of this joint.
HISAi	Value of the high stop angle (degrees) for the i-th DOF of this joint.

Card 4 - Required.

Card 4	1	2	3	4	5	6	7	8
Variable	UNK1	UNK2	UNK3					
Type	F	F	F					
Default	0.	0.	0.					

VARIABLE	DESCRIPTION
UNKi	Unloading stiffness (torque/radian) for the i-th degree of freedom of the joint. This must be a positive number. Units of torque depend on the choice of UNITS in card 1 of *COMPONENT_GEBOD_*OPTION*.

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$   *COMPONENT_GEBOD_JOINT_LEFT_SHOULDER
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  The damping coefficients applied to all three degrees of freedom of the left
$  shoulder of dummy 7 are set to 2.5.  All other characteristics of this joint
$  remain set to the default value.
$
*COMPONENT_GEBOD_JOINT_LEFT_SHOULDER
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$      did       lc1       lc2       lc3      scf1      scf2      scf3
         7         0         0         0         0         0         0
$       c1        c2        c3     neut1     neut2     neut3
       2.5       2.5       2.5         0         0         0
$    losa1     hisa1     losa2     hisa2     losa3     hisa3
         0         0         0         0         0         0
$     unk1      unk2      unk3
         0         0         0
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$


$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$   *COMPONENT_GEBOD_JOINT_WAIST
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Load curve 8 gives the torque versus rotation relationship for the 2nd DOF
$  (lateral flexion) of the waist of dummy 7.  Also, the high stop angle of the
$  1st DOF (forward flexion) is set to 45 degrees.  All other characteristics
$  of this joint remain set to the default value.
$
*COMPONENT_GEBOD_JOINT_WAIST
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$      did       lc1       lc2       lc3      scf1      scf2      scf3
         7         0         8         0         0         0         0
$       c1        c2        c3     neut1     neut2     neut3
         0         0         0         0         0         0
$    losa1     hisa1     losa2     hisa2     losa3     hisa3
         0        45         0         0         0         0
$     unk1      unk2      unk3
         0         0         0
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

*COMPONENT_HYBRIDIII

Purpose: Define a HYBRID III dummy. The motion of the dummy is governed by equations integrated within LS-DYNA separately from the finite element model. The dummy interacts with the finite element structure through contact interfaces. Joint characteristics (stiffnesses, damping, friction, etc.) are set internally and should give reasonable results, however, they may be altered using the *COMPONENT_HYBRIDIII_JOINT command. Joint force and moments can be written to an ASCII file (see *DATABASE_H3OUT).

Card 1	1	2	3	4	5	6	7	8
Variable	DID	SIZE	UNITS	DEFRM	VX	VY	VZ	
Type	I	I	I	I	F	F	F	
Default	none	none	none	1	0.	0.	0.	

VARIABLE	DESCRIPTION
DID	Dummy ID. A unique number must be specified.
SIZE	Size of dummy. EQ.1: 5th percentile adult EQ.2: 50th percentile adult EQ.3: 95th percentile adult Note: If negative then the best of currently available joint properties are applied.
UNITS	System of units used in the finite element model. EQ.1: lbf*sec^2/in - inch - sec EQ.2: kg - meter - sec EQ.3: kgf*sec^2/mm - mm - sec EQ.4: metric ton - mm - sec EQ.5: kg - mm - msec
DEFRM	Deformability type. EQ.1: all dummy segments entirely rigid EQ.2: deformable abdomen (low density foam, mat #57) EQ.3: deformable jacket (low density foam, mat #57) EQ.4: deformable headskin (viscoelastic, mat #6) EQ.5: deformable abdomen/jacket EQ.6: deformable jacket/headskin EQ.7: deformable abdomen/headskin EQ.8: deformable abdomen/jacket/headskin
VX,VY,VZ	Initial velocity of the dummy in the global x, y and z directions.

Card 2	1	2	3	4	5	6	7	8
Variable	HX	HY	HZ	RX	RY	RZ		
Type	F	F	F	F	F	F		
Default	0.	0.	0.	0.	0.	0.		

VARIABLE	DESCRIPTION
HX,HY,HZ	Initial global x,y, and z coordinate values of the H-point.
RX,RY,RZ	Initial rotation of dummy about the H-point with respect to the global x,y, and z axes (degrees).

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$  *COMPONENT_HYBRIDIII
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  A 50th percentile adult rigid HYBRID III dummy with an ID number of 7 is defined
$  in the lbf*sec^2-inch-sec system of units. The dummy is assigned an initial
$  velocity of 616 in/sec in the negative x direction. The H-point is initially
$  situated at (x,y,z)=(38,20,0) and the dummy is rotated 18 degrees about the
$  global x-axis.
$
*COMPONENT_HYBRIDIII
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$      did    . size     units     defrm        vx        vy        vz
         7         2         1         1     -616.        0.        0.
$       hx        hy        hz        rx        ry        rz
        38.       20.        0.       18.        0.        0.
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

COMPONENT_HYBRIDIII_JOINT_*OPTION

Purpose: Alter the joint characteristics of a HYBRID III dummy. Setting a joint parameter value to zero retains the default value set internally. Joint force and moments can be written to an ASCII file (see *DATABASE_H3OUT). Further details pertaining to the joints are found in the Hybrid III Dummies section of Appendix N.

The following options are available:

LUMBAR	**RIGHT_ELBOW**	**RIGHT_KNEE**
LOWER_NECK	**LEFT_WRIST**	**LEFT_ANKLE**
UPPER_NECK	**RIGHT_WRIST**	**RIGHT_ANKLE**
LEFT_SHOULDER	**LEFT_HIP**	**STERNUM**
RIGHT_SHOULDER	**RIGHT_HIP**	**LEFT_KNEE_SLIDER**
LEFT_ELBOW	**LEFT_KNEE**	**RIGHT_KNEE_SLIDER**

Card 1 - Required.

Card 1	1	2	3	4	5	6	7	8
Variable	DID	Q1	Q2	Q3	FRIC			
Type	F	F	F	F	F			

Card 2 - Required.

Card 2	1	2	3	4	5	6	7	8
Variable	C1	ALO1	BLO1	AHI1	BHI1	QLO1	QHI1	SCLK1
Type	F	F	F	F	F	F	F	F

Card 3 - Required. Left blank if joint has only one degree of freedom.

Card 3	1	2	3	4	5	6	7	8
Variable	C2	ALO2	BLO2	AHI2	BHI2	QLO2	QHI2	SCLK2
Type	F	F	F	F	F	F	F	F

Card 4 - Required. Left blank if the joint has only two degrees of freedom.

Card 4	1	2	3	4	5	6	7	8
Variable	C3	ALO3	BLO3	AHI3	BHI3	QLO3	QHI3	SCLK3
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
DID	Dummy ID, see *COMPONENT_HYBRIDIII
Qi	Initial value of the joint's i-th degree of freedom. Units of degrees are defined for rotational DOF. See Appendix N for a listing of the applicable DOF.
FRIC	Friction load on the joint.
Ci	Linear viscous damping coefficient applied to the i-th DOF of the joint.
ALOi	Linear coefficient for the low regime spring of the joint's i-th DOF.
BLOi	Cubic coefficient for the low regime spring of the joint's i-th DOF.
AHIi	Linear coefficient for the high regime spring of the joint's i-th DOF.
BHIi	Cubic coefficient for the high regime spring of the joint's i-th DOF.
QLOi	Value at which the low regime spring definition becomes active.
QHIi	Value at which the high regime spring definition becomes active.
SCLKi	Scale value applied to the stiffness of the joint's i-th DOF (default=1.0).

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$  *COMPONENT_HYBRIDIII_JOINT_LEFT_ANKLE
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  The damping coefficients applied to all three degrees of freedom of the left
$  ankle of dummy 7 are set to 2.5.  All other characteristics of this joint
$  remain set to the default value.  The dorsi-plantar flexion angle is set to
$  20 degrees.
$
*COMPONENT_HYBRIDIII_JOINT_LEFT_ANKLE
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$      did       q1       q2       q3     fric
         7        0      20.        0        0        0
$       c1     alo1     blo1     ahi1     bhi1     qlo1     qhi1
       2.5        0        0        0        0        0        0
$       c2     alo2     blo2     ahi2     bhi2     qlo2     qhi2
       2.5        0        0        0        0        0        0
$      2.5     alo3     blo3     ahi3     bhi3     qlo3     qhi3
```

*CONSTRAINED

The keyword *CONSTRAINED provides a way of constraining degrees of freedom to move together in some way. The keyword cards in this section are defined in alphabetical order:

***CONSTRAINED_ADAPTIVITY**

***CONSTRAINED_BUTT_WELD**

***CONSTRAINED_EULER_IN_EULER**

CONSTRAINED_EXTRA_NODES_OPTION*

CONSTRAINED_GENERALIZED_WELD_OPTION***_{***OPTION***}**

***CONSTRAINED_GLOBAL**

***CONSTRAINED_INTERPOLATION_{***OPTION***}**

CONSTRAINED_JOINT_OPTION***_{***OPTION***}_{***OPTION***}_{***OPTION***}**

CONSTRAINED_JOINT_AUTO_OPTION***_{***OPTION***}_{***OPTION***}_{***OPTION***}**

CONSTRAINED_JOINT_STIFFNESS_*OPTION

***CONSTRAINED_LAGRANGE_IN_SOLID**

***CONSTRAINED_LINEAR_GLOBAL**

***CONSTRAINED_LINEAR_LOCAL**

***CONSTRAINED_LOCAL**

***CONSTRAINED_NODAL_RIGID_BODY_{***OPTION***}_{***OPTION***}**

***CONSTRAINED_NODE_SET_{***OPTION***}**

***CONSTRAINED_POINTS**

***CONSTRAINED_RIGID_BODIES**

***CONSTRAINED_RIGID_BODY_STOPPERS**

***CONSTRAINED_RIVET_{***OPTION***}**

***CONSTRAINED_SHELL_TO_SOLID**

***CONSTRAINED_SPLINE**

***CONSTRAINED_SPOTWELD_{***OPTION***}_{***OPTION***}**

***CONSTRAINED_TIE-BREAK**

***CONSTRAINED_TIED_NODES_FAILURE**

*CONSTRAINED_ADAPTIVITY

Purpose: Define an adaptive constraint which constrains a node to the midpoint along an edge of a shell element. This keyword is also created by LS-DYNA during an adaptive calculation. This option applies to shell elements.

Card	1	2	3	4	5	6	7	8
Variable	SN	MN1	MN2					
Type	I	I	I					
Default	none	none	none					

VARIABLE	DESCRIPTION
SN	Slave node. This is the node constrained at the midpoint of an edge of a shell element.
MN1	One node along the edge of the shell element.
MN2	The second node along the edge.

*CONSTRAINED_BUTT_WELD

Purpose: Define a line of coincident nodes that represent a structural butt weld between two parts defined by shell elements. Failure is based on nodal plastic strain for ductile failure and stress resultants for brittle failure. This input is much simpler than the alternative approach for defining butt welds, see *CONSTRAINED_GENERALIZED_WELD_BUTT. The local coordinate system, the effective length, and thickness for each pair of butt welded nodes are determined automatically in the definition below. In the GENERALIZED option these quantities must be defined in the input.

Card 1 2 3 4 5 6 7 8

Variable	SNSID	MNSID	EPPF	SIGF	BETA			
Type	I	I	F	F	F			
Default	none	none	0.	1.e+16	1.0			
Remarks		1, 2	3, 4	3	3			

VARIABLE	DESCRIPTION
SNSID	Slave node set ID, see *SET_NODE_*OPTION*.
MNSID	Master node set ID, see *SET_NODE_*OPTION*.
EPPF	Plastic strain at failure
SIGF	σ_f , stress at failure for brittle failure.
BETA	β , failure parameter for brittle failure.

Remarks:

1. Nodes in the master and slave sets must be given in the order they appear as one moves along the edge of the surface. An equal number of coincident nodes must be defined in each set. In a line weld the first and last node in a string of nodes can be repeated in the two sets. If the first and last pair of nodal points are identical, a circular or closed loop butt weld is assumed. See Figure 6.1, where the line butt weld and closed loop weld are illustrated.

2. Butt welds may not cross. For complicated welds, this option can be combined with the input in *CONSTRAINED_GENERALIZED_WELD_BUTT to handle the case where crossing occurs. Nodes in a butt weld must not be members of rigid bodies.

3. If the average volume-weighted effective plastic strain in the shell elements adjacent to a node exceeds the specified plastic strain at failure, the node is released. Brittle failure of the butt welds occurs when:

$$\beta\sqrt{\sigma_n^2 + 3\left(\tau_n^2 + \tau_t^2\right)} \geq \sigma_f$$

where

σ_n = normal stress (local x)
τ_n = shear stress in direction of weld (local y)
τ_t = shear stress normal to weld (local z)
σ_f = failure stress
β = failure parameter

Component σ_n is nonzero for tensile values only. The nodes defining the slave and master sides of the butt weld must coincide. The local z-axis at a master node is normal to the *master side* plane of the butt weld at the node, and the local y-axis is taken as the vector in the direction of a line connecting the mid-points of the line segments lying on either side of the master node. The normal vector is found by summing the unit normal vectors of all shell elements on the *master side* sharing the butt welded node. The direction of the normal vector at the node is chosen so that the x-local vector points towards the elements on the slave side in order to identify tensile versus compressive stresses. The thickness of the butt weld and length of the butt weld are needed to compute the stress values. The thickness is based on the average thickness of the shell elements that share the butt welded nodal pair, and the chosen length of the butt weld is shown in Figure 6.1.

4. Butt welds may be used to simulate the effect of failure along a predetermined line, such as a seam or structural joint. When the failure criterion is reached at a nodal pair, the nodes begin to separate. As this effect propagates, the weld will appear to “unzip,” thus simulating failure of the connection.

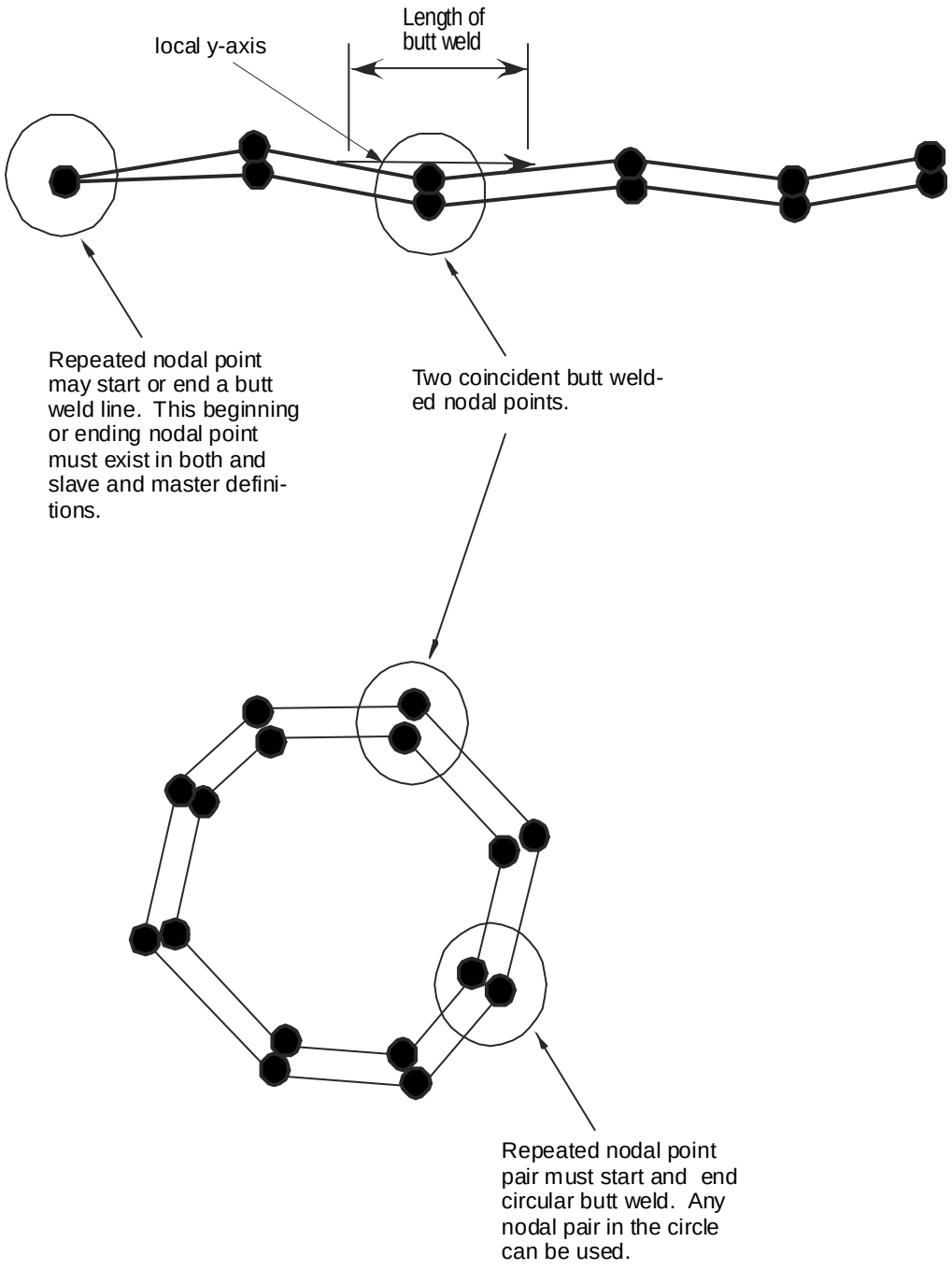


Figure 6.1. Definition of butt welds are shown above. The butt weld can be represented by a line of nodal points or by a closed loop.

*CONSTRAINED_EULER_IN_EULER

Purpose: This command defines the coupling interaction between EULERIAN materials in two overlapping, geometrically similar, multi-material Eulerian mesh sets. The command allows a frictionless "contact" between two or more different Eulerian materials.

Card	1	2	3	4	5	6	7	8
Variable	PSIDSLV	PSIDMST	PFAC					
Type	I	I	F					
Default	0	0	0.1					

VARIABLE	DESCRIPTION
PSIDSLV	Part set ID of the 1st ALE or Eulerian set of mesh(es) (slave).
PSIDMST	Part set ID of the 2nd ALE or Eulerian set of mesh(es) (master).
PFAC	A penalty factor for the coupling interaction between the two PSIDs.

Remarks:

1. The 2 meshes must be of Eulerian formulation (the meshes are fixed in space, not moving). Consider 2 overlapping Eulerian meshes. Each Eulerian mesh contains 2 physical materials, say a vacuum and a metal. This card provides a frictionless "contact" or interaction between the 2 metals, each resides in a different Eulerian mesh system. Due to its restrictive nature, this option is currently only an experimental feature.

2. Contact pressure is built up in two overlapping Eulerian elements if their combined material fill fraction exceeds 1.0 (penalty formulation).

3. This feature needs to be combined with *MAT_VACUUM (element formulation 11).

Example:

Consider an ALE/Eulerian multi-material model (ELFORM=11) consisting of:

- PID 1 = *MAT_NULL (material 1)
- PID 2 = *MAT_VACUUM $\Rightarrow$ PID 1 is merged at its boundary to PID 2.
- PID 3 = *MAT_NULL (material 3)
- PID 4 = *MAT_VACUUM $\Rightarrow$ PID 3 is merged at its boundary to PID 4.

The mesh set containing PID 1 & 2 intersects or overlaps with the mesh set containing PID 3 & 4. PID 1 is given an initial velocity in the positive x direction. This will cause material 1 to contact material 3 (note that materials 2 & 4 are void). The interaction between materials 1 & 3 is possible by defining this coupling command. In this case material 1 can flow within the mesh region of PID 1 & 2 only, and material 3 can flow within the mesh region of PID 3 & 4 only.

```
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|...8
*ALE_MULTI-MATERIAL_GROUP
$       SID   SIDYTPE
          1         1
          2         1
          3         1
          4         1
*CONSTRAINED_EULER_IN_EULER
$     PSID1     PSID2     PENAL
         11        12       0.1
*SET_PART_LIST
         11
          1         2
*SET_PART_LIST
         12
          3         4
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|...8
```

CONSTRAINED_EXTRA_NODES_*OPTION

Available options include:

NODE

SET

Purpose: Define extra nodes for rigid body.

Card	1	2	3	4	5	6	7	8
Variable	PID	NID/NSID	IFLAG					
Type	I	I	I					
Default	none	none	0					

VARIABLE	DESCRIPTION
PID	Part ID of rigid body to which the nodes will be added, see *PART.
NID/NSID	Node (option: _NODE) or node set ID (option: _SET), see *SET_ NODE, of added nodes.
IFLAG	This flag is meaningful if and only if the inertia properties of the Part ID are defined in PART_INERTIA. If set to unity, the center-of-gravity, the translational mass, and the inertia matrix of the PID will be updated to reflect the merged nodal masses of the node or node set. If IFLAG is defaulted to zero, the merged nodes will not affect the properties defined in PART_INERTIA since it is assumed the properties already account for merged nodes.

Remarks:

Extra nodes for rigid bodies may be placed anywhere, even outside the body, and they are assumed to be part of the rigid body. They have many uses including:

1. The definition of draw beads in metal forming applications by listing nodes along the draw bead.

2. Placing nodes where joints will be attached between rigid bodies.

3. Defining a node where point loads are to be applied or where springs may be attached.

4. Defining a lumped mass at a particular location.

and so on. The coordinates of the extra nodes are updated according to the rigid body motion.

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$   *CONSTRAINED_EXTRA_NODES_NODE
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Rigidly attach nodes 285 and 4576 to part 14. (Part 14 MUST be a rigid body.)
$
*CONSTRAINED_EXTRA_NODES_NODE
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$      pid       nid
        14       285
        14      4576
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$


$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$   *CONSTRAINED_EXTRA_NODES_SET
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Rigidly attach all nodes in set 4 to part 17. (Part 17 MUST be a rigid body.)
$
$  In this example, four nodes from a deformable body are attached
$  to rigid body 17 as a means of joining the two parts.
$
*CONSTRAINED_EXTRA_NODES_SET
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$      pid      nsid
        17         4
$
$
*SET_NODE_LIST
$      sid
         4
$     nid1      nid2      nid3      nid4      nid5      nid6      nid7      nid8
       665       778       896       827
$
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

CONSTRAINED_GENERALIZED_WELD_OPTION*_{*OPTION*}

Available options include:

SPOT

FILLET

BUTT

CROSS_FILLET

COMBINED

To define an ID for the weld use the option:

ID

Purpose: Define spot, fillet, butt, and other types of welds. Coincident nodes are permitted if the local coordinate ID is defined. For the spot weld a local coordinate ID is not required if the nodes are offset. Failures can include both the plastic and brittle failures. These can be used either independently or together. Failure occurs when either criteria is met. The welds may undergo large rotations since the equations of rigid body mechanics are used to update their motion.

ID Card - Required if the option _ID is active on the keyword card.

Card 1	1	2	3	4	5	6	7	8
Variable	WID							
Type	I							
Default	0							

This card is required for all weld options.

Card 1	1	2	3	4	5	6	7	8
Variable	NSID	CID	FILTER	WINDOW	NPR	NPRT		
Type	I	I	I	E	I	I		
Default	none	none						

VARIABLE	DESCRIPTION
WID	Optional weld ID.
NSID	Nodal set ID, see *SET_NODE_*OPTION*.
CID	Coordinate system ID for output of spot weld data to SWFORC in local system, see *DEFINE_COORDINATE_*OPTION*. CID is not required for spot welds if the nodes are not coincident.
FILTER	Number of force vectors saved for filtering. This option can eliminate spurious failures due to numerical force spikes; however, memory requirements are significant since 6 force components are stored with each vector. LE.1: no filtering EQ.n: simple average of force components divided by n or the maximum number of force vectors that are stored for the time window option below.
WINDOW	Time window for filtering. This option requires the specification of the maximum number of steps which can occur within the filtering time window. If the time step decreases too far, then the filtering time window will be ignored and the simple average is used. EQ.0: time window is not used
NPR	NFW, number of individual nodal pairs in the cross fillet or combined general weld.
NPRT	Print option in file RBDOUT. EQ.0: default from the control card, *CONTROL_OUTPUT, is used, see variable name IPRTF. EQ.1: data is printed EQ.2: data is not printed

Additional Card required for the CONSTRAINED_GENERALIZED_WELD_SPOT option:

Card 2	1	2	3	4	5	6	7	8
Variable	TFAIL	EPSF	SN	SS	N	M		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
TFAIL	Failure time for constraint set, t_f. (default=1.E+20)
EPSF	Effective plastic strain at failure, ε_{fail}^{p} defines ductile failure.
SN	S_n, normal force at failure, only for the brittle failure of spot welds.
SS	S_s, shear force at failure, only for the brittle failure of spot welds.
N	n, exponent for normal force, only for the brittle failure of spot welds.
M	m, exponent for shear force, only for the brittle failure of spot welds.

Remarks:

Spot weld failure due to plastic straining occurs when the effective nodal plastic strain exceeds the input value, ε_{fail}^{p}. This option can model the tearing out of a spot weld from the sheet metal since the plasticity is in the material that surrounds the spot weld, not the spot weld itself. A least squares algorithm is used to generate the nodal values of plastic strains at the nodes from the element integration point values. The plastic strain is integrated through the element and the average value is projected to the nodes via a least square fit. This option should only be used for the material models related to metallic plasticity and can result in slightly increased run times.

Brittle failure of the spot welds occurs when:

$$\left(\frac{\max\left(f_n,0\right)}{S_n}\right)^n + \left(\frac{|f_s|}{S_s}\right)^m \geq 1$$

where f_n and f_s are the normal and shear interface force. Component f_n contributes for tensile values only. When the failure time, t_f, is reached the nodal rigid body becomes inactive and the constrained nodes may move freely. In Figure 6.1 the ordering of the nodes is shown for the 2 node and 3 node spot welds. This order is with respect to the local coordinate system where the local z-axis determines the tensile direction. The nodes in the spot weld may coincide. The

failure of the 3 node spot weld may occur gradually with first one node failing and later the second node may fail. For n noded spot welds the failure is progressive starting with the outer nodes (1 and n) and then moving inward to nodes 2 and n-1. Progressive failure is necessary to preclude failures that would create new rigid bodies.

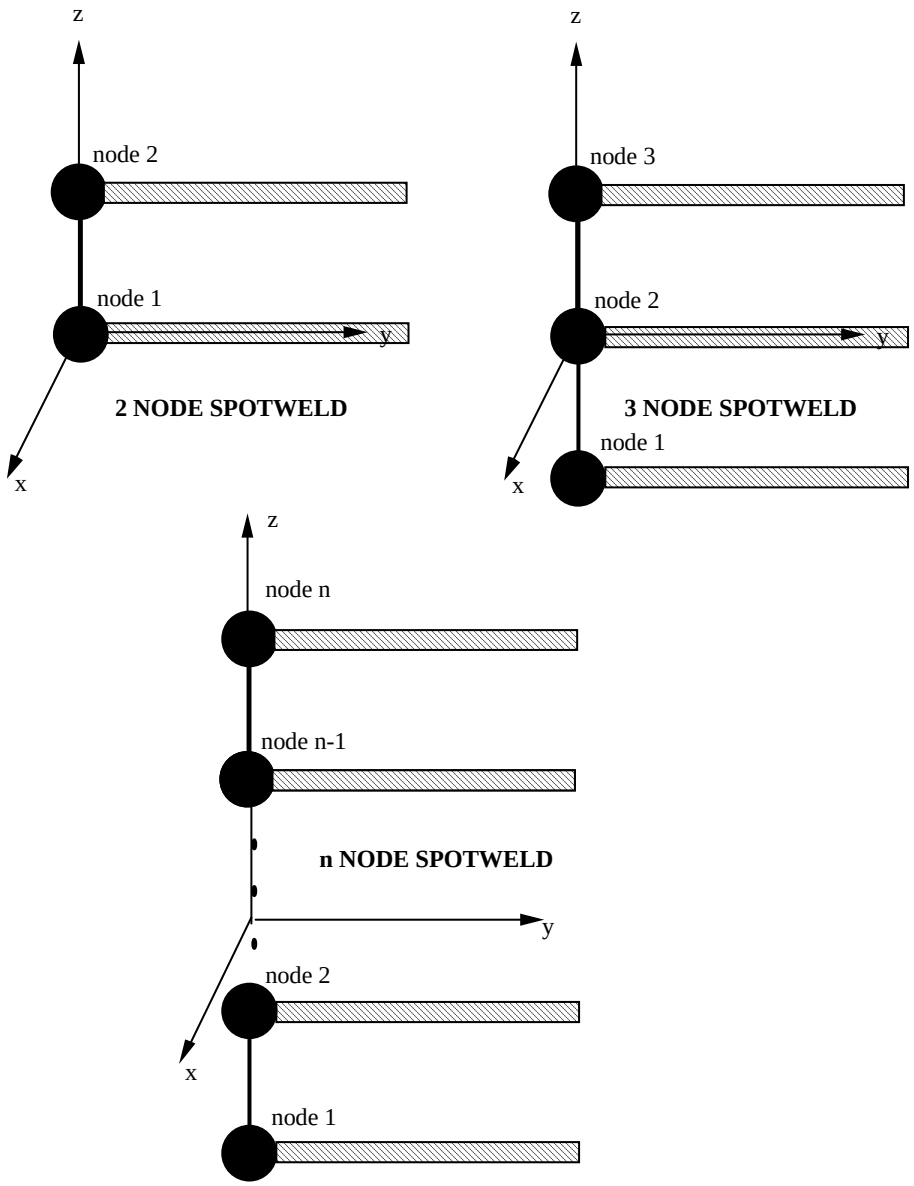


Figure 6.2. Nodal ordering and orientation of the local coordinate system is important for determining spotweld failure.

Additional Card required for the FILLET option:

Card 2	1	2	3	4	5	6	7	8
Variable	TFAIL	EPSF	SIGF	BETA	L	W	A	ALPHA
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
TFAIL	Failure time for constraint set, t_f. (default=1.E+20)
EPSF	Effective plastic strain at failure, ε_{fail}^{p} defines ductile failure.
SIGF	σ_f, stress at failure for brittle failure.
BETA	β, failure parameter for brittle failure.
L	L, length of fillet/butt weld (see Figure 6.2 and 6.3).
W	w, width of flange (see Figure 6.2).
A	a, width of fillet weld (see Figure 6.2).
ALPHA	α, weld angle (see Figure 6.2) in degrees.

Remarks:

Ductile fillet weld failure, due to plastic straining, is treated identically to spot weld failure. Brittle failure of the fillet welds occurs when:

$$\beta\sqrt{\sigma_n^2 + 3\left(\tau_n^2 + \tau_t^2\right)} \geq \sigma_f$$

where
- σ_n = normal stress
- τ_n = shear stress in direction of weld (local y)
- τ_t = shear stress normal to weld (local x)
- σ_f = failure stress
- β = failure parameter

Component σ_n is nonzero for tensile values only. When the failure time, t_f, is reached the nodal rigid body becomes inactive and the constrained nodes may move freely. In Figure 6.2 the ordering of the nodes is shown for the 2 node and 3 node fillet welds. This order is with respect

to the local coordinate system where the local z axis determines the tensile direction. The nodes in the fillet weld may coincide. The failure of the 3 node fillet weld may occur gradually with first one node failing and later the second node may fail.

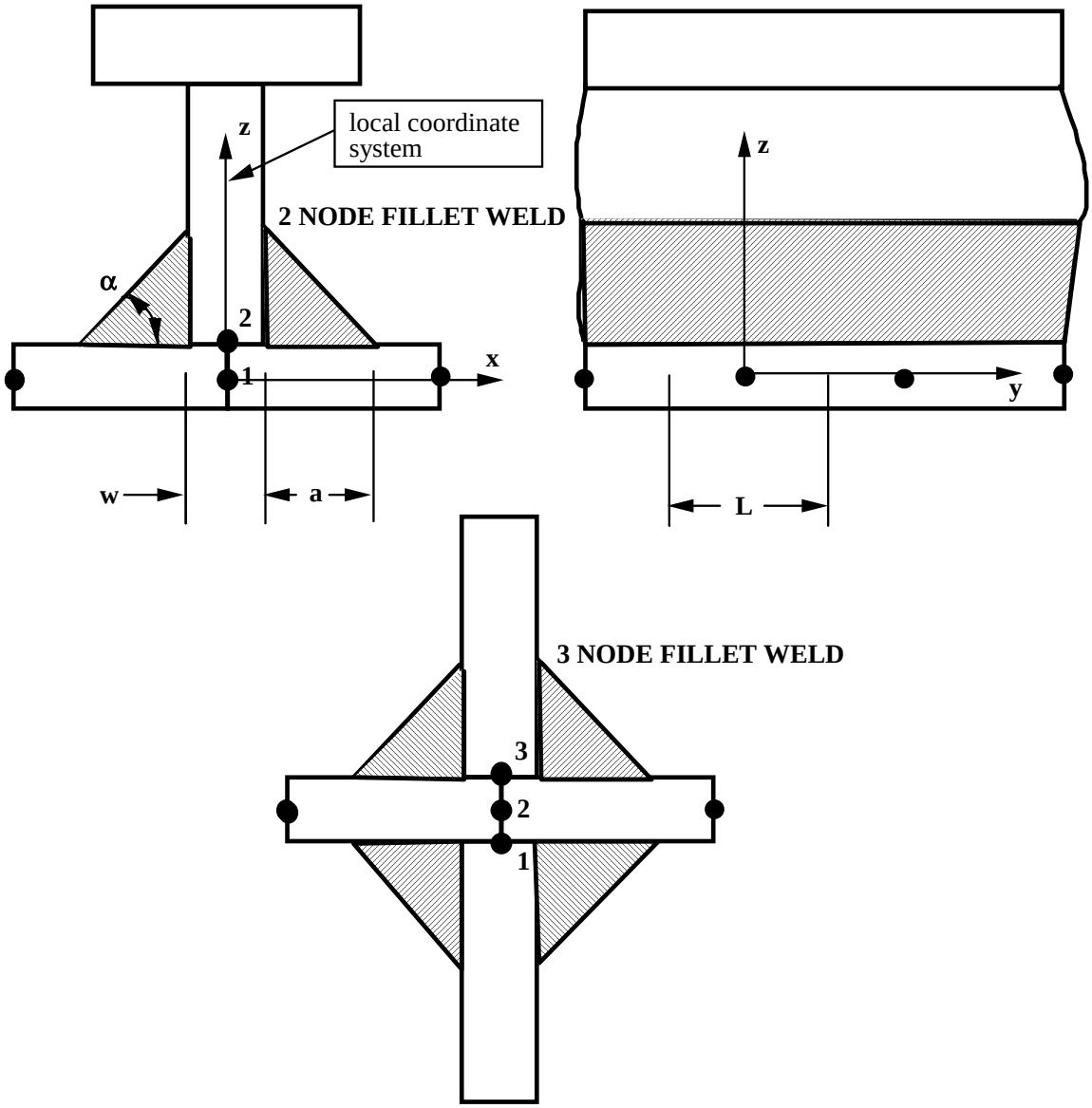


Figure 6.3. Nodal ordering and orientation of the local coordinate system is shown for fillet weld failure. The angle is defined in degrees.

Additional Card required for the BUTT option:

Card 2	1	2	3	4	5	6	7	8
Variable	TFAIL	EPSF	SIGY	BETA	L	D	LT	
Type	F	F	F	F	F	F	F	

VARIABLE	DESCRIPTION
TFAIL	Failure time for constraint set, t_f. (default=1.E+20)
EPSF	Effective plastic strain at failure, ε_{fail}^{p} defines ductile failure.
SIGY	σ_f, stress at failure for brittle failure.
BETA	β, failure parameter for brittle failure.
L	L, length of fillet/butt weld (see Figure 6.2 and 6.3).
D	d, thickness of butt weld (see Figure 6.3).
LT	L_t, transverse length of butt weld (see Figure 6.3).

Remarks:

Ductile butt weld failure, due to plastic straining, is treated identically to spot weld failure. Brittle failure of the butt welds occurs when:

$$\beta\sqrt{\sigma_n^2 + 3\left(\tau_n^2 + \tau_t^2\right)} \geq \sigma_f$$

where

σ_n = normal stress
τ_n = shear stress in direction of weld (local y)
τ_t = shear stress normal to weld (local z)
σ_f = failure stress
β = failure parameter

Component σ_n is nonzero for tensile values only. When the failure time, t_f, is reached the nodal rigid body becomes inactive and the constrained nodes may move freely. The nodes in the butt weld may coincide.

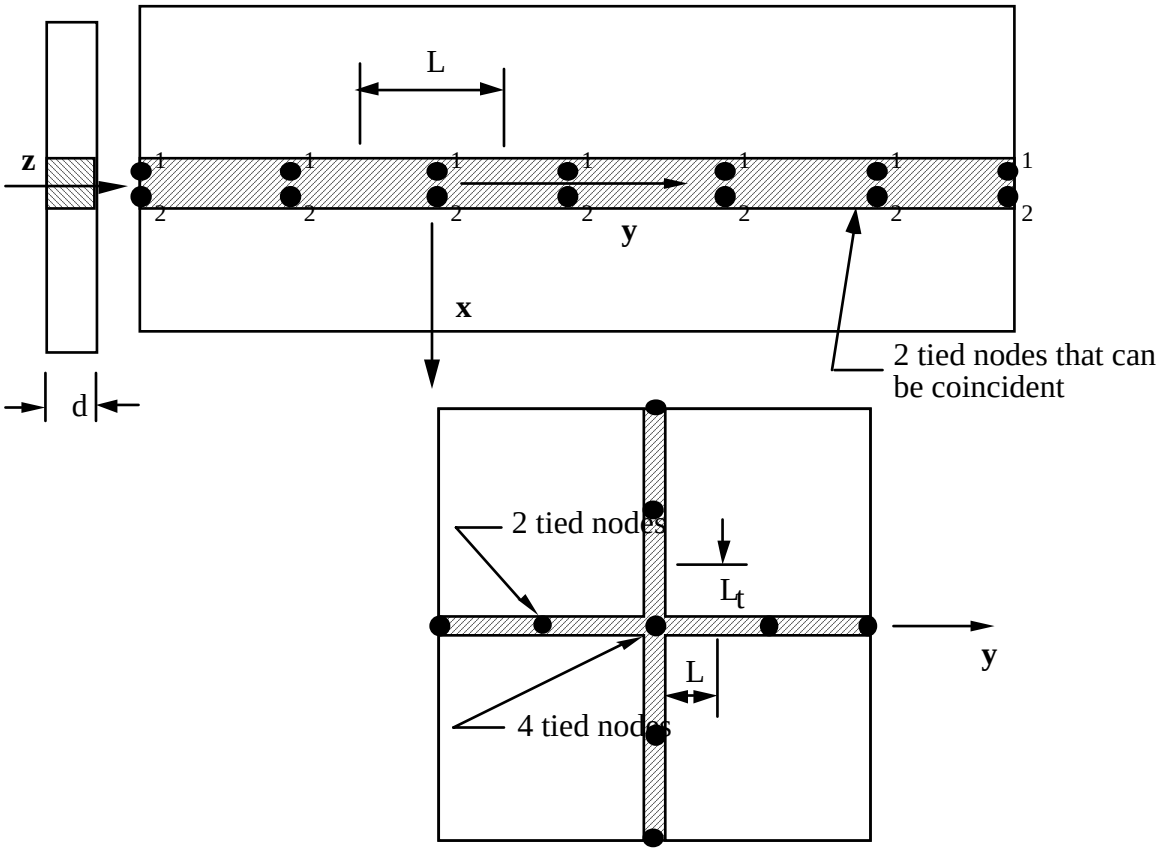


Figure 6.4. Orientation of the local coordinate system and nodal ordering is shown for butt weld failure.

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$  *CONSTRAINED_GENERALIZED_WELD_BUTT
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Weld two plates that butt up against each other at three nodal pair
$  locations.  The nodal pairs are 32-33, 34-35 and 36-37.
$
$  This requires 3 separate *CONSTRAINED_GENERALIZED_WELD_BUTT definitions,
$  one for each nodal pair.  Each weld is to have a length (L) = 10,
$  thickness (D) = 2, and a transverse length (Lt) = 1.
$
$  Failure is defined two ways:
$    Ductile failure if effective plastic strain exceeds 0.3
$    Brittle failure if the stress failure criteria exceeds 0.25
$       - scale the brittle failure criteria by beta = 0.9.
$    Note: beta > 1 weakens weld          beta < 1 strengthens weld
$
*CONSTRAINED_GENERALIZED_WELD_BUTT
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$     nsid       cid
         21
$    tfail      epsf      sigy      beta         L         D        Lt
          0.3     0.250       0.9      10.0       2.0       1.0
$
$
*CONSTRAINED_GENERALIZED_WELD_BUTT
$     nsid       cid
         23
$    tfail      epsf      sigy      beta         L         D        Lt
          0.3     0.250       0.9      10.0       2.0       1.0
$
$
*CONSTRAINED_GENERALIZED_WELD_BUTT
$     nsid       cid
         25
$    tfail      epsf      sigy      beta         L         D        Lt
          0.3     0.250       0.9      10.0       2.0       1.0
$
$
*SET_NODE_LIST
$      sid
         21
$     nid1      nid2
         32        33
*SET_NODE_LIST
         23
         34        35
*SET_NODE_LIST
         25
         36        37
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$

$
```

Additional Cards (1+NPR) required for the CROSS_FILLET option:

Card 2	1	2	3	4	5	6	7	8
Variable	TFAIL	EPSF	SIGY	BETA	L	W	A	ALPHA
Type	F	F	F	F	F	F	F	F

Cards 3,4, ...,2+NPR								
Variable	NODEA	NODEB	NCID					
Type	I	I	I					

VARIABLE	DESCRIPTION
TFAIL	Failure time for constraint set, t_f. (default=1.E+20)
EPSF	Effective plastic strain at failure, ε_{fail}^{p} defines ductile failure.
SIGY	σ_f, stress at failure for brittle failure.
BETA	β, failure parameter for brittle failure.
L	L, length of fillet/butt weld (see Figure 6.2 and 6.3).
W	w, width of flange (see Figure 6.2).
A	a, width of fillet weld (see Figure 6.2).
ALPHA	α, weld angle (see Figure 6.2) in degrees.
NODEA	Node ID, A, in weld pair (CROSS or COMBINED option only). See Figure 6.4.
NODEB	Node ID, B, in weld pair (CROSS or COMBINED option only).
NCID	Local coordinate system ID (CROSS or COMBINED option only).

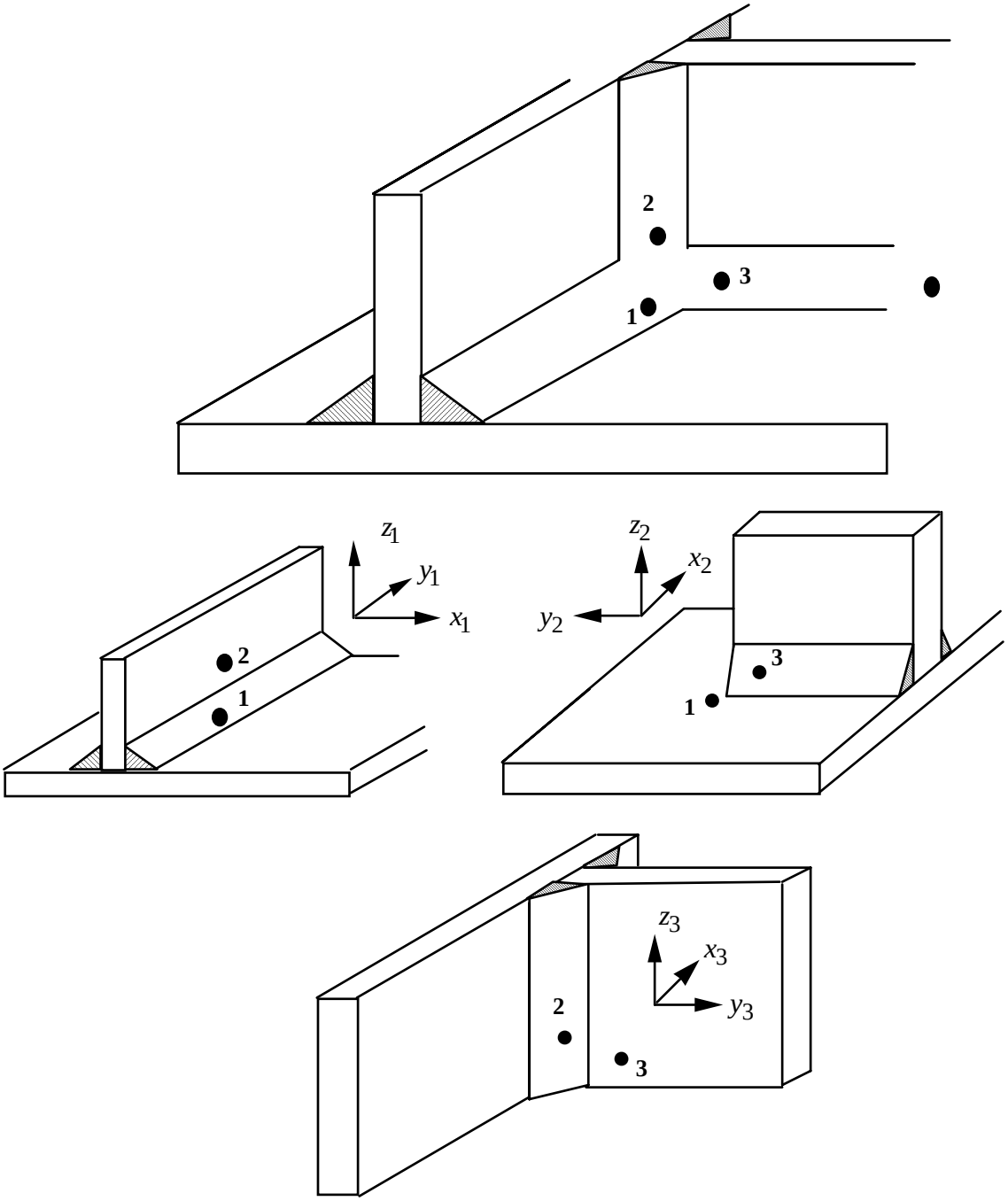


Figure 6.5. A simple cross fillet weld illustrates the required input. Here NFW=3 with nodal pairs (A=2, B=1), (A=3, B=1), and (A=3, B=2). The local coordinate axes are shown. These axes are fixed in the rigid body and are referenced to the local rigid body coordinate system which tracks the rigid body rotation.

Additional NPR Card Sets required for the COMBINED option. Repeat cards 2 and 3 below NPR times:

Card 2

	1	2	3	4	5	6	7	8
Variable	TFAIL	EPSF	SIGY	BETA	L	W	A	ALPHA
Type	F	F	F	F	F	F	F	F

Card 3

	1	2	3	4	5	6	7	8
Variable	NODEA	NODEB	NCID	WTYP				
Type	I	I	I	I				

VARIABLE	DESCRIPTION
TFAIL	Failure time for constraint set, t_f. (default=1.E+20)
EPSF	Effective plastic strain at failure, ε_{fail}^{p} defines ductile failure.
SIGY	σ_f, stress at failure for brittle failure.
BETA	β, failure parameter for brittle failure.
L	L, length of fillet/butt weld (see Figure 6.2 and 6.3).
W	w, width of flange (see Figure 6.2).
A	a, width of fillet weld (see Figure 6.2).
ALPHA	α, weld angle (see Figure 6.2) in degrees.
NODEA	Node ID, A, in weld pair (CROSS or COMBINED option only).
NODEB	Node ID, B, in weld pair (CROSS or COMBINED option only).
NCID	Local coordinate system ID (CROSS or COMBINED option only).
WTYPE	Weld pair type (GENERAL option only). See Figure 6.5. EQ.0: fillet weld EQ.1: butt weld

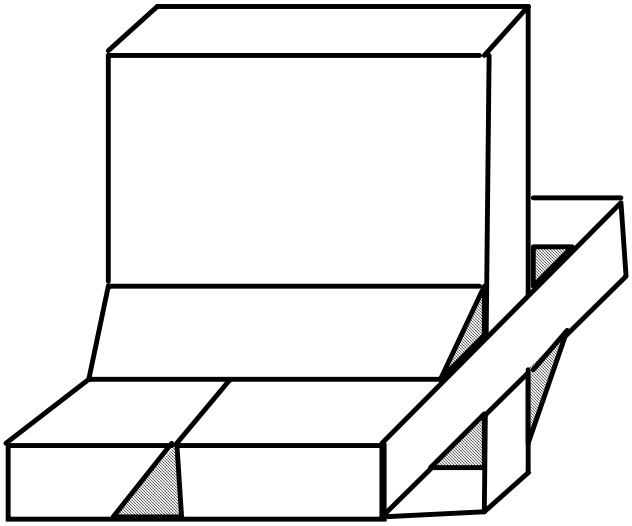


Figure 6.5. A combined weld is a mixture of fillet and butt welds.

*CONSTRAINED_GLOBAL

Purpose: Define a global boundary constraint plane.

Card	1	2	3	4	5	6	7	8
Variable	TC	RC	DIR	X	Y	Z		
Type	I	I	I	F	F	F		
Default	0	0	0	0	0	0		

VARIABLE **DESCRIPTION**

TC Translational Constraint:
- EQ.1: constrained x translation,
- EQ.2: constrained y translation,
- EQ.3: constrained z translation,
- EQ.4: constrained x and y translations,
- EQ.5: constrained y and z translations,
- EQ.6: constrained x and z translations,
- EQ.7: constrained x, y, and z translations,

RC Rotational Constraint:
- EQ.1: constrained x-rotation,
- EQ.2: constrained y-rotation,
- EQ.3: constrained z-rotation,
- EQ.4: constrained x and y rotations,
- EQ.5: constrained y and z rotations,
- EQ.6: constrained z and x rotations,
- EQ.7: constrained x, y, and z rotations.

DIR Direction of normal
- EQ.1: global x,
- EQ.2: global y,
- EQ.3: global z.

X x-offset coordinate

Y y-offset coordinate

Z z-offset coordinate

Remarks:

Nodes within a mesh-size-dependent tolerance are constrained on a global plane. This option is recommended for use with r-method adaptive remeshing where nodal constraints are lost during the remeshing phase.

***CONSTRAINED_INTERPOLATION_**{*OPTION*}

Available options include:

<BLANK>

LOCAL

Purpose: Define an interpolation constraint. With this constraint type, the motion of a single dependent node is interpolated from the motion of a set of independent nodes. This option is useful for the load redistribution of a load, which can be either a translational force or moment, applied to the dependent node to the surrounding independent nodes, and it can also be used to model shell-brick and beam-brick interfaces. The mass and rotary inertia of the dependent nodal point is also redistributed. This constraint is applied in the global coordinate system unless the option LOCAL is active. One *CONSTRAINED_INTERPOLATION card is required for each constraint definition. The input list of independent nodes is terminated when the next "*" card is found. In explicit calculations the independent nodes cannot be dependent nodes in other constraints such as nodal rigid bodies. This latter restriction does not apply to implicit calculations.

Card	1	2	3	4	5	6	7	8
Variable	ICID	DNID	DDOF	CIDD				
Type	I	I	I	I				
Default	0	0	123456	optional				

Cards 2, 3, 4, etc. Define one card per independent node. If the option LOCAL is active, define two cards per independent node. Input is terminated when a "*" card is found.

	1	2	3	4	5	6	7	8
Variable	INID	IDOF	TWGHTX	TWGHTY	TWGHTZ	RWGHTX	RWGHTY	RWGHTZ
Type	I	I	F	F	F	F	F	F
Default	0	123456	1.0	TWGHTX	TWGHTX	TWGHTX	TWGHTX	TWGHTX

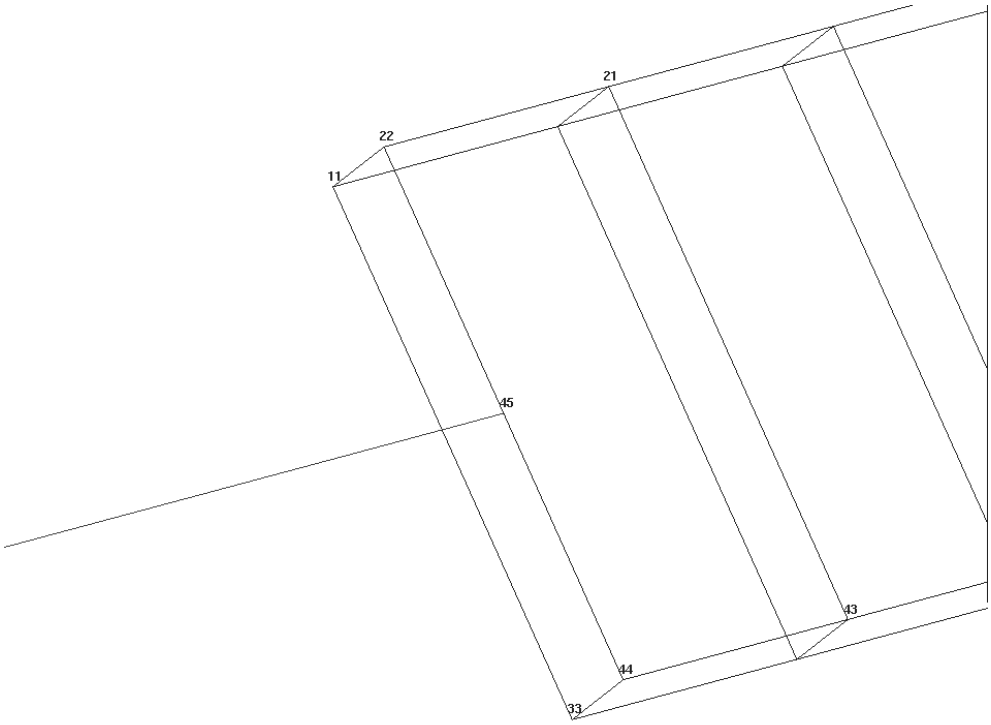
Define the second card if and only if the option LOCAL is active

Card	1	2	3	4	5	6	7	8
Variable	CIDI							
Type	I							
Default	0							

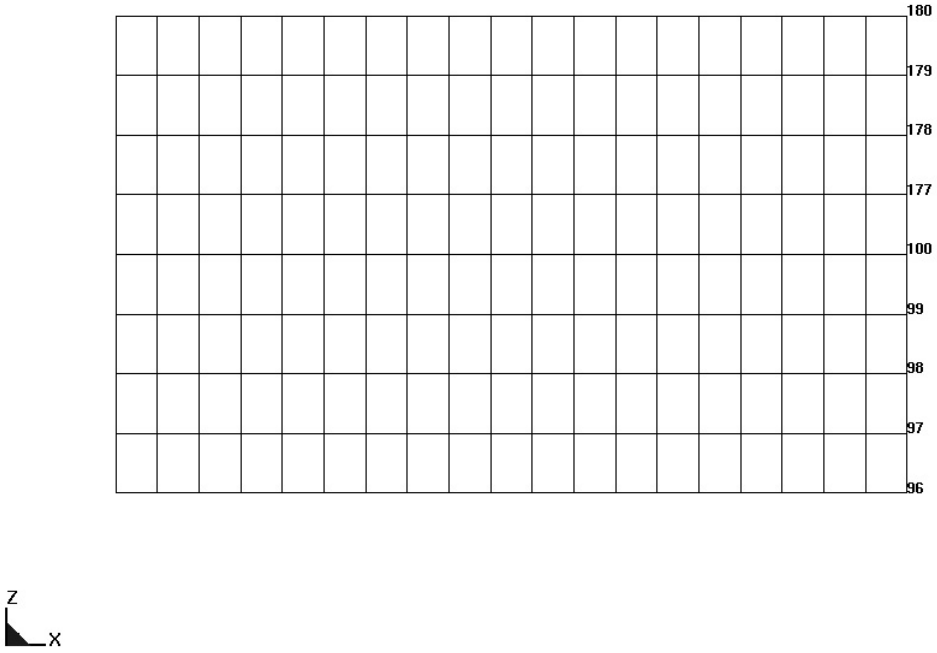
VARIABLE	DESCRIPTION
ICID	Interpolation constraint ID.
DNID	Dependent node ID. This node should not be a member of a rigid body, or elsewhere constrained in the input.
DDOF	Dependent degrees-of-freedom. The list of dependent degrees-of-freedom consists of a number with up to six digits, with each digit representing a degree of freedom. For example, the value 1356 indicates that degrees of freedom 1, 3, 5, and 6 are controlled by the constraint. The default is 123456. Digit: degree of freedom ID's: EQ.1: x EQ.2: y EQ.3: z EQ.4: rotation about x axis EQ.5: rotation about y axis EQ.6: rotation about z axis
CIDD	Local coordinate system ID if LOCAL option is active. If blank the global coordinate system is assumed.
INID	Independent node ID.
IDOF	Independent degrees-of-freedom using the same form as for the dependent degrees-of-freedom, DDOF, above.
TWGHTX	Weighting factor for node INID with active degrees-of-freedom IDOF. This weight scales the x-translational component. It is normally sufficient to define only TWGHTX even if its degree-of-freedom is inactive since the other factors are set equal to this input value as the default. There is no requirement on the values that are chosen as the weighting factors, i.e., that they sum to unity. The default value for the weighting factor is unity.

VARIABLE	DESCRIPTION
TWGHTY	Weighting factor for node INID with active degrees-of-freedom IDOF. This weight scales the y-translational component.
TWGHTZ	Weighting factor for node INID with active degrees-of-freedom IDOF. This weight scales the z-translational component.
RWGHTX	Weighting factor for node INID with active degrees-of-freedom IDOF. This weight scales the x-rotational component.
RWGHTY	Weighting factor for node INID with active degrees-of-freedom IDOF. This weight scales the y-rotational component.
RWGHTZ	Weighting factor for node INID with active degrees-of-freedom IDOF. This weight scales the z-rotational component.
CIDI	Local coordinate system ID if LOCAL option is active. If blank the global coordinate system is assumed.

```
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$  *CONSTRAINED_INTERPOLATION  (Beam to solid coupling)
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Tie a beam element to a solid element.
$
$  The node of the beam to be tied does not share a common node with the solids.
$  If the beam node is shared, for example, then set ddof=456.
$
*CONSTRAINED_INTERPOLATION
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$     icid      dnid      ddof
         1        45    123456
$     inid      idof    twghtx    twghty    twghtz    rwghtx    rwghty    rwghtz
        22       123
        44       123
        43       123
$
*.........
$
```




```
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$  *CONSTRAINED_INTERPOLATION  (Load redistribution)
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Moment about normal axis of node 100 is converted to an equivalent load by
$  applying x-force resultants to the nodes lying along the right boundary
$
*DEFINE_CURVE
1,0,0.,0.,0.,0.,0
0.,0.
.1,10000.
*LOAD_NODE_POINT
100,6,1,1.0
$
*CONSTRAINED_INTERPOLATION
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$     icid      dnid      ddof
         1       100         5
$     inid      idof    twghtx    twghty    twghtz    rwghtx    rwghty    rwghtz
        96         1
        97         1
        98         1
        99         1
       177         1
       178         1
       179         1
       180         1
$
*.........
$
```


180
179
178
177
100
99
98
97
96
Z
X

CONSTRAINED_JOINT_OPTION***_{***OPTION***}_{***OPTION***}_{***OPTION***}**

Available forms include (one is mandatory):

CONSTRAINED_JOINT_SPHERICAL

CONSTRAINED_JOINT_REVOLUTE

CONSTRAINED_JOINT_CYLINDRICAL

CONSTRAINED_JOINT_PLANAR

CONSTRAINED_JOINT_UNIVERSAL

CONSTRAINED_JOINT_TRANSLATIONAL

CONSTRAINED_JOINT_LOCKING

CONSTRAINED_JOINT_TRANSLATIONAL_MOTOR

CONSTRAINED_JOINT_ROTATIONAL_MOTOR

CONSTRAINED_JOINT_GEARS

CONSTRAINED_JOINT_RACK_AND_PINION

CONSTRAINED_JOINT_CONSTANT_VELOCITY

CONSTRAINED_JOINT_PULLEY

CONSTRAINED_JOINT_SCREW

If the force output data is to be transformed into a local coordinate use the option:

LOCAL

to define a joint ID and heading the following option is available:

ID

and to define failure for penalty-based joints (LMF=0 in *CONTROL_RIGID) use:

FAILURE

The ordering of the bracketed options is arbitrary.

Purpose: Define a joint between two rigid bodies, see Figure 6.6.

Card Format:

Card 1 is required for all joint types.
Card 2 is required for joint types: MOTOR, GEARS, RACK_AND_PINION, PULLEY, and SCREW
Optional Card is required only if LOCAL is specified in the keyword.

In the first seven joint types above excepting the Universal joint, the nodal points within the nodal pairs (1,2), (3,4), and (5,6) (see Figure 6.6) should coincide in the initial configuration, and the nodal pairs should be as far apart as possible to obtain the best behavior. For the Universal Joint the nodes within the nodal pair (3,4) do not coincide, but the lines drawn between nodes (1,3) and (2,4) must be perpendicular.

For the Gear joint the nodes within the nodal pair (1,2) must not coincide.

The geometry of joints is defined in Figure 6.6. When the penalty method is used (see *CONTROL_RIGID), at each time step, the relative penalty stiffness is multiplied by a function dependent on the step size to give the maximum stiffness that will not destroy the stability of the solution. Instabilities can result in the explicit time integration scheme if the penalty stiffness is too large. If instabilities occur, the recommended way to eliminate these problems is to decrease the time step or reduce the scale factor on the penalties.

For cylindrical joints, by setting node 3 to zero, it is possible to use a cylindrical joint to join a node that is not on a rigid body (node 1) to a rigid body (nodes 2 and 4).

The following card is read if and only if the ID option is specified.

Optional 1 2-8

Variable	JID	HEADING
Type	I	A70

The heading is picked up by some of the peripheral LS-DYNA codes to aid in post-processing.

VARIABLE	DESCRIPTION
JID	Joint ID. This must be a unique number.
HEADING	Joint descriptor. It is suggested that unique descriptions be used.

Card 1 - Required

Card 1 1 2 3 4 5 6 7 8

Variable	N1	N2	N3	N4	N5	N6	RPS	DAMP
Type	I	I	I	I	I	I	F	F
Default	0	0	0	0	0	0	1.0	1.0

VARIABLE	DESCRIPTION
N1	Node 1, in rigid body A. Define for all joint types.
N2	Node 2, in rigid body B. Define for all joint types.

VARIABLE	DESCRIPTION
N3	Node 3, in rigid body A. Define for all joint types except SPHERICAL.
N4	Node 4, in rigid body B. Define for all joint types except SPHERICAL.
N5	Node 5, in rigid body A. Define for joint types TRANSLATIONAL, LOCKING, ROTATIONAL_MOTOR, CONSTANT_VELOCITY, GEARS, RACK_AND_PINION, PULLEY, and SCREW
N6	Node 6, in rigid body B. Define for joint types TRANSLATIONAL, LOCKING, ROTATIONAL_MOTOR, CONSTANT_VELOCITY, GEARS, RACK_AND_PINION, PULLEY, and SCREW
RPS	Relative penalty stiffness (default = 1.0).
DAMP	Damping scale factor on default damping value. (Revolute and Spherical Joints): EQ.0.0: default is set to 1.0, LE.0.01 and GT.0.0: no damping is used.

Card 2. Required for joint types MOTOR, GEARS, RACK_AND_PINION, PULLEY, and SCREW only.

Card 1	1	2	3	4	5	6	7	8
Variable	PARM	LCID	TYPE	R1				
Type	F	I	I	F				
Default	None							

VARIABLE	DESCRIPTION
PARM	Parameter which a function of joint type. Leave blank for MOTORS. Gears: define R_2 / R_1 Rack and Pinion: define h Pulley: define R_2 / R_1 Screw: define $\dot{x} / \omega$
LCID	Define load curve ID for MOTOR joints.
TYPE	Define integer flag for MOTOR joints as follows: EQ.0: translational/rotational velocity EQ.1: translational/rotational acceleration EQ.2: translational/rotational displacement
R1	Radius, R_1, for the gear and pulley joint type. If left undefined, nodal points 5 and 6 are assumed to be on the outer radius.

Optional: Required only if LOCAL is specified after the keyword.

Card 1	1	2	3	4	5	6	7	8
Variable	RAID	LST						
Type	I	I						
Default	0	0						

VARIABLE	DESCRIPTION
RAID	Rigid body or accelerometer ID. The force resultants are output in the local system of the rigid body or accelerometer.
LST	Flag for local system type: EQ.0: rigid body EQ.1: accelerometer

Optional: Required only if FAILURE is specified after the keyword.

Card 1	1	2	3	4	5	6	7	8
Variable	CID	TFAIL	COUPL					
Type	I	F	F					
Default	0	0	0.					

Card 2	1	2	3	4	5	6	7	8
Variable	NXX	NYY	NZZ	MXX	MYY	MZZ		
Type	F	F	F	F	F	F		
Default	0	0	0	0	0	0		

VARIABLE	DESCRIPTION
CID	Coordinate ID for resultants in the failure criteria. If zero, the global coordinate system is used.
TFAIL	Time for joint failure. If zero, joint never fails.
COUPL	Coupling between the force and moment failure criteria. If COUPL is less than or equal to zero, the failure criteria is identical to the spotwelds. When COUPL is greater than zero, the force and moment results are considered independently. See the remark below.
NXX	Axial force resultant N_{xx_F} at failure. If zero, failure due to this component is not considered.
NYY	Force resultant N_{yy_F} at failure. If zero, failure due to this component is not considered.
NZZ	Force resultant N_{zz_F} at failure. If zero, failure due to this component is not considered.
MXX	Torsional moment resultant M_{xx_F} at failure. If zero, failure due to this component is not considered.
MYY	Moment resultant M_{xx_F} at failure. If zero, failure due to this component is not considered.
MZZ	Moment resultant M_{zz_F} at failure. If zero, failure due to this component is not considered.

Remarks:

The moments for the revolute, cylindrical, planar, translational, and locking joints are calculated at the midpoint of nodes N1 and N3. The moments for the spherical, universal, constant velocity, gear, pulley, and rack and pinion joints are calculated at node N1.

When COUPL is less than or equal to zero, the failure criteria is

$$\left(\frac{max\left(N_{xx},0\right)}{N_{xx_F}}\right)^2+\left(\frac{N_{yy}}{N_{yy_F}}\right)^2+\left(\frac{N_{zz}}{N_{zz_F}}\right)^2+\left(\frac{M_{xx}}{M_{xx_F}}\right)^2+\left(\frac{M_{yy}}{M_{yy_F}}\right)^2+\left(\frac{M_{zz}}{M_{zz_F}}\right)^2-1=0$$

otherwise, it is

$$\left(\frac{max\left(N_{xx},0\right)}{N_{xx_F}}\right)^2+\left(\frac{N_{yy}}{N_{yy_F}}\right)^2+\left(\frac{N_{zz}}{N_{zz_F}}\right)^2-1=0 \text{ and } \left(\frac{M_{xx}}{M_{xx_F}}\right)^2+\left(\frac{M_{yy}}{M_{yy_F}}\right)^2+\left(\frac{M_{zz}}{M_{zz_F}}\right)^2-1=0\,.$$

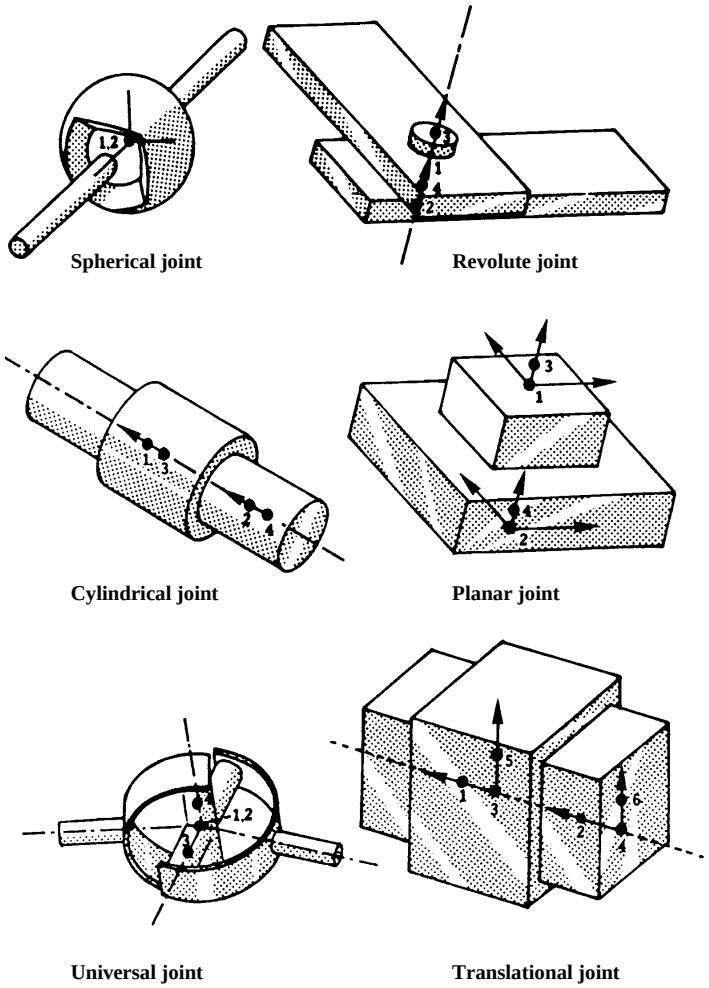


Figure 6.6 Joint definitions 1-6.

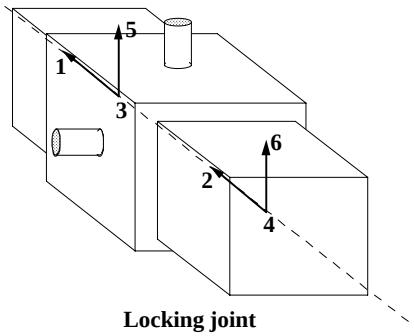


Figure 6.7. Locking joint.

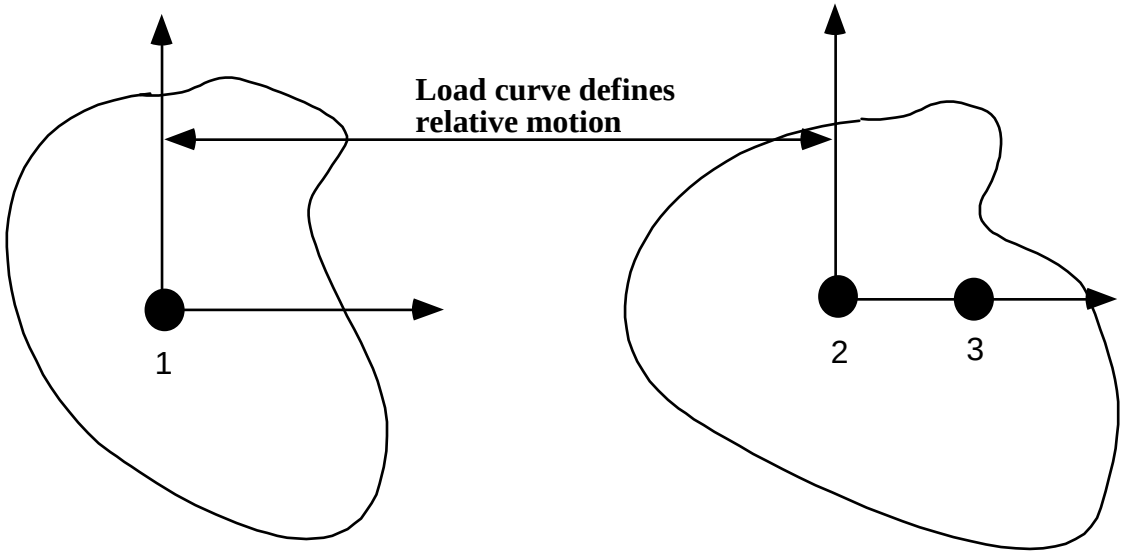


Figure 6.8. Translational motor joint. This joint can be used in combination with the translational or the cylindrical joint.

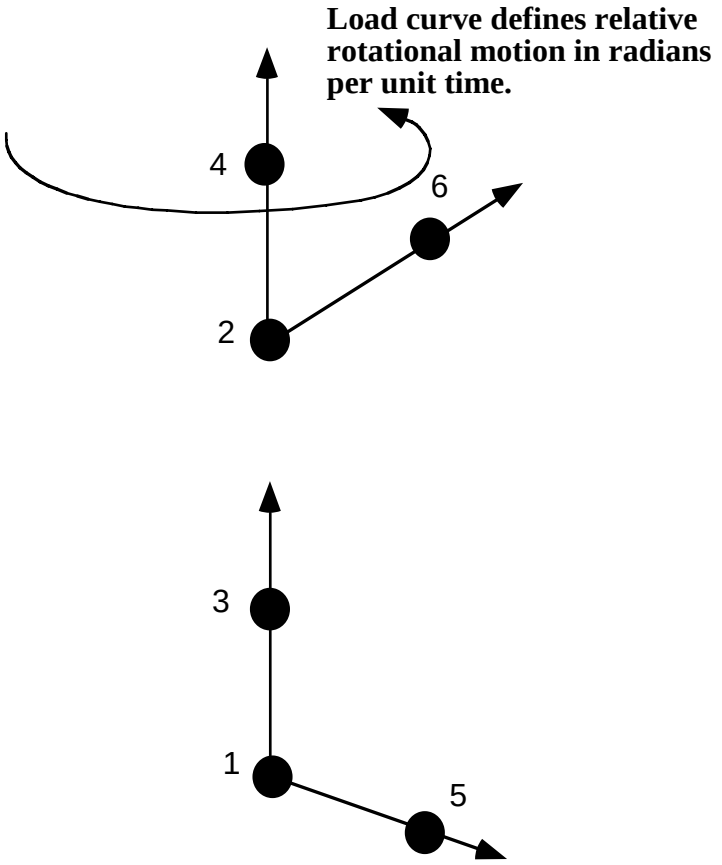


Figure 6.9. Rotational motor joint. This joint can be used in combination with other joints such as the revolute or cylindrical joints.

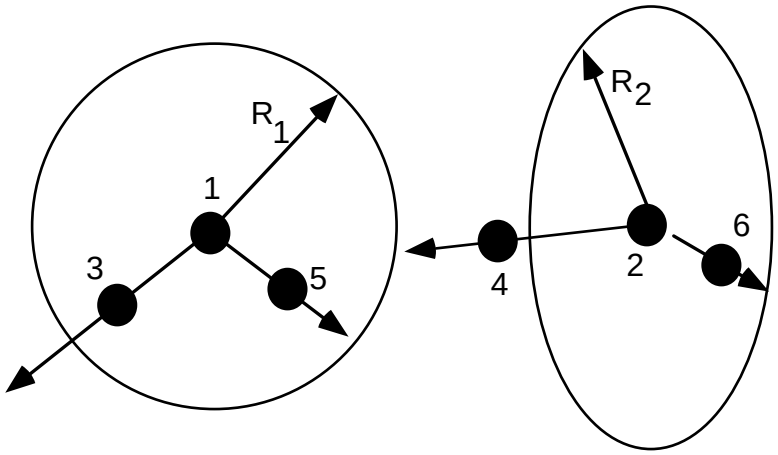


Figure 6.10. Gear joint. Nodal pairs (1,3) and (2,4) define axes that are orthogonal to the gears. Nodal pairs (1,5) and (2,6) define vectors in the plane of the gears. The ratio $\frac{R_2}{R_1}$ is specified.

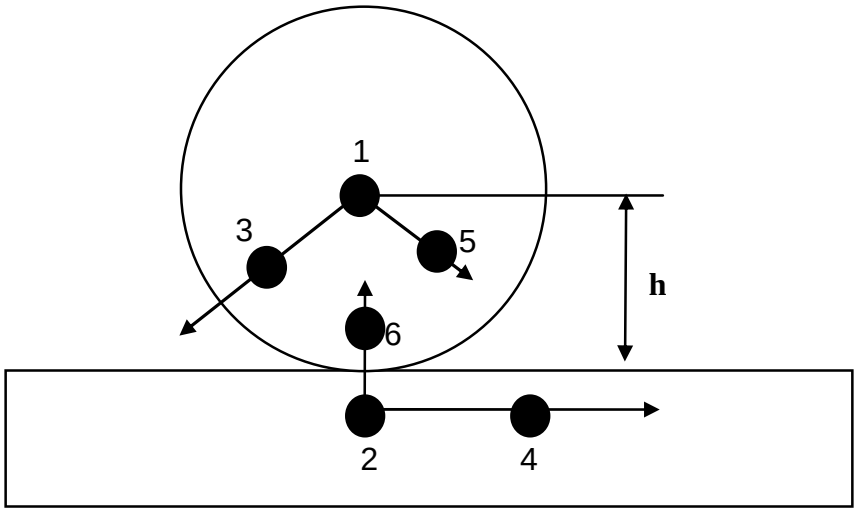


Figure 6.11. Rack and pinion joint. Nodal pair (1,3) defines a vector that is orthogonal to the plane of the gear. Nodal pair (1,5) is a vector in the plane of the gear. Nodal pair (2,4) defines the direction of travel for the second body. The value h is specified. The velocity of the rack is ω_{pinion} x h.

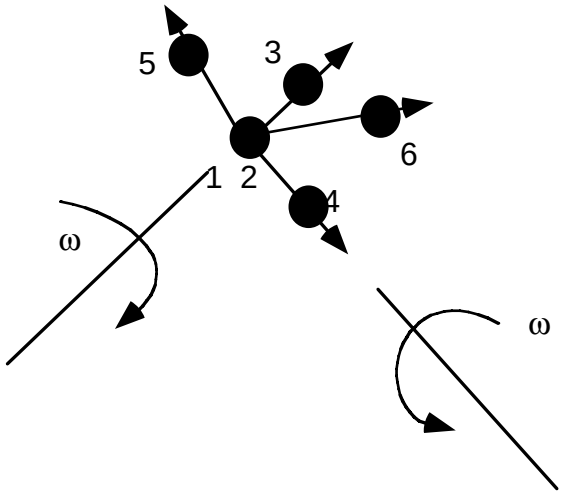


Figure 6.12. Constant velocity joint. Nodal pairs (1,3) and (2,4) define an axes for the constant angular velocity, and nodal pairs (1,5) are orthogonal vectors. Here nodal points 1 and 2 must be coincident.

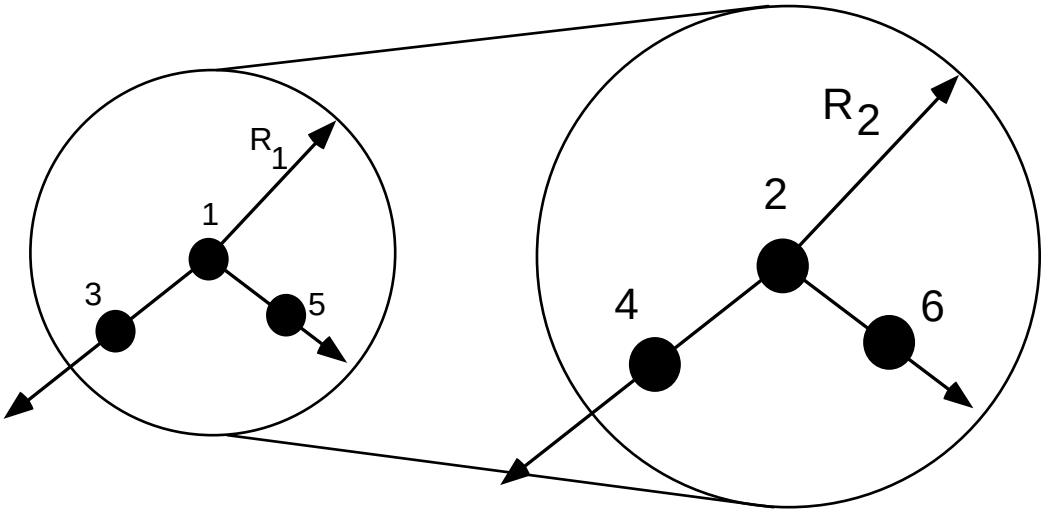


Figure 6.13. Pulley joint. Nodal pairs (1,3) and (2,4) define axes that are orthogonal to the pulleys. Nodal pairs (1,5) and (2,6) define vectors in the plane of the pulleys. The ratio $\frac{R_2}{R_1}$ is specified.

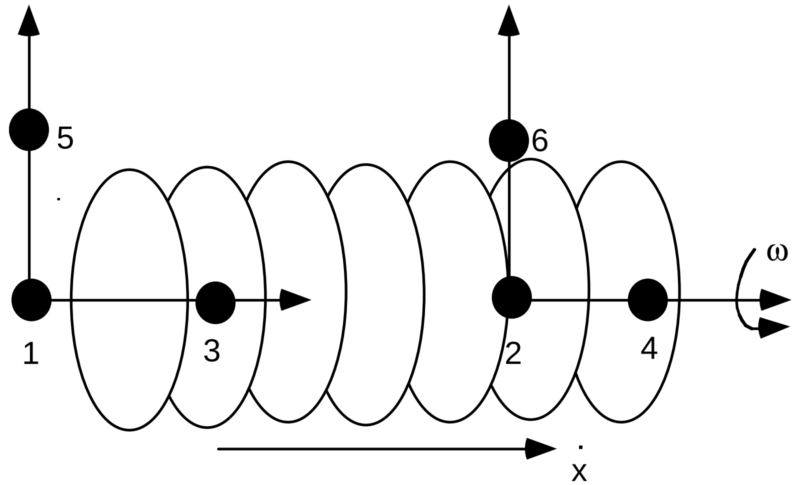


Figure 6.14. Screw joint. The second body translates in response to the spin of the first body. Nodal pairs (1,3) and (2,4) lie along the same axis and nodal pairs (1,5) and (2,6) are orthogonal vectors. The helix ratio, $\frac{\dot{x}}{\omega}$, is specified.

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$  *CONSTRAINED_JOINT_PLANAR
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Define a planar joint between two rigid bodies.
$    - Nodes 91 and 94 are on rigid body 1.
$    - Nodes 21 and 150 are on rigid body 2.
$    - Nodes 91 and 21 must be coincident.
$        * These nodes define the origin of the joint plane.
$    - Nodes 94 and 150 must be coincident.
$        * To accomplish this, massless node 150 is artificially created at
$           the same coordinates as node 94 and then added to rigid body 2.
$        * These nodes define the normal of the joint plane (e.g., the
$           vector from node 91 to 94 defines the planes' normal).
$
*CONSTRAINED_JOINT_PLANAR
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$       n1        n2        n3        n4        n5        n6       rps
        91        21        94       150                     0.000E+00
$
$
*NODE
$    nid               x               y               z      tc      rc
     150            0.00            3.00            0.00       0       0
$
*CONSTRAINED_EXTRA_NODES_SET
$      pid      nsid
         2         6
$
```

```
*SET_NODE_LIST
$      sid
         6
$     nid1
        150
$
$$$  request output for joint force data
$
*DATABASE_JNTFORC
$  dt/cycl      lcdt
    0.0001
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$  *CONSTRAINED_JOINT_REVOLUTE
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Create a revolute joint between two rigid bodies.  The rigid bodies must
$  share a common edge to define the joint along.  This edge, however, must
$  not have the nodes merged together.  Rigid bodies A and B will rotate
$  relative to each other along the axis defined by the common edge.
$
$  Nodes 1 and 2 are on rigid body A and coincide with nodes 9 and 10
$  on rigid body B, respectively.  (This defines the axis of rotation.)
$
$  The relative penalty stiffness on the revolute joint is to be 1.0,
$  the joint is well lubricated, thus no damping at the joint is supplied.
$
*CONSTRAINED_JOINT_REVOLUTE
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$       n1        n2        n3        n4        n5        n6       rps      damp
         1         9         2        10                            1.0
$
$  Note:  A joint stiffness is not mandatory for this joint to work.
$         However, to see how a joint stiffness can be defined for this
$         particular joint, see the corresponding example listed in:
$            *CONSTRAINED_JOINT_STIFFNESS_GENERALIZED
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

CONSTRAINED_JOINT_COOR_OPTION*_{*OPTION*}_{*OPTION*}_{*OPTION*}

Available forms include (one is mandatory):

CONSTRAINED_JOINT_ COOR_SPHERICAL

CONSTRAINED_JOINT_ COOR_REVOLUTE

CONSTRAINED_JOINT_ COOR_CYLINDRICAL

CONSTRAINED_JOINT_ COOR_PLANAR

CONSTRAINED_JOINT_ COOR_UNIVERSAL

CONSTRAINED_JOINT_ COOR_TRANSLATIONAL

CONSTRAINED_JOINT_ COOR_LOCKING

CONSTRAINED_JOINT_ COOR_TRANSLATIONAL_MOTOR

CONSTRAINED_JOINT_ COOR_ROTATIONAL_MOTOR

CONSTRAINED_JOINT_ COOR_GEARS

CONSTRAINED_JOINT_ COOR_RACK_AND_PINION

CONSTRAINED_JOINT_ COOR_CONSTANT_VELOCITY

CONSTRAINED_JOINT_ COOR_PULLEY

CONSTRAINED_JOINT_ COOR_SCREW

If the force output data is to be transformed into a local coordinate use the option:

LOCAL

to define a joint ID and heading the following option is available:

ID

and to define failure for penalty-based joints (LMF=0 in *CONTROL_RIGID) use:

FAILURE

The ordering of the bracketed options is arbitrary.

Purpose: Define a joint between two rigid bodies, see Figure 6.6. The connection coordinates are given instead of the nodal point IDs required in the previous section, *CONSTRAINED_ JOINT_.{*Joint_Type}*. Nodes are automatically generated for each coordinate and are constrained to the rigid body. Where coincident nodes are expected in the initial configuration, only one connection coordinate is needed since the connection coordinate for the second node, if given, is ignored. The created nodal ID's are chosen to exceed the maximum user ID. The coordinates of the joint nodes are specified on Cards 2-7. The input which follows Card 7 is identical to that in the previous section.

Card Format:

Cards 1-7 are required for all joint types.
Card 8 is required for joint types: MOTOR, GEARS, RACK_AND_PINION, PULLEY, and SCREW
Optional Card is required only if LOCAL is specified in the keyword.

In the first seven joint types above excepting the Universal joint, the coordinate points within the nodal pairs (1,2), (3,4), and (5,6) (see Figure 6.6) should coincide in the initial configuration, and the nodal pairs should be as far apart as possible to obtain the best behavior. For the Universal Joint the nodes within the coordinate pair (3,4) do not coincide, but the lines drawn between nodes (1,3) and (2,4) must be perpendicular.

For the Gear joint the nodes within the coordinate pair (1,2) must not coincide.

The geometry of joints is defined in Figure 6.6. When the penalty method is used (see *CONTROL_RIGID), at each time step, the relative penalty stiffness is multiplied by a function dependent on the step size to give the maximum stiffness that will not destroy the stability of the solution. Instabilities can result in the explicit time integration scheme if the penalty stiffness is too large. If instabilities occur, the recommended way to eliminate these problems is to decrease the time step or reduce the scale factor on the penalties.

For cylindrical joints, by setting node 3 to zero, it is possible to use a cylindrical joint to join a node that is not on a rigid body (node 1) to a rigid body (nodes 2 and 4).

The following card is read if and only if the ID option is specified.

Optional 1 2-8

Variable	JID	HEADING
Type	I	A70

The heading is picked up by some of the peripheral LS-DYNA codes to aid in post-processing.

VARIABLE	DESCRIPTION
JID	Joint ID. This must be a unique number.
HEADING	Joint descriptor. It is suggested that unique descriptions be used.

Card 1 – 7 Required

Card 1 1 2 3 4 5 6 7 8

Variable	RBID_A	RBID_B	RPS	DAMP	TMASS	RMASS		
Type	I	I	F	F	F	F		

Card 2

Variable	X1	Y1	Z1					
Type	F	F	F					

Card 3	1	2	3	4	5	6	7	8

Variable	X2	Y2	Z2					
Type	F	F	F					

Card 4

Variable	X3	Y3	Z3					
Type	F	F	F					

Card 5

Variable	X4	Y4	Z4					
Type	F	F	F					

Card 6

Variable	X5	Y5	Z5					
Type	F	F	F					

Card 7

Variable	X6	Y6	Z6					
Type	F	F	F					

VARIABLE	DESCRIPTION
RBID_A	Part ID of rigid body A.
RBID_B	Part ID of rigid body B.
RPS	Relative penalty stiffness (default = 1.0).
DAMP	Damping scale factor on default damping value. (Revolute and Spherical Joints): EQ.0.0: default is set to 1.0, LE.0.01 and GT.0.0: no damping is used.
TMASS	Lumped translational mass. The mass is equally split between the first points defined for rigid bodies A and B.
RMASS	Lumped rotational inertia. The inertia is equally split between the first points defined for rigid bodies A and B.
X1, Y1, Z1	Coordinate of point 1, in rigid body A. Define for all joint types.
X2, Y2, Z2	Coordinate of point 2, in rigid body B. If points 1 and 2 are coincident in the specified joint type, the coordinate for point 1 is used.
X3, Y3, Z31	Coordinate of point 3, in rigid body A. Define for all joint types.
X4, Y4, Z4	Coordinate of point 4, in rigid body B. If points 3 and 4 are coincident in the specified joint type, the coordinate for point 3 is used.
X5, Y5, Z5	Coordinate of point 5, in rigid body A. Define for all joint types.
X6, Y6, Z6	Coordinate of point 6, in rigid body B. If points 5 and 6 are coincident in the specified joint type, the coordinate for point 5 is used.

Card 8. Required for joint types MOTOR, GEARS, RACK_AND_PINION, PULLEY, and SCREW only.

Card 1	1	2	3	4	5	6	7	8
Variable	PARM	LCID	TYPE	R1				
Type	F	I	I	F				
Default	None							

VARIABLE	DESCRIPTION
PARM	Parameter which a function of joint type. Leave blank for MOTORS. Gears: define R_2 / R_1 Rack and Pinion: define h Pulley: define R_2 / R_1 Screw: define $\dot{x} / \omega$
LCID	Define load curve ID for MOTOR joints.
TYPE	Define integer flag for MOTOR joints as follows: EQ.0: translational/rotational velocity EQ.1: translational/rotational acceleration EQ.2: translational/rotational displacement
R1	Radius, R_1, for the gear and pulley joint type. If left undefined, nodal points 5 and 6 are assumed to be on the outer radius.

Optional: Required only if LOCAL is specified after the keyword.

Optional	1	2	3	4	5	6	7	8
Variable	RAID	LST						
Type	I	I						
Default	0	0						

VARIABLE	DESCRIPTION
RAID	Rigid body or accelerometer ID. The force resultants are output in the local system of the rigid body or accelerometer.
LST	Flag for local system type: EQ.0: rigid body EQ.1: accelerometer

Optional cards: Required only if FAILURE is specified after the keyword.

Optional 1

Variable	CID	TFAIL	COUPL					
Type	I	F	F					
Default	0	0	0.					

Optional 2

Variable	NXX	NYY	NZZ	MXX	MYY	MZZ		
Type	F	F	F	F	F	F		
Default	0	0	0	0	0	0		

VARIABLE	DESCRIPTION
CID	Coordinate ID for resultants in the failure criteria. If zero, the global coordinate system is used.
TFAIL	Time for joint failure. If zero, joint never fails.
COUPL	Coupling between the force and moment failure criteria. If COUPL is less than or equal to zero, the failure criteria is identical to the spotwelds. When COUPL is greater than zero, the force and moment results are considered independently. See the remark below.

VARIABLE	DESCRIPTION
NXX	Axial force resultant N_{xx_F} at failure. If zero, failure due to this component is not considered.
NYY	Force resultant N_{yy_F} at failure. If zero, failure due to this component is not considered.
NZZ	Force resultant N_{zz_F} at failure. If zero, failure due to this component is not considered.
MXX	Torsional moment resultant M_{xx_F} at failure. If zero, failure due to this component is not considered.
MYY	Moment resultant M_{xx_F} at failure. If zero, failure due to this component is not considered.
MZZ	Moment resultant M_{zz_F} at failure. If zero, failure due to this component is not considered.

CONSTRAINED_JOINT_STIFFNESS_OPTION*_**{***OPTION***}**

Available options include:

FLEXION-TORSION

GENERALIZED

TRANSLATIONAL

If desired a description of the joint stiffness can be provided with the option:

TITLE

which is written into the D3HSP and JNTFORC files.

Purpose: Define optional rotational and translational joint stiffness for joints defined by *CONSTRAINED_JOINT_*OPTION*. These definitions apply to all joints even though degrees of freedom that are considered in the joint stiffness capability may be constrained out in some joint types. The energy that is dissipated with the joint stiffness option is written for each joint in joint force file with the default name, JNTFORC. In the global energy balance this energy is included with the energy of the discrete elements, i.e., the springs and dampers.

Card Formats:

The optional TITLE card and card 1 are common to all joint stiffness types.
Cards 2 to 4 are unique for each stiffness type.

The following card is read if and only if the TITLE option is specified.

Optional

Variable	TITLE
Type	A80

Card 1 - Required for all joint stiffness types.

Card 1	1	2	3	4	5	6	7	8
Variable	JSID	PIDA	PIDB	CIDA	CIDB	JID		
Type	I	I	I	I	I	I		
Default	none	none	none	none	CIDA	none		

VARIABLE	DESCRIPTION
TITLE	Description of joint stiffness for output files JNTFORC and D3HSP.
JSID	Joint stiffness ID
PIDA	Part ID for rigid body A, see *PART.
PIDB	Part ID for rigid body B, see *PART.
CIDA	Coordinate ID for rigid body A, see *DEFINE_COORDINATE_*OPTION*. *For the translational stiffness the local coordinate system must be defined by nodal points, *DEFINE_COORDINATE_NODES, since the first nodal point in each coordinate system is used to track the motion.*
CIDB	Coordinate ID for rigid body B. If zero, the coordinate ID for rigid body A is used, see *DEFINE_COORDINATE_*OPTION*. *For the translational stiffness the local coordinate system must be defined by nodal points, *DEFINE_COORDINATE_NODES, since the first nodal point in each coordinate system is used to track the motion.*
JID	Joint ID for the joint reaction forces. If zero, tables can't be used in place of load curves for defining the frictional moments.

Card 2 of 4 - Required for FLEXION-TORSION stiffness.

Card 2	1	2	3	4	5	6	7	8
Variable	LCIDAL	LCIDG	LCIDBT	DLCIDAL	DLCIDG	DLCIDBT		
Type	I	I	I	I	I	I		
Default	none	1.0	none	none	1.0	none		

Card 3

Variable	ESAL	FMAL	ESBT	FMBT				
Type	F	F	F	F				
Default	0.0	0.0	0.0	0.0				

Card 4

Variable	SAAL	NSABT	PSABT					
Type	F	F	F					
Default	not used	not used	not used					

VARIABLE	DESCRIPTION
LCIDAL	Load curve ID for α–moment versus rotation in radians. See Figure 6.15 where it should be noted that $0 \le \alpha \le \pi$. If zero, the applied moment is set to zero. See *DEFINE_CURVE.
LCIDG	Load curve ID for γ versus a scale factor which scales the bending moment due to the α rotation. This load curve should be defined in the interval $-\pi \le \gamma \le \pi$. If zero the scale factor defaults to 1.0. See *DEFINE_ CURVE.
LCIDBT	Load curve ID for β–torsion moment versus twist in radians. If zero the applied twist is set to zero. See *DEFINE_CURVE.

VARIABLE	DESCRIPTION
DLCIDAL	Load curve ID for α–damping moment versus rate of rotation in radians per unit time. If zero, damping is not considered. See *DEFINE_CURVE.
DLCIDG	Load curve ID for γ–damping scale factor versus rate of rotation in radians per unit time. This scale factor scales the α–damping moment. If zero, the scale factor defaults to one. See *DEFINE_CURVE.
DLCIDBT	Load curve ID for β–damping torque versus rate of twist. If zero damping is not considered. See *DEFINE_CURVE.
ESAL	Elastic stiffness per unit radian for friction and stop angles for α rotation, see Figure 6.15. If zero, friction and stop angles are inactive for α rotation.
FMAL	Frictional moment limiting value for α rotation. If zero, friction is inactive for α rotation. This option may also be thought of as an elastic-plastic spring. If a negative value is input then the absolute value is taken as the load curve or table ID defining the yield moment versus α rotation, see Figure 6.15. A table permits the moment to also be a function of the joint reaction force and requires the specification of JID on Card 1.
ESBT	Elastic stiffness per unit radian for friction and stop angles for β twist, see Figure 6.15. If zero, friction and stop angles are inactive for β twist.
FMBT	Frictional moment limiting value for β twist. If zero, friction is inactive for β twist. This option may also be thought of as an elastic-plastic spring. If a negative value is input then the absolute value is taken as the load curve or table ID defining the yield moment versus β rotation, see Figure 6.15 A table permits the moment to also be a function of the joint reaction force and requires the specification of JID on Card 1.
SAAL	Stop angle in degrees for α rotation where $0 \le \alpha \le \pi$. Ignored if zero.
NSABT	Stop angle in degrees for negative β rotation. Ignored if zero.
PSABT	Stop angle in degrees for positive β rotation. Ignored if zero.

Remarks:

This option simulates a flexion-torsion behavior of a joint in a slightly different fashion than with the generalized joint option.

After the stop angles are reached the torques increase linearly to resist further angular motion using the stiffness values on Card 3. If the stiffness value is too low or zero, the stop will be violated.

The moment resultants generated from the moment versus rotation curve, damping moment versus rate-of-rotation curve, and friction are evaluated independently and are added together.

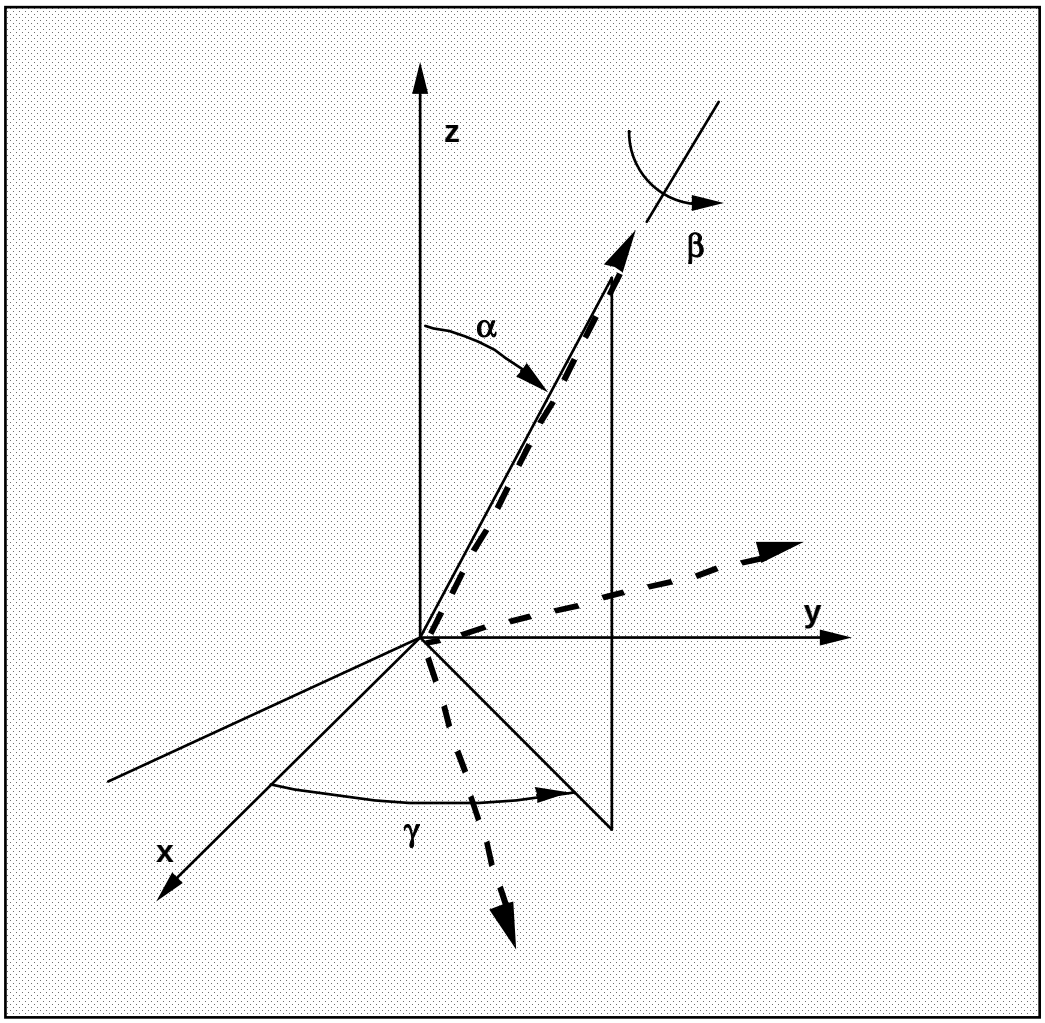


Figure 6.15. Flexion-torsion joint angles. If the initial positions of the local coordinate axes of the two rigid bodies connected by the joint do not coincide, the angles, α and γ, are initialized and torques will develop instantaneously based on the defined load curves. The angle β is also initialized but no torque will develop about the local axis on which β is measured. Rather, β will be measured relative to the computed offset.

Card 2-4 - Required for GENERALIZED stiffness.

Card 2

Variable	LCIDPH	LCIDT	LCIDPS	DLCIDPH	DLCIDT	DLCIDPS		
Type	I	I	I	I	I	I		
Default	none	none	none	none	none	none		

Card 3

Variable	ESPH	FMPH	EST	FMT	ESPS	FMPS		
Type	F	F	F	F	F	F		
Default	0.0	0.0	0.0	0.0	0.0	0.0		

Card 4

Variable	NSAPH	PSAPH	NSAT	PSAT	NSAPS	PSAPS		
Type	F	F	F	F	F	F		
Default	not used	not used	not used	not used	not used	not used		

VARIABLE	DESCRIPTION
LCIDPH	Load curve ID for ϕ–moment versus rotation in radians. See Figure 6.16. If zero, the applied moment is set to 0.0. See *DEFINE_CURVE.
LCIDT	Load curve ID for θ–moment versus rotation in radians. If zero, the applied moment is set to 0.0. See *DEFINE_CURVE.
LCIDPS	Load curve ID for ψ–moment versus rotation in radians. If zero, the applied moment is set to 0.0. See *DEFINE_CURVE.

VARIABLE	DESCRIPTION
DLCIDPH	Load curve ID for ϕ–damping moment versus rate of rotation in radians per unit time. If zero, damping is not considered. See *DEFINE_CURVE.
DLCIDT	Load curve ID for θ–damping moment versus rate of rotation in radians per unit time. If zero, damping is not considered. See *DEFINE_CURVE.
DLCIDPS	Load curve ID for ψ–damping torque versus rate of rotation in radians per unit time. If zero, damping is not considered. See *DEFINE_CURVE.
ESPH	Elastic stiffness per unit radian for friction and stop angles for ϕ rotation. See Figure 6.17. If zero, friction and stop angles are inactive for ϕ rotation.
FMPH	Frictional moment limiting value for ϕ rotation. If zero, friction is inactive for ϕ rotation. This option may also be thought of as an elastic-plastic spring. If a negative value is input then the absolute value is taken as the load curve or table ID defining the yield moment versus ϕ rotation. See Figure 6.17. A table permits the moment to also be a function of the joint reaction force and requires the specification of JID on card 1.
EST	Elastic stiffness per unit radian for friction and stop angles for θ rotation. See Figure 6.17. If zero, friction and stop angles are inactive for θ rotation.
FMT	Frictional moment limiting value for θ rotation. If zero, friction is inactive for θ rotation. This option may also be thought of as an elastic-plastic spring. If a negative value is input then the absolute value is taken as the load curve or table ID defining the yield moment versus θ rotation. See Figure 6.17. A table permits the moment to also be a function of the joint reaction force and requires the specification of JID on card 1.
ESPS	Elastic stiffness per unit radian for friction and stop angles for ψ rotation. See Figure 6.17. If zero, friction and stop angles are inactive for ψ rotation.
FMPS	Frictional moment limiting value for ψ rotation. If zero, friction is inactive for ψ rotation. This option may also be thought of as an elastic-plastic spring. If a negative value is input then the absolute value is taken as the load curve or table ID defining the yield moment versus

VARIABLE	DESCRIPTION
	ψ rotation. See Figure 6.17. A table permits the moment to also be a function of the joint reaction force and requires the specification of JID on card 1.
NSAPH	Stop angle in degrees for negative ϕ rotation. Ignored if zero.
PSAPH	Stop angle in degrees for positive ϕ rotation. Ignored if zero.
NSAT	Stop angle in degrees for negative θ rotation. Ignored if zero.
PSAT	Stop angle in degrees for positive θ rotation. Ignored if zero.
NSAPS	Stop angle in degrees for negative ψ rotation. Ignored if zero.
PSAPS	Stop angle in degrees for positive ψ rotation. Ignored if zero.

Remarks:

After the stop angles are reached the torques increase linearly to resist further angular motion using the stiffness values on Card 3. Reasonable stiffness values have to be chosen. If the stiffness values are too low or zero, the stop will be violated.

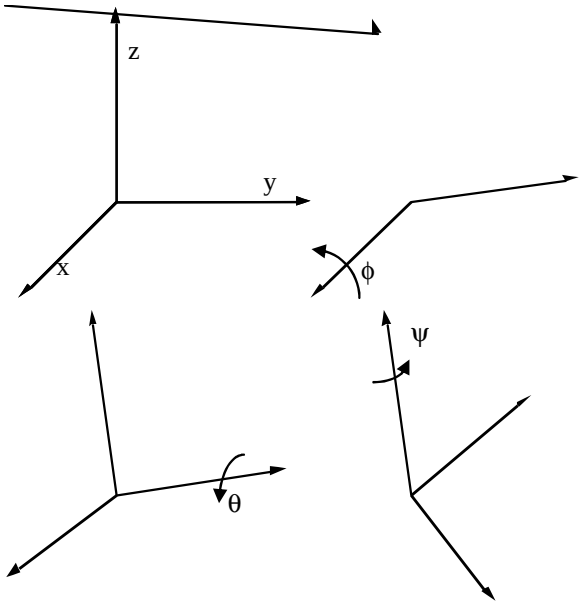


Figure 6.16. Definition of angles for the generalized joint stiffness. The magnitude of the angular rotations are limited by the stop angles defined on Card 4. If the initial local coordinate axes do not coincide, the angles, ϕ, θ, and ψ, will be initialized and torques will develop instantaneously based on the defined load curves.

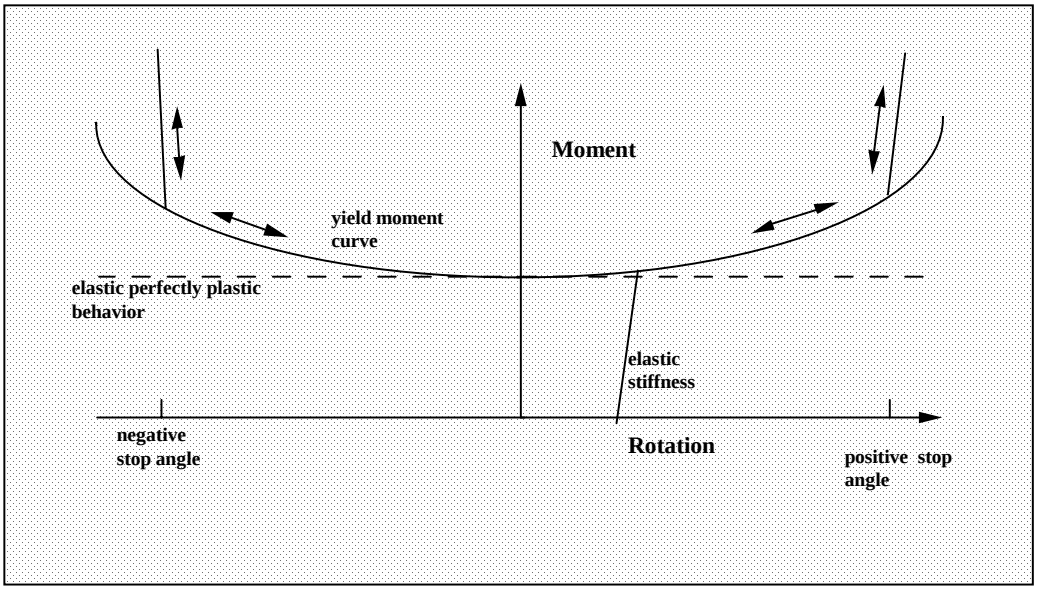


Figure 6.17. Frictional behavior is modeled by a plasticity model. Elastic behavior is obtained once the stop angles are reached. The same elastic stiffness is used to simulate sticking situations.

Card 2- 4 - Required for TRANSLATIONAL stiffness.

Card 2

Variable	LCIDX	LCIDY	LCIDZ	DLCIDX	DLCIDY	DLCIDZ		
Type	I	I	I	I	I	I		
Default	none	none	none	none	none	none		

Card 3

Variable	ESX	FFX	ESY	FFY	ESZ	FFZ		
Type	F	F	F	F	F	F		
Default	0.0	0.0	0.0	0.0	0.0	0.0		

Card 4

Variable	NSDX	PSDX	NSDY	PSDY	NSDZ	PSDZ		
Type	F	F	F	F	F	F		
Default	not used	not used	not used	not used	not used	not used		

VARIABLE	DESCRIPTION
LCIDX	Load curve ID for x–force versus x-translational relative displacement between the origins of CIDA and CIDB based on the x-direction of CIDB. If zero, the applied force is set to 0.0. See *DEFINE_CURVE. See Figure 6.18.
LCIDY	Load curve ID for y–force versus y-translational relative displacement between the origins of CIDA and CIDB based on the y-direction of CIDB. If zero, the applied force is set to 0.0. See *DEFINE_CURVE.

VARIABLE	DESCRIPTION
LCIDZ	Load curve ID for z–force versus z-translational relative displacement between the origins of CIDA and CIDB based on the z-direction of CIDB. If zero, the applied force is set to 0.0. See *DEFINE_CURVE.
DLCIDX	Load curve ID for x–damping force versus rate of x-translational displacement per unit time between the origins of CIDA and CIDB based on the x-direction of CIDB. If zero, damping is not considered. See *DEFINE_CURVE.
DLCIDY	Load curve ID for y–damping force versus rate of y-translational displacement per unit time between the origins of CIDA and CIDB based on the y-direction of CIDB. If zero, damping is not considered. See *DEFINE_CURVE.
DLCIDZ	Load curve ID for z–damping force versus rate of z-translational displacement per unit time between the origins of CIDA and CIDB based on the z-direction of CIDB. If zero, damping is not considered. See *DEFINE_CURVE.
ESX	Elastic stiffness for friction and stop displacement for x-translation. If zero, friction and stop angles are inactive for x-translation.
FFX	Frictional force limiting value for x-translation. If zero, friction is inactive for x-translation. This option may also be thought of as an elastic-plastic spring. If a negative value is input then the absolute value is taken as the load curve ID defining the yield force versus x-translation.
ESY	Elastic stiffness for friction and stop displacement for y-translation. If zero, friction and stop angles are inactive for y-translation.
FFY	Frictional force limiting value for y-translation. If zero, friction is inactive for y-translation. This option may also be thought of as an elastic-plastic spring. If a negative value is input then the absolute value is taken as the load curve ID defining the yield force versus y-translation.
ESZ	Elastic stiffness for friction and stop displacement for z-translation. If zero, friction and stop angles are inactive for z-translation..
FMZ	Frictional force limiting value for z-translation. If zero, friction is inactive for z-translation. This option may also be thought of as an elastic-plastic spring. If a negative value is input then the absolute value is taken as the load curve ID defining the yield force versus z-translation.

VARIABLE	DESCRIPTION
NSDX	Stop displacement for negative x-translation. Ignored if zero.
PSDX	Stop displacement for positive x-translation. Ignored if zero.
NSDY	Stop displacement for negative y-translation. Ignored if zero.
PSDY	Stop displacement for positive y-translation. Ignored if zero.
NSDZ	Stop displacement for negative z-translation. Ignored if zero.
PSDZ	Stop displacement for positive z-translation. Ignored if zero.

Remarks:

After the stop displacements are reached the force increases linearly to resist further translational motion using the stiffness values on Card 3. Reasonable stiffness values must be chosen. If the stiffness values are too low or zero, the stop will be violated.

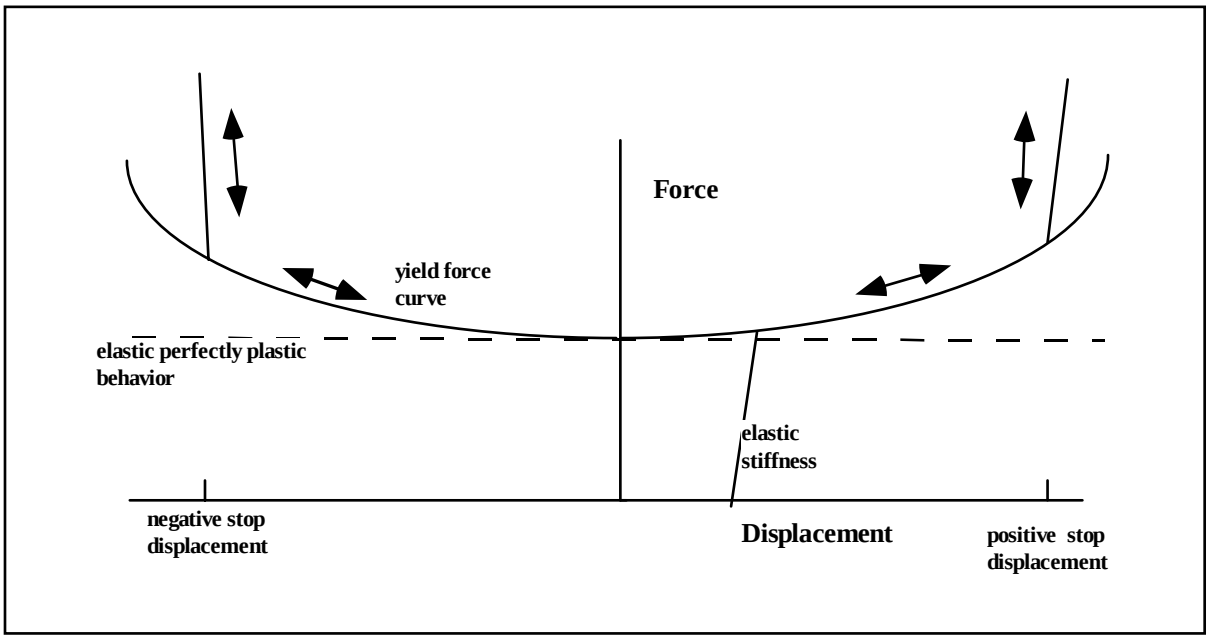


Figure 6.18. Frictional behavior is modeled by a plasticity model. Elastic behavior is obtained once the stop displacements are reached. The same elastic stiffness is used to simulate sticking situations.

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$  *CONSTRAINED_JOINT_STIFFNESS_GENERALIZED
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Define a joint stiffness for the revolute joint described in
$    *CONSTRAINED_JOINT_REVOLUTE
$
$  Attributes of the joint stiffness:
$    - Used for defining a stop angle of 30 degrees rotation
$        (i.e., the joint allows a positive rotation of 30 degrees and
$           then imparts an elastic stiffness to prevent further rotation)
$    - Define between rigid body A (part 1) and rigid body B (part 2)
$    - Define a local coordinate system along the revolute axis
$        on rigid body A - nodes 1, 2 and 3 (cid = 5).  This is used to
$        define the revolute angles phi (PH), theta (T), and psi (PS).
$    - The elastic stiffness per unit radian for the stop angles
$        are 100, 10, 10 for PH, T, and PS, respectively.
$    - Values not specified are not used during the simulation.
$
*CONSTRAINED_JOINT_STIFFNESS_GENERALIZED
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$     jsid      pida      pidb      cida      cidb
         1         1         2         5         5
$
$   lcidph     lcidt    lcidps   dlcidph    dlcidt   dlcidps

$
$     esph      fmps       est       fmt      esps      fmps
     100.0                10.0                10.0
$
$    nsaph     psaph      nsat      psat     nsaps     psaps
                30.0
$
$
*DEFINE_COORDINATE_NODES
$      cid        n1        n2        n3
         5         1         2         3
$
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

*CONSTRAINED_JOINT_USER_FORCE

Purpose: Define input data for a user subroutine to generate force resultants as a function of time and joint motion.

Card 1

Variable	FID	JID	NHISV					
Type	I	I	I					
Default	none	none	0					

Define up to 48 optional user constants for the user subroutine. This input is terminated after 48 constants are defined or when the next "*" keyword card is encountered.

Card 2,3,…

Variable	CONST1	CONST2	CONST3	CONST4	CONST5	CONST6	CONST7	CONST8
Type	F	F	F	F	F	I	I	F
Default	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0

VARIABLE	DESCRIPTION
FID	Joint user force ID.
JID	Joint ID for which this user force input applies.
NHISV	Number of history variables required for this definition. An array NHISV long is allocated and passed into the user subroutine. This array is updated in the user subroutine.
CONST*n*	A constant which is passed into the user subroutine.

***CONSTRAINED_LAGRANGE_IN_SOLID_**{*OPTION1*}_{*OPTION2*}

Purpose: This command provides the coupling mechanism for modeling Fluid-Structure Interaction (FSI). The structure can be constructed from Lagrangian shell and/or solid entities. The multi-material fluids are modeled by ALE formulation (see Remark 15).

Note: For RIGID slave PARTS a penalty coupling method (CTYPE=4) must be used, see parameter CTYPE below.

Available options for *OPTION1* include:

<BLANK>

EDGE

This option may be used to allow the coupling between the edge of a shell part or part set and one or more ALE multi-material groups (AMMG). It accounts for the shell thickness in the coupling calculation. The edge thickness is the same as the shell thickness. This option only works when the Lagrangian slave set is defined as a part or a part set ID. It will not work for a slave segment set. One application of this option is a simulation of a Lagrangian blade (a shell part) cutting through some ALE material.

Available options for *OPTION2* include:

<BLANK>

TITLE

To define a coupling (card) ID number and title for each coupling card. If a title is not defined LS-DYNA will automatically create an internal title for this coupling definition. The ID number can be used to delete coupling action in a restart input deck via the *DELETE_FSI card.

Card Format (I10, A70)
The following card is read if and only if the TITLE option is specified.

Optional 1 2

Variable	COUPID	TITLE
Type	I	A70

Card 1 is mandatory for all coupling definitions.

Card 1	1	2	3	4	5	6	7	8
Variable	SLAVE	MASTER	SSTYP	MSTYP	NQUAD	CTYPE	DIREC	MCOUP
Type	I	I	I	I	I	I	I	I
Default	none	none	0	0	0	2	1	0

Card 2 is mandatory for all coupling definitions.

Card 2	1	2	3	4	5	6	7	8
Variable	START	END	PFAC	FRIC	FRCMIN	NORM	NORMTYP	DAMP
Type	F	F	F	F	F	I	I	F
Default	0	1.0E10	0.1	0.0	0.5	0	0	0.0

Card 3 is mandatory for all coupling definitions.

Card 3	1	2	3	4	5	6	7	8
Variable	CQ	HMIN	HMAX	ILEAK	PLEAK	LCIDPOR	NVENT	IBLOCK
Type	F	F	F	I	F	I	I	I
Default	0.0	none	none	0	0.1	none	0	0

Optional Card 4a may be defined with or without 4b. It is required for CTYPE 11 & 12. (define THKF if CTYPE=11).

Card 4a	1	2	3	4	5	6	7	8
Variable	IBOXID	IPENCHK	INTFORC	IALESOF	LAGMUL	PFACMM	THKF	
Type	I	I	I	I	F	I	F	
Default	0	0	0	0	0.0	0	0.0	

Optional Card 4b is required for CTYPE 11 & 12. If 4b is defined, 4a must be defined before 4b.

Card 4b	1	2	3	4	5	6	7	8
Variable	A1	B1	A2	B2	A3	B3		
Type	F	F	F	F	F	F		
Default	none	none	none	none	none	none		

Optional 4c card(s) defining venting geometry. It is repeated NVENT times (one line for defining each vent hole). It is defined only if NVENT > 0 in card 3. If either or both 4a and 4b are defined, they are defined before card(s) 4c.

Card 4c	1	2	3	4	5	6	7	8
Variable	VENTSID	VENTYP	VTCOEF	POPPRES	COEFLC			
Type	I	I	I	F	I			
Default	0	0	0	0.0	0			

VARIABLE	DESCRIPTION
COUPID	Coupling (card) ID number (I10). This ID can be used in a restart input deck to delete or reactivate this coupling action via the *DELETE_FSI card. If not defined, LSDYNA will assign an internal coupling ID based on the order of appearance in the input deck.
TITLE	A description of this coupling definition (A70).
SLAVE	Slave set ID defining a part, part set or segment set ID of the Lagrangian or slave structure (see *PART, *SET_PART or *SET_SEGMENT). See Remark 1.
MASTER	Master set ID defining a part or part set ID of the ALE or master solid elements (see *PART or *SET_PART, and see Remark 1).
SSTYP	Slave set type of "SLAVE" (see Remark 1): EQ.0: part set ID (PSID). EQ.1: part ID (PID). EQ.2: segment set ID (SGSID).
MSTYP	Master set type of "MASTER" (see Remark 1): EQ.0: part set ID (PSID). EQ.1: part ID (PID).
NQUAD	Number of coupling points distributed over each coupled Lagrangian surface segment. EQ.0: NQUAD will be set by default to 2, EQ.n: An nXn coupling points distribution over each Lagrangian segment is defined, EQ.-n: NQUAD is reset to a positive value. Coupling at nodes is obsolete.
CTYPE	Fluid-Structure coupling method: EQ.1: constrained acceleration. EQ.2: constrained acceleration and velocity (default, see Remark 3). EQ.3: constrained acceleration and velocity, normal direction only. EQ.4: penalty coupling for shell (with or without erosion) and solid elements (without erosion). EQ.5: penalty coupling allowing erosion in the Lagrangian entities (solid elements, see Remark 3). EQ.6: penalty coupling designed for airbag modeling which automatically controls the DIREC parameter internally. It is equivalent to setting {CTYPE=4; DIREC=1} for unfolded region; and {CTYPE=4; DIREC=2}; in folded region. For both cases: {ILEAK=2; FRCMIN=0.3}.

VARIABLE	DESCRIPTION
	EQ.11: coupling designed to couple Lagrangian porous shell to ALE material. When this option is used, THKF, the 7th column parameter of optional card 4a and the first 2 parameters of optional card 4b must be defined. See *LOAD_BODY_POROUS and remark 14 below. EQ.12: coupling designed to couple Lagrangian porous solid to ALE material. When this option is used, A_i & B_i parameters of optional card 4b must be defined (card 4a must be defined but can be blank). See *LOAD_BODY_POROUS and Remark 14 below.
DIREC	Coupling direction (CTYPE 4 and 5, see Remark 4). EQ.1: normal direction, compression and tension (default) EQ.2: normal direction, compression only, EQ.3: all directions.
MCOUP	Multi-material option (CTYPE 4, 5, 6, 11 and 12, see Remark 5). EQ.0: couple with all multi-material groups, EQ.1: couple with material with highest density. EQ.-n: refers to a set ID of an ALE multi-material groups defined in *SET_MULTI-MATERIAL_GROUP card in which its set ID=n.
START	Start time for coupling.
END	End time for coupling.
PFAC	Penalty factor (CTYPE 4, 5 and 6). PFAC is a scale factor for scaling the estimated stiffness of the interacting (coupling) system. It is used to compute the coupling forces to be distributed on the slave and master parts. GT.0.0: Fraction of estimated critical stiffness. LT.0: -n: where n refers to a load curve ID. The curve defines the coupling pressure (y-axis) as a function of the penetration (x-axis) (See Remark 6).
FRIC	Coefficient of friction (used with DIREC 2 only).
FRCMIN	Minimum volume fraction of a coupled ALE multi-material group (AMMG) or fluid in a multi-material ALE element to activate coupling. Default value is 0.5. Reducing FRCMIN (typically, between 0.1 and 0.3) would turn on coupling earlier to prevent leakage in hypervelocity impact cases.
NORM	A flag indicating the rule for defining which side of the Lagrangian segment the fluid is supposed to be coupled to. By default (NORM=0) the fluid on the side pointed to by the Lagrangian segment normal (head-side) is coupled to. To couple to the fluid on the side not pointed to by the segment normals (tail-side), set NORM=1 (see Remark 7). EQ.0: Couple fluid to head-side of Lagrangian segment. EQ.1: Couple fluid to tail-side of Lagrangian segment.

VARIABLE	DESCRIPTION
NORMTYP	Penalty coupling spring (or force) direction (DIREC 1 and 2): EQ.0: normal vectors are interpolated from nodal normals (default). EQ.1: normal vectors are interpolated from segment normals. This is sometimes a little more robust for sharp Lagrangian corners, and folds.
DAMP	Damping factor for penalty coupling. This is a coupling-damping scaling factor. Typically it may be between 0 and 1 (see Remark 8).
CQ	Heat transfer coefficient, C_q (see Remark 9).
HMIN	Minimum air gap in heat transfer, h_{min} (see Remark 9).
HMAX	Maximum air gap in heat transfer, h_{max}. There is no heat transfer above this value (see Remark 9).
ILEAK	Coupling leakage control flag (Remark 10): EQ.0: none (default), EQ.1: weak, leakage control is turned off if the penetrating volfrac > FRCMIN+0.1. EQ.2: strong, with improved energy consideration. Leakage control is turned off if the penetrating volfrac > FRCMIN+0.3.
PLEAK	Leakage control penalty factor, 0 < PLEAK < 0.2 is recommended. This factor influences the additional coupling force magnitude to prevent leakage. It is conceptually similar to PFAC. Almost always, the default value (0.1) is adequate.
LCIDPOR	If this is a positive integer: A load curve ID (LCID) defining porous flow through coupling segment: Abscissa = x = (P_{up} - P_{down}) Ordinate = y = relative porous fluid velocity Where P_{up} and P_{down} are, respectively, the upstream and downstream pressures across of the porous coupling segment. The relative porous velocity is the ALE fluid velocity relative to the moving Lagrangian segment. This experimental data curve must be provided by the user. If LCIDPOR is a negative integer: The porous flow is controlled by the parameters FLC, FAC, ELA under *MAT_FABRIC card. *CAUTION*: The pressure under the FAC load curve is "absolute upstream pressure" (see Remark 11). Abscissa = x = absolute upstream pressure Ordinate = y = relative porous fluid velocity

VARIABLE	DESCRIPTION
NVENT	The number of vent surface areas to be defined. Each venting flow surface is represented by one or more Lagrangian segments (or surfaces). For airbag application, this may be referred to as "isentropic" venting where the isentropic flow equation is used to compute the mass flow rate across an area given P_{up}/P_{down} ratio. If NVENT > 0, there will be NVENT number of lines required (optional cards 4c). Each line defines the geometrical and flow information for each vent surface: VENTSID, VNTYPE, VTCOEF, POPPRES, COEFLC. The vented mass will simply be deleted from the system and cannot be visualized as in the case of physical venting (see Remark 12).
IBLOCK	Flag (1=ON or 0=OFF) to control the venting (or porous) flow blockage due to Lagrangian contact during ALE computation. The venting definition is defined in this command. However, the venting flow may be defined via either the LCIDPOR parameter in this command or via the *MAT_FABRIC parameters (FLC, FAC, ELA). However, note that FVOPT (blocking) parameter under *MAT_FABRIC applies only to CV computation.
IBOXID	A box ID defining a box region in space in which ALE coupling is activated. At time=0.0, the number of Lagrangian segments inside this box is remembered. In subsequent coupling computation steps, there is no need to search for the Lagrangian segments again.
IPENCHK	Initial penetration check flag (only for CTYPE=4, Remark 13): EQ.0: Do not check for initial penetration. EQ.1: Check and save initial ALE material penetration across a Lagrangian surface (d_0), but do not activate coupling at t=0. In subsequent steps (t>0) the actual penetration is computed as follows actual penetration = total penetration – initial penetration $d_a = d_T - d_0$
INTFORC	A flag to turn on or off (0=OFF or 1=ON) the output of ALE coupling pressure and forces on the slave Lagrangian segments (or surfaces). Note that the coupling pressures and forces are computed based on the ALE fluid penetrations and coupling stiffness of the system. When (1) INTFORC=1 and (2) a *DATABASE_BINARY_FSIFOR (DBF) card is defined, LS-DYNA writes out the segment coupling pressure and forces to the binary interface force file for contour plotting. This interface force file is activated by executing ls971 as follows (3): **ls971 i=inputfilename.k … h=interfaceforcefilename** The time interval between output is defined by "dt" in the DBF card. To plot the binary data in this file: **lsprepost interfaceforcefilename**.

VARIABLE	DESCRIPTION
IALESOF	An integer flag to turn ON/OFF a supplemental Lagrange multiplier FSI constraint which provides a coupling force in addition to the basic penalty coupling contribution. This is a hybrid coupling method. EQ.0: OFF (default). EQ.1: Turn ON the hybrid Lagrange-multiplier method. LAGMUL multiplier factor is read.
LAGMUL	A Lagrange multiplier factor with a range between 0.0 and 0.05 may be defined. A typical value may be 0.01. This should never be greater than 0.1. EQ.0: OFF (default). GT.0: Turn ON the Lagrange-multiplier method and use LAGMUL as a coefficient for scaling the penalty factor.
PFACMM	Mass-based penalty stiffness factor computational options. This works in conjunction with PFAC=constant (not a load curve). The coupling penalty stiffness (CPS) is computed based on an estimated effective coupling mass. EQ.0: $CPS \propto PFAC \bullet \min(m_{slave}, m_{master})$, default. EQ.1: $CPS \propto PFAC \bullet \max(m_{slave}, m_{master})$. EQ.2: $CPS \propto PFAC \bullet \sqrt{m_{slave} m_{master}}$, geometric-mean of the masses. EQ.3: $CPS \propto PFAC \bullet K_{Lagrangian}$ where K is the bulk modulus of the slave or Lagrangian part
THKF	(For all CTYPE choices except 11) A flag to account for the coupling thickness of the Lagrangian shell (slave) part. LT.0: Use positive value of |THKF| for coupling segment thickness. EQ.0: Do not consider coupling segment thickness. GT.0: Coupling segment thickness scale factor. For CTYPE=11 case (see Remark 14): This thickness is required for volume calculation. GT.0: (Fabric) Thickness scale factor. The base shell thickness is taken from the *PART definition. LT.0: User-defined (Fabric) thickness. The fabric thickness is set to |THKF|.
A1	Viscous coefficient for the porous flow Ergun equation (see Remark 14). For CTYPE=11, A1 = A_n = coefficient for normal-to-segment direction. For CTYPE=12: A1 = A_x = coefficient for global X-direction.
B1	Inertial coefficient for the porous flow Ergun equation (see Remark 14). For CTYPE=11, B1 = B_n = coefficient for normal-to-segment direction. For CTYPE=12: B1 = B_x = coefficient for global X-direction.

VARIABLE	DESCRIPTION
A2	Viscous coefficient for the porous flow Ergun equation (see Remark 14). For CTYPE=12: A2 = A_y = coefficient for global Y-direction.
B2	Inertial coefficient for the porous flow Ergun equation (see Remark 14). For CTYPE=12: B2 = B_y = coefficient for global Y-direction.
A3	Viscous coefficient for the porous flow Ergun equation (see Remark 14). For CTYPE=12: A3 = A_z = coefficient for global Z-direction.
B3	Inertial coefficient for the porous flow Ergun equation (see Remark 14). For CTYPE=12: B3 = B_z = coefficient for global Z-direction.
VENTSID	Set ID of the vent hole shape.
VENTYP	Vent surface area set ID type: EQ.0: Part set ID (PSID). EQ.1: Part ID (PID). EQ.2: Segment set ID (SGSID).
VTCOEF	Flow coefficient for each vent surface area.
POPPRES	Venting pop pressure limit. If the pressure inside the airbag is lower than this pressure, then nothing is vented. Only when the pressure inside the airbag is greater than POPPRES that venting can begin.
COEFLC	A time-dependent multiplier load curve for correcting the vent flow coefficient, with values ranging from 0.0 to 1.0.

Remarks:

1. In order for a fluid-structure interaction (FSI) to occur, a Lagrangian (structure or slave) mesh must spatially overlap with an ALE (fluid or master) mesh. Each mesh should be defined with independent node IDs. LS-DYNA searches for the spatial intersection of between the Lagrangian and ALE meshes. Where the meshes overlap, there is a possibility that interaction may occur. In general, SLAVE, MASTER, SSTYP and MSTYPE are required definitions for specifying overlapping-domains coupling search.

2. The number of coupling points, NQUADXNQUAD, is distributed over the surface of each Lagrangian segment. Generally, 2 or 3 coupling points per each Eulerian/ALE element width is adequate. Consequently, the appropriate NQUAD values must be estimated based on the relative resolutions between the Lagrangian and ALE meshes. Consider Case 1 where 1 Lagrangian shell element spans, say, 2 ALE elements. Then NQUAD for each Lagrangian segment should be 4 or 6. Consider Case 2 where 2 or 3 Lagrangian segments span 1 ALE element, then maybe NQUAD=1 would be adequate. If either mesh compresses or expands during the interaction, the number of coupling points per ALE element will also change. The user must account for this and try to maintain at least 2 coupling points per each ALE element side length during the whole process to

prevent leakage. Too many coupling points can result in instability, and not enough can result in leakage.

3. CTYPE=2 is sometimes used to couple, via constraint method (energy not conserved), Lagrangian beam nodes to ALE solid, for example, in the modeling of rebar in concrete, or tire cords in rubber. The slave set is coupled to the master set. Constraint based method does not try to conserve energy and is seldom used. Penalty approach is the current method of choice. For better accuracy, when using the penalty method, it is better to couple to a specific set of AMMGs (set MCOUP to a negative integer, see MCOUP).

4. DIREC=2 may be generally a more stable and robust choice for coupling direction. However a choice of when to activate coupling should be made based on the physics of the problem. DIREC=1 couples under both tension and compression. This is sometimes useful as in the case of suddenly accelerating liquid container. DIREC=3 is rarely applicable to real physics (because it simulates an extremely sticky fluid).

5. When MCOUP is a negative integer, say for example MCOUP= -123, then an ALE multi-material set-ID (AMMSID) of 123 must exist. This is an ID defined by a *SET_MULTI-MATERIAL_GROUP_LIST card. This generally seems to be a better approach to couple to a specific set of AMMGs, and have a clearly defined fluid interface interacting with a Lagrangian surface. That way, any leakage may be visualized and the penalty force can be computed more precisely.

6. The user can usually start with PFAC=default (0.1). If leakage occurs, leakage control may be turned on, ILEAK=2.

   The next thing to try may be using a load curve for PFAC. When PFAC is a negative integer, for example PFAC= -321, then a load curve with LCID=321 must exist via a *DEFINE_CURVE card. This choice allows the application of an estimated coupling pressure (y-axis) given a penetration distance (x-axis). The curve consists of {0,0} as the first point and {maximum allowable penetration (MAP), estimated maximum coupling pressure (EMCP)} as a second point. MAP may be a small penetration with respect to the minimum ALE element width (maybe 10% or less). EMCP can be estimated from a maximum fluid pressure observed from a previous run when leakage first occurs. This curve may be scaled to vary the stiffness of the coupling spring. The approach is to gradually increase the coupling stiffness until leakage stops. The best coupling stiffness is one which provides just enough force to prevent leakage and not more.

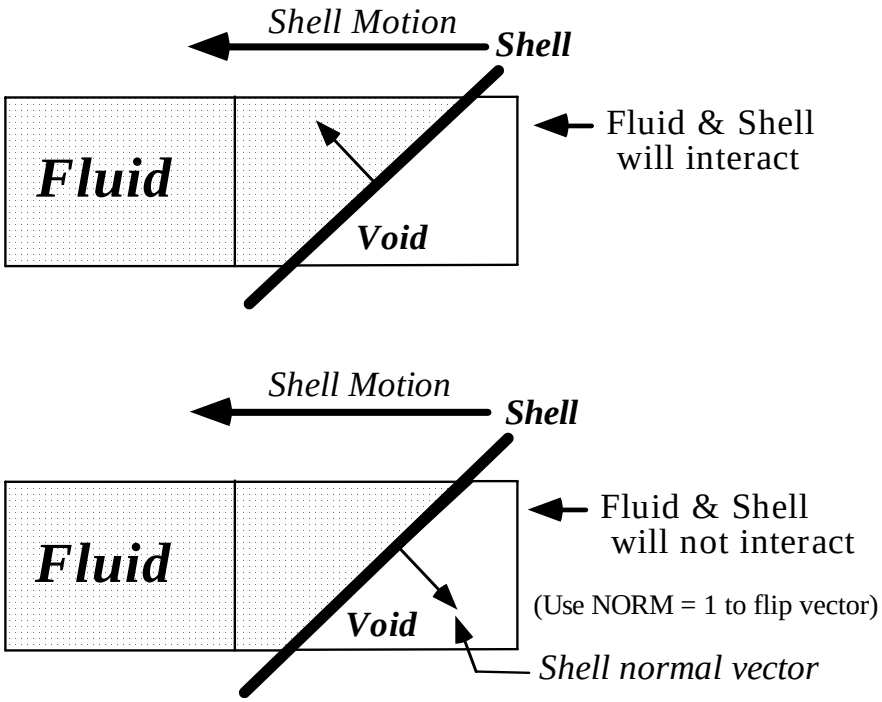
   A 3rd approach is to try PFACMM=3 on optional card 4a (with PFAC=constant).

   Consider a coupling between a Lagrangian airbag with the ALE inflator gas, a penetration of 1.0E-3 m of the gas across the bag surface will trigger a coupling pressure (say, about 4 atm or 405300 Pascals). The coupling pressure for arbitrary penetration is scaled from this curve.

   This example demonstrates the usage of both MCOUP and PFAC when they are negative integers.

```
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|....8
$ PID 21 = ALE inflator gas;  PID 22 = air mesh surrounding the airbag
*ALE_MULTI-MATERIAL_GROUP
$      SID    IDTYPE
        21         1
        22         1
$ ALEMMGID = 1 <= PID 21 <== see 1st line of the *ALE_MULTI-MATERIAL_GROUP card.
*SET_MULTI-MATERIAL_GROUP_LIST
$   AMMSID
       123
$ ALEMMGID
         1
$ Lagrangian = slave = PSID 1 contains all airbag parts
*SET_PART
$      SID       DA1       DA2       DA3       DA4
         1        0.        0.        0.        0.
$     PID1      PID2      PID3      PID4      PID5      PID6      PID7      PID8
         3         4         5         6         7         8         9         0
$ Eulerian = master = PSID 11 contains all fluid mesh (geometrical space to search).
*SET_PART_LIST
        11
         2
*CONSTRAINED_LAGRANGE_IN_SOLID
$    SLAVE    MASTER     SSTYP     MSTYP     NQUAD     CTYPE     DIREC     MCOUP
         1        11         0         0         4         4         2      -123
$    START       END      PFAC      FRIC    FRCMIN      NORM
       0.0       0.0      -321      0.00       0.3         1
$       CQ      HMIN      HMAX     ILEAK     PLEAK VLK_PLCID
        0         0         0         2      0.10
$ We should couple to only the inflator gas ➔ MCOUP = -123
$ fluid penetration ~ 1 mm  <====>  Pmax = 405300 pascal ~ 4 atm
*DEFINE_CURVE
       321
                  0.0000               0.0
                  1.0e-3          405300.0
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|....8
```

7. The normal vectors (NV) of a Lagrangian shell part are defined by the order of the nodes in *ELEMENT definitions, via the right hand rule, and for a segment set, the order of nodes defined in *SET_SEGMENT. Let the side pointed to by NV be “positive”. The penalty method measure penetration as the distance the ALE fluid penetrates from the positive side to the negative side of the Lagrangian segment. Only fluid on the positive side will be “seen” and coupled to. Hence, all normal vectors of the Lagrangian segments should point uniformly toward the ALE fluid(s), AMMGs, to be coupled to. If NV point uniformly away from the fluid, coupling is not activated. In this case, coupling can be activated by setting NORM=1. Sometimes a shell part or mesh is generated such that its normal vectors do not point uniformly in a consistent direction (all toward the inside or outside of a container, etc.) The user should always check for the normal vectors of any Lagrangian shell part interacting with any fluid. The NORM parameter may be used to flip the normal direction of all the segments included in the Lagrangian slave set.



8. The user-input coupling-damping factor (DAMP) is used to scale down the critical-damping force (~ damper constant*velocity). For a mass-to-rigid-wall system connected by a parallel-spring-damper connector, we can obtain solution for a critically-damped case. DAMP is a factor for scaling down the amount of damping, with DAMP=1 being a critically-damped case.

9. The method used is similar to that done by *CONTACT_…_THERMAL_… card, except radiation heat transfer is not considered. A gap (l) is assumed to exist between the 2 materials undergoing heat transfer (one is Lagrangian and the other ALE). The convection heat transfer in the gap is assumed to approach simple conduction across the medium in the gap.

$$q = \kappa \frac{dT}{dx} \sim h\Delta T \Rightarrow$$

$$h \sim \frac{\kappa}{l}$$

The heat flux is typically defined as an energy transfer rate per unit area, $q \sim \frac{[J/s]}{m^2}$. κ is the thermal conductivity of the material in the gap, h, the equivalent convection heat transfer coefficient, and ΔT, the temperature difference between the master and slave sides. There are 3 possible scenarios:

(a) $l > l_{max}$ ➔ No heat transfer.

(b) $l_{\min} \le l \le l_{\max}$ ➔ $h \sim \dfrac{\kappa}{\max(l_{\min}, l)}$.

(c) $l < l_{\min}$ ➔ $h \sim \dfrac{\kappa}{l_{\min}}$.

CQ (κ), **HMIN ($l_{\min}$), HMAX ($l_{\max}$)** are defined for this heat transfer estimation.

10. In general, a coupling force stopping "fluid" leakage across a Lagrangian surface should come predominantly from the (penalty) coupling force. Leakage control force should be of secondary effect and should not be bigger than that from the main penalty coupling action. *DATABASE_FSI command output (dbfsi) allows for the monitoring of both the coupling forces and the leakage control force contribution. It may be used as a debugging or fine-tuning tool for coupling design. ILEAK=2 has a slightly more accurate energy accounting algorithm, thus is better for airbag applications. Leakage control should only be turned on when coupling to a specific AMMG (MCOUP as a negative integer) with its fluid interface clearly defined and tracked via the *ALE_MULTI-MATERIAL_GROUP card.

11. There are currently 2 methods to model porous flow across a Lagrangian shell structure. Both methods involve defining an empirical data curve of relative porous gas velocity as a function of system pressure. However the pressure definitions are slightly different depending on the choice of parameter defined:

    a) Via LCIDPOR parameter under *CONSTRAINED_LAGRANGE_IN_SOLID (CLIS). If this option is used the data curve contains ($P_{upstream} - P_{downstream}$) in the x-axis of the curve.

    b) Via *MAT_FABRIC's FLC, FAC, ELA parameters. If LCIDPOR is negative, and FAC defines a load curve, then this data curve contains absolute upstream pressure (not pressure difference) in the x-axis.

    When *AIRBAG_ALE is used, it assumes that absolute upstream P is given in the curve defined by FAC under *MAT_FABRIC. During CV phase it uses that. In ALE phase, LS-DYNA automatically subtracts 1 atmospheric pressure from the given pressure in the FAC curve. Thus giving it a gauge pressure for the porous coupling calculation. The amount of accumulated mass flowing across a porous Lagrangian surface may be tracked via the *DATABASE_FSI card ("pleak" parameter in the "dbfsi" ASCII output file).

12. There are 2 methods to model (airbag) venting. The accumulated mass output of both may be tracked via the *DATABASE_FSI card ("pleak" parameter in the "dbfsi" ASCII output file).

    a) In isentropic venting, (define NVENT on card 3) the flow crossing the vent hole surface is estimated from isentropic equation. All airbag shell normal vectors should point uniformly in the same direction (typically, inward). The shell elements for the

vent holes, included in the Lagrangian coupling set, should also point in the same direction as the airbag (i.e., inward).

b) In physical venting, there can be either physical holes in the Lagrangian structure (airbag), or shell parts covering the holes would have their normal vectors pointing outward. Either way, there is no coupling force to stop fluid leakage. It is recommended that *ALE_FSI_SWITCH_MMG_ID (AFSM) be used to switch the AMMG ID of the vented gas so that the vented gas outside the bag does not cause leakage (when the same AMMG is present on both sides of the same Lagrangian shell surface, penalty coupling can break down). The shell parts representing the vent holes may be either (i) excluded form the Lagrangian coupling set, or (ii) if included, they should have their normals pointing in opposite direction with respect to the rest of the airbag (typically outward since the rest of the airbag have their normals pointing inward).

13. Typically, penetration check (IPENCHK) should only be used if there is high coupling force applied at t=0. For example, consider a Lagrangian container, filled with non-gaseous fluid (i.e. ALE liquid or solid) via the *INITIAL_VOLUME_FRACTON_ GEOMETRY command. Sometimes due to mesh resolution or complex container geometry, there is initial penetration of the fluid across the container surface. This can give rise to a sharp and immediate coupling force on the fluid at t=0. Turning on IPENCHK may help eliminate this spike in coupling force.

14. For shell, CTYPE=11, the Ergun-type empirical porous flow equation is applied to the normal flow direction across the porous surface. The pressure gradient along the segment normal direction is

$$\frac{dP}{dx_n} = A_n(\varepsilon,\mu)V_n + B_n(\varepsilon,\rho)|V_n|V_n$$

where the subscript “n” refers to the direction normal to the porous Lagrangian shell surface.

V_n is the relative normal-to-porous-shell-surface fluid velocity component.

$A_n(\varepsilon,\mu) = A_1(\varepsilon,\mu)$ is a viscous coefficient of the Ergun-type porous flow equation. As applied here it should contain the fluid dynamic viscosity (μ) and shell porosity (ε) information.

$B_n(\varepsilon,\rho) = B_1(\varepsilon,\rho)$ is an inertial coefficient of the Ergun-type porous flow equation. As applied here it should contain the fluid density (ρ) and shell porosity (ε) information.

The force increment applied per segment is

$F_n = \frac{dp}{dx_n} \cdot t \times S$ where

S is the segment surface area.

t is the shell thickness (THKF).

A1 ($A_1(\varepsilon,\mu)$), B1 ($B_1(\varepsilon,\mu)$) and THKF (t) are required input for porous shell coupling.

For porous solid, CTYPE=12, the pressure gradient along each global direction (i) maybe computed similarly.

$$\frac{dP}{dx_i} = A_i(\varepsilon,\mu)V_i + B_i(\varepsilon,\rho)|V_i|V_i$$

$$i = 1,2,3$$

V_i is the relative fluid velocity component through the porous solid in the 3 global directions.

$A_i(\varepsilon,\mu)$ is a viscous coefficient of the Ergun-type porous flow equation in the i[th] direction. As applied here it should contain the fluid dynamic viscosity (μ) and shell porosity (ε) information.

$B_i(\varepsilon,\rho)$ is an inertial coefficient of the Ergun-type porous flow equation in the i[th] direction. As applied here it should contain the fluid density (ρ) and solid porosity (ε) information.

A_i ($A_i(\varepsilon,\mu)$), B_i ($B_i(\varepsilon,\mu)$) are required input for porous solid coupling.

Currently, only constant porosity structures are considered. See also *LOAD_BODY_POROUS.

15. Due to the complexity of this card, some comments on simple, efficient and robust coupling approach are given here. These are not rigid guidelines, but simply some experience-based observations.

The term “fluid”, in the Fluid-Structure Interaction (FSI), refers to materials with ALE element formulation, not indicating the phase (solid, liquid or gas) of those materials. In fact, solid, liquid and gas can all be modeled by the ALE formulation. The term “structure” refers to materials with Lagrangian element formulation.

In general, penalty coupling (CTYPE 4 & 5) is recommended, and MCOUP=negative integer is a better choice to define a specific ALE multi-material group (AMMG) to be coupled to the Lagrangian surface. At the minimum, all parameters on card 1 are to be specified, and the default values for most are good starting choices (except MCOUP).

If there is leakage, PFAC, FRCMIN, NORMTYPE and ILEAK are the 4 parameters that can be adjusted.

For hard structure (steel) and very compressible fluid (air), PFAC may be set to 0.1 (or higher). PFAC=constant value.

Next, keeping PFAC=constant and set PFACMM=3 (optional card 4a). This option scales the penalty factor by the bulk modulus of the Lagrangian structure. This new approach has also shown to be effective for some airbag application.

The next approach may be switching from constant PFAC to a load curve approach (i.e. PFAC=load curve, and PFACMM=0). By looking at the pressure in the system near leakage original location, we can get a feel for the pressure required to stop it.

If leakage persists after some iterations on the coupling force controls, one can subsequently try to set ILEAK=2 in combination with the other controls to prevent leakage.

If the modifications fail to stop the leakage, maybe the meshes have to be redesigned to allow better interactions between the Lagrangian and Ale materials.

In the example below, the underlined parameters are usually defined parameters. A full card definition is shown for reference.

```
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|....8
*CONSTRAINED_LAGRANGE_IN_SOLID
$     SLAVE     MASTER      SSTYP      MSTYP      NQUAD      CTYPE      DIREC      MCOUP
          1         11          0          0          4          4          2       -123
$     START       END       PFAC       FRIC     FRCMIN       NORM   NORMTYPE       DAMP
        0.0       0.0        0.1       0.00        0.3          0          0        0.0
$        CQ      HMIN      HMAX      ILEAK     PLEAK   LCIDPOR      NVENT     IBLOCK
           0         0         0          0       0.0          0          0          0
$4A IBOXID   IPENCHK   INTFORC   IALESOF    LAGMUL    PFACMM       THKF
$          0         0         0          0         0         0          0
$4B      A1        B1        A2         B2        A3        B3
$       0.0       0.0       0.0        0.0       0.0       0.0
$4C VNTSID   VENTYPE  VENTCOEF   POPPRES  COEFLCID  (STYPE:0=PSID;1=PID;2=SGSID)
$          0         0         0        0.0         0
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|....8
```

*CONSTRAINED_LINEAR_GLOBAL

Purpose: Define linear constraint equations between displacements and rotations, which can be defined in global coordinate systems. For a newer and for a more general constraint see *CONSTRAINED_INTERPOLATION

Card 1 - Required

Card 1	1	2	3	4	5	6	7	8
Variable	LCID							
Type	I							
Default	none							

Card 2 - Define one card for each constrained degree-of-freedom. Input is terminated when a "*" card is found.

Card 2	1	2	3	4	5	6	7	8
Variable	NID	DOF	COEF					
Type	I	I	I					
Default	none	0	0					
Remark	1							

VARIABLE	DESCRIPTION
LCID	Linear constraint definition ID. This ID can be used to identify a set to which this constraint is a member.
NID	Node ID
DOF	Degree of freedom in the global coordinate system; EQ.1: displacement along global x-direction EQ.2: displacement along global y-direction EQ.3: displacement along global z-direction EQ.4: global rotation about global x-axis

VARIABLE	DESCRIPTION
	EQ.5: global rotation about global y-axis EQ.6: global rotation about global z-axis
COEF	Nonzero coefficient, C_k

Remarks:

Nodes of a nodal constraint equation cannot be members of another constraint equation or constraint set that constrain the same degrees-of-freedom, a tied interface, or a rigid body; i.e. nodes cannot be subjected to multiple, independent, and possibly conflicting constraints. Also care must be taken to ensure that single point constraints applied to nodes in a constraint equation do not conflict with the constraint sets constrained degrees-of-freedom.

In this section linear constraint equations of the form:

$$\sum_{k=1}^{n} C_k u_k = C_0$$

can be defined, where u_k are the displacements and C_k are user defined coefficients. Unless LS-DYNA is initialized by linking to an implicit code to satisfy this equation at the beginning of the calculation, the constant C_0 is assumed to be zero. The first constrained degree-of-freedom is eliminated from the equations-of-motion:

$$u_1 = C_0 - \sum_{k=2}^{n} \frac{C_k}{C_1} u_k$$

Its velocities and accelerations are given by

$$\dot{u}_1 = -\sum_{k=2}^{n} \frac{C_k}{C_1} \dot{u}_k$$

$$\ddot{u}_1 = -\sum_{k=2}^{n} \frac{C_k}{C_1} \ddot{u}_k ,$$

respectively. In the implementation a transformation matrix, $\underset{\sim}{L}$, is constructed relating the unconstrained, $\underset{\sim}{u}$, and constrained, $\underset{\sim}{u}_c$, degrees-of-freedom. The constrained accelerations used in the above equation are given by:

$$\ddot{\underset{\sim}{u}}_c = \left[\underset{\sim}{L}^t \underset{\sim}{M} \underset{\sim}{L} \right]^{-1} \underset{\sim}{L}^t \underset{\sim}{F}$$

where $\underset{\sim}{M}$ is the Diagonal lumped mass matrix and $\underset{\sim}{F}$ is the right hand side force vector. This requires the inversion of the condensed mass matrix which is equal in size to the number of constrained degrees-of-freedom minus one.

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$  *CONSTRAINED_LINEAR_GLOBAL
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Constrain nodes 40 and 42 to move identically in the z-direction.
$
$   When the linear constraint equation is applied, it goes like this:
$
$      0 = C40uz40 + C42uz42
$
$        = uz40 - uz42
$
$      uz40 = uz42
$
$    where,
$      C40  =  1.00  coefficient for node 40
$      C42  = -1.00  coefficient for node 42
$      uz40 = displacement of node 40 in z-direction
$      uz42 = displacement of node 42 in z-direction
$
$
*CONSTRAINED_LINEAR
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>...
.8
$         i
$        id
          2
$
$       nid       dof      coef
          40         3      1.00
          42         3     -1.00
$
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

***CONSTRAINED_LINEAR_LOCAL**

Purpose: Define linear constraint equations between displacements and rotations, which can be defined in a local coordinate system. Each node may have a unique coordinate ID.

Card 1 - Required

Card 1	1	2	3	4	5	6	7	8
Variable	LCID							
Type	I							
Default	none							

Card 2 - Define one card for each constrained degree-of-freedom. Input is terminated when a "*" card is found.

Card 2	1	2	3	4	5	6	7	8
Variable	NID	DOF	CID	COEF				
Type	I	I	I	I				
Default	none	0	0	0				
Remark	1							

VARIABLE	DESCRIPTION
LCID	LCID for linear constraint definition. This ID can be used to identify a set to which this constraint is a member.
NID	Node ID
DOF	Degree of freedom in the local coordinate system; EQ.1: displacement along local x-direction EQ.2: displacement along local y-direction EQ.3: displacement along local z-direction EQ.4: local rotation about local x-axis EQ.5: local rotation about local y-axis EQ.6: local rotation about local z-axis

VARIABLE	DESCRIPTION
CID	Local coordinate system ID number. If the number is zero, the global coordinate system is used.
COEF	Nonzero coefficient, C_k

Remarks:

In this section linear constraint equations of the form:

$$\sum_{k=1}^{n} C_k u_k^L = C_0$$

can be defined, where u_k^L are the displacements in the local coordinate systems and C_k are user defined coefficients. Unless LS-DYNA is initialized by linking to an implicit code to satisfy this equation at the beginning of the calculation, the constant C_0 is assumed to be zero. The first constrained degree-of-freedom is eliminated from the equations-of-motion:

$$u_1^L = C_0 - \sum_{k=2}^{n} \frac{C_k}{C_1} u_k^L$$

Its velocities and accelerations are given by

$$\dot{u}_1^L = -\sum_{k=2}^{n} \frac{C_k}{C_1} \dot{u}_k^L$$

$$\ddot{u}_1^L = -\sum_{k=2}^{n} \frac{C_k}{C_1} \ddot{u}_k^L$$

respectively. The local displacements are calculated every time step using the local coordinate systems defined by the user. More than one degree of freedom for a node can be constrained by specifying a card for each degree of freedom.

Nodes of a nodal constraint equation cannot be members of another constraint equation or constraint set that constrain the same degrees-of-freedom, a tied interface, or a rigid body; i.e. nodes cannot be subjected to multiple, independent, and possibly conflicting constraints. Also care must be taken to ensure that single point constraints applied to nodes in a constraint equation do not conflict with the constraint sets constrained degrees-of-freedom.

***CONSTRAINED_LOCAL**

Purpose: Define a boundary constraint in a local coordinate system.

Card	1	2	3	4	5	6	7	8
Variable	TC	RC	CIR	X	Y	Z	CID	
Type	1	1	1	F	F	F	1	
Default	0	0	0	0	0	0	none	

VARIABLE	DESCRIPTION
TC	Translational Constraint: EQ.1: constrained x translation, EQ.2: constrained y translation, EQ.3: constrained z translation, EQ.4: constrained x and y translations, EQ.5: constrained y and z translations, EQ.6: constrained x and z translations, EQ.7: constrained x, y, and translations.
RC	Rotational Constraint: EQ.1: constrained x-rotation, EQ.2: constrained y-rotation, EQ.3: constrained z-rotation, EQ.4: constrained x and y rotations, EQ.5: constrained y and z rotations, EQ.6: constrained z and x rotations, EQ.7: constrained x, y, and z rotations.
DIR	Direction of normal EQ.1: local x, EQ.2: local y, EQ.3: local z.
X	x-offset coordinate
Y	y-offset coordinate
Z	z-offset coordinate
CID	Coordinate ID defining the orientation of the wall.

Remarks:

Nodes within a mesh-size-dependent tolerance are constrained on a local plane. This option is recommended for use with r-method adaptive remeshing where nodal constraints are lost during the remeshing phase.

***CONSTRAINED_NODAL_RIGID_BODY_{*OPTION*}_{*OPTION*}_{*OPTION*}**

Available options include:

<BLANK>

SPC

INERTIA

TITLE

If the center of mass is constrained use the **SPC** option. If the inertial properties are defined rather than computed use the **INERTIA** option. A description for the nodal rigid body can be defined with the **TITLE** option.

Purpose: Define a nodal rigid body. This is a rigid body which consists of the defined nodes. If the INERTIA option is not used, then the inertia tensor is computed from the nodal masses. Arbitrary motion of this rigid body is allowed. If the INERTIA option is used, constant translational and rotational velocities can be defined in a global or local coordinate system.

The first node in the nodal rigid body definition is treated as the master for the case where DRFLAG and RRFLAG are nonzero. The first node always has six degrees-of-freedom. The release conditions applied in the global system are sometimes convenient in small displacement linear analysis, but, otherwise, are not recommended. It is strongly recommended, especially for implicit calculations, that release conditions are only used for a two noded nodal rigid body.

Card Format:

Card 1 is required.
Cards 2 - 4 are required for the INERTIA option.
Card 5 is required if a local coordinate system is used to specify the inertia tensor when the INERTIA option is used.

Remarks:

1. Unlike the *CONSTRAINED_NODE_SET which permits only constraints on translational motion, here the equations of rigid body dynamics are used to update the motion of the nodes and therefore rotations of the nodal sets are admissible. Mass properties are determined from the nodal masses and coordinates. Inertial properties are defined if and only if the INERTIA option is specified.

The following card is read if and only if the TITLE option is specified.

Optional

Variable	TITLE
Type	A80

Card 1	1	2	3	4	5	6	7	8
Variable	PID	CID	NSID	PNODE	IPRT	DRFLAG	RRFLAG	
Type	I	I	I	I	I	I	I	
Default	none	none	none	0	0	0	0	

Define if and only if SPC is specified in the keyword.

Card 2	1	2	3	4	5	6	7	8
Variable	CMO	CON1	CON2					
Type	F	F	F					
Default	0	0	0					

VARIABLE	DESCRIPTION
PID	Part ID of the nodal rigid body.
CID	Optional coordinate system ID for the rigid body local system, see *DEFINE_COORDINATE_OPTION. Output of the rigid body data and the degree-of- freedom releases are done in this local system. This local system rotates with the rigid body.
NSID	Nodal set ID, see *SET_NODE_OPTION. This nodal set defines the rigid body. If NSID=0, then NSID=PID, i.e., the node set ID and the part ID are assumed to be identical.
PNODE	An optional, possibly massless, nodal point located at the mass center of the nodal rigid body. The initial nodal coordinates will be reset if necessary to ensure that they lie at the mass center. In the output files, the coordinates, accelerations, velocities, and displacements of this node will correspond to the mass center of the nodal rigid body. If CID is defined, the velocities and accelerations of PNODE will be output in the local system in the D3PLOT and D3THDT files unless PNODE is specified as a negative number in which case the global system is used.

VARIABLE	DESCRIPTION
IPRT	Print flag. For nodal rigid bodies the following values apply: EQ.1: write data into RBDOUT EQ.2: do not write data into RBDOUT Printing is suppressed for two noded rigid bodies unless IPRT is set to unity. This is to avoid excessively large RBDOUT files when many, two-noded welds are used.
DRFLAG	Displacement release flag for all nodes except the first node in the definition. EQ.-7: release x, y, and z displacement in global system EQ.-6: release z and x displacement in global system EQ.-5: release y and z displacement in global system EQ.-4: release x and y displacement in global system EQ.-3: release z displacement in global system EQ.-2: release y displacement in global system EQ.-1: release x displacement in global system EQ. 0: off for rigid body behavior EQ. 1: release x displacement in rigid body local system EQ. 2: release y displacement in rigid body local system EQ. 3: release z displacement in rigid body local system EQ. 4: release x and y displacement in rigid body local system EQ. 5: release y and z displacement in rigid body local system EQ. 6: release z and x displacement in rigid body local system EQ. 7: release x, y, and z displacement in rigid body local system
RRFLAG	Rotation release flag for all nodes except the first node in the definition. EQ.-7: release x, y, and z rotations in global system EQ.-6: release z and x rotations in global system EQ.-5: release y and z rotations in global system EQ.-4: release x and y rotations in global system EQ.-3: release z rotation in global system EQ.-2: release y rotation in global system EQ.-1: release x rotation in global system EQ. 0: off for rigid body behavior EQ. 1: release x rotation in rigid body local system EQ. 2: release y rotation in rigid body local system EQ. 3: release z rotation in rigid body local system EQ. 4: release x and y rotations in rigid body local system EQ. 5: release y and z rotations in rigid body local system EQ. 6: release z and x rotations in rigid body local system EQ. 7: release x, y, and z rotations in rigid body local system
CMO	Center of mass constraint option, CMO: EQ.+1.0: constraints applied in global directions, EQ.0.0: no constraints, EQ.-1.0: constraints applied in local directions (SPC constraint).

VARIABLE	DESCRIPTION
CON1	First constraint parameter: If CMO=+1.0, then specify global translational constraint: EQ.0: no constraints, EQ.1: constrained x displacement, EQ.2: constrained y displacement, EQ.3: constrained z displacement, EQ.4: constrained x and y displacements, EQ.5: constrained y and z displacements, EQ.6: constrained z and x displacements, EQ.7: constrained x, y, and z displacements. If CM0=-1.0, then specify local coordinate system ID. See *DEFINE_COORDINATE_*OPTION*: This coordinate system is fixed in time.
CON2	Second constraint parameter: If CMO=+1.0, then specify global rotational constraint: EQ.0: no constraints, EQ.1: constrained x rotation, EQ.2: constrained y rotation, EQ.3: constrained z rotation, EQ.4: constrained x and y rotations, EQ.5: constrained y and z rotations, EQ.6: constrained z and x rotations, EQ.7: constrained x, y, and z rotations. If CM0=-1.0, then specify local (SPC) constraint: EQ.000000 no constraint, EQ.100000 constrained x translation, EQ.010000 constrained y translation, EQ.001000 constrained z translation, EQ.000100 constrained x rotation, EQ.000010 constrained y rotation, EQ.000001 constrained z rotation. Any combination of local constraints can be achieved by adding the number 1 into the corresponding column.

Required for the INERTIA option.

Card 2

Variable	XC	YC	ZC	TM	IRCS	NODEID		
Type	F	F	F	F	I	I		
Default	0	0	0	0	0	0		

VARIABLE	DESCRIPTION
XC	x-coordinate of center of mass. If nodal point, NODEID, is defined, XC, YC, and ZC are ignored and the coordinates of the nodal point, NODEID, are taken as the center of mass.
YC	y-coordinate of center of mass
ZC	z-coordinate of center of mass
TM	Translational mass
IRCS	Flag for inertia tensor reference coordinate system: EQ.0: global inertia tensor, EQ.1: principal moments of inertias with orientation vectors as given below.
NODEID	Optional nodal point defining the CG of the rigid body. If this node is not a member of the set NSID above, its motion will not be updated to correspond with the nodal rigid body after the calculation begins. PNODE and NODEID can be identical if and only if PNODE physically lies at the mass center at time zero.

Required for the INERTIA option.

Card 3	1	2	3	4	5	6	7	8
Variable	IXX	IXY	IXZ	IYY	IYZ	IZZ		
Type	F	F	F	F	F	F		
Default	none	0	0	none	0	0		

VARIABLE	DESCRIPTION
IXX	I_{xx}, xx component of inertia tensor
IXY	I_{xy} (set to zero if IRCS=1)
IXZ	I_{xz} (set to zero if IRCS=1)
IYY	I_{yy}, yy component of inertia tensor
IYZ	I_{yz} (set to zero if IRCS=1)
IZZ	I_{ZZ}, zz component of inertia tensor

Required for the INERTIA option.

Card 4	1	2	3	4	5	6	7	8
Variable	VTX	VTY	VTZ	VRX	VRY	VRZ		
Type	F	F	F	F	F	F		
Default	0	0	0	0	0	0		

VARIABLE	DESCRIPTION
VTX	x-rigid body initial translational velocity in global coordinate system.
VTY	y-rigid body initial translational velocity in global coordinate system.
VTZ	z-rigid body initial translational velocity in global coordinate system.
VRX	x-rigid body initial rotational velocity in global coordinate system.
VRY	y-rigid body initial rotational velocity in global coordinate system.
VRZ	z-rigid body initial rotational velocity in global coordinate system.

Remarks:

The velocities defined above can be overwritten by the *INITIAL_VELOCITY card.

Optional card required for IRCS=1. Define two local vectors or a local coordinate system ID.

Card 5	1	2	3	4	5	6	7	8
Variable	XL	YL	ZL	XLIP	YLIP	ZLIP	CID2	
Type	F	F	F	F	F	F	I	
Default	none	none	none	none	none	none	none	

VARIABLE	DESCRIPTION
XL	x-coordinate of local x-axis. Origin lies at (0,0,0).
YL	y-coordinate of local x-axis
ZL	z-coordinate of local x-axis
XLIP	x-coordinate of local in-plane vector
YLIP	y-coordinate of local in-plane vector
ZLIP	z-coordinate of local in-plane vector
CID2	Local coordinate system ID, see *DEFINE_COORDINATE_.... With this option leave fields 1-6 blank.

Remarks:

The local coordinate system is set up in the following way. After the local x-axis is defined, the local z-axis is computed from the cross-product of the local x-axis vector with the given in-plane vector. Finally, the local y-axis is determined from the cross-product of the local z-axis with the local x-axis. The local coordinate system defined by CID has the advantage that the local system can be defined by nodes in the rigid body which makes repositioning of the rigid body in a preprocessor much easier since the local system moves with the nodal points.

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$  *CONSTRAINED_NODAL_RIGID_BODY
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Define a rigid body consisting of the nodes in nodal set 61.
$
$  This particular example was used to connect three separate deformable
$  parts.  Physically, these parts were welded together.  Modeling wise,
$  however, this joint is quit messy and is most conveniently modeled
$  by making a rigid body using several of the nodes in the area.  Physically,
$  this joint was so strong that weld failure was never of concern.
$
*CONSTRAINED_NODAL_RIGID_BODY
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$      pid       cid      nsid
        45                  61
$
$     nsid = 61   nodal set ID number, requires a *SET_NODE_option
$      cid        not used in this example, output will be in global coordinates
$
$
*SET_NODE_LIST
$      sid
        61
$     nid1      nid2      nid3      nid4      nid5      nid6      nid7      nid8
        823      1057      1174      1931      2124      1961      2101
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

***CONSTRAINED_NODE_SET_{*OPTION*}**

To define an ID for the constrained node set the following option is available:

<BLANK>

ID

If the ID is defined an additional card is required.

Purpose: Define nodal constraint sets for translational motion in global coordinates. No rotational coupling. See Figure 6.19. Nodal points included in the sets should not be subjected to any other constraints including prescribed motion, e.g., with the *BOUNDARY_ PRESCRIBED_MOTION options.

ID Card - Required if the option ID is active on the keyword card.

Card 1	1	2	3	4	5	6	7	8
Variable	CNSID							
Type	I							
Default	0							

Card	1	2	3	4	5	6	7	8
Variable	NSID	DOF	TF					
Type	I	I	F					
Default	none	none	1.E+20					
Remarks	1		2					

VARIABLE	DESCRIPTION
CNSID	Optional constrained node set ID.
NSID	Nodal set ID, see *SET_NODE_*OPTION*.

VARIABLE	DESCRIPTION
DOF	Applicable degrees-of-freedom: EQ.1: x-translational degree-of-freedom, EQ.2: y-translational degree-of-freedom, EQ.3: z-translational degree-of-freedom, EQ.4: x and y-translational degrees-of-freedom, EQ.5: y and z-translational degrees-of-freedom, EQ.6: z and x-translational degrees-of-freedom, EQ.7: x, y, and z-translational degrees-of-freedom.
TF	Failure time for nodal constraint set.

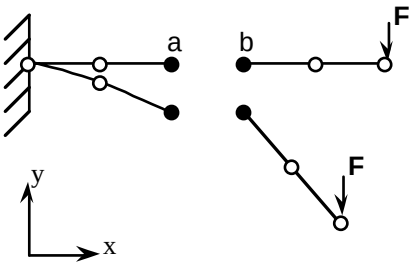
Remarks:

1. The masses of the nodes are summed up to determine the total mass of the constrained set. It must be noted that the definition of a nodal rigid body is not possible with this input. For nodal rigid bodies the keyword input: *CONSTRAINED_NODAL_RIGID_BODY_ *OPTION*, must be used.

2. When the failure time, *TF*, is reached the nodal constraint becomes inactive and the constrained nodes may move freely.

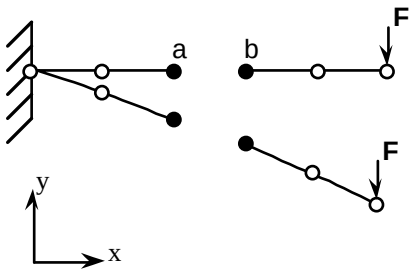
***CONSTRAINED_NODE_SET**

Since no rotation is permitted, this option should not be used to model rigid body behavior that involves rotations.



***CONSTRAINED_NODAL_RIGID_BODY**
***CONSTRAINED_SPOTWELD**

Behavior is like a rigid beam. These options may be used to model spotwelds.



Offset nodes a and b are constrained to move together.

Figure 6.19. *CONSTRAINED_NODE_SET can lead to nonphysical responses.

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$   *CONSTRAINED_NODE_SET
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Constrain all the nodes in a nodal set to move equivalently
$  in the z-direction.
$
*CONSTRAINED_NODE_SET
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
      nsid       dof        tf
         7         3      10.0
$
$     nsid = 7    nodal set ID number, requires a *SET_NODE_option
$      dof = 3    nodal motions are equivalent in z-translation
$      tf  = 3    at time=10. the nodal constraint is removed
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

*CONSTRAINED_POINTS

Purpose: Constrain two points with the specified coordinates connecting two shell elements at locations other than nodal points. In this option, the penalty method is used to constrain the translational and rotational degrees-of-freedom of the points. Force resultants are written into the SWFORC ASCII file for post-processing.

Card Format (I10)

Card 1

1	2	3	4	5	6	7	8
CID							
I							
none							

Card Format (I8,3E16.0)

Card 2

1	2	3	4	5	6	7	8	9	10
EID1	X1		Y1		Z1				
I	F		F		F				
none	0.		0.		0.				

Card 3

1	2	3	4	5	6	7	8	9	10
EID2	X2		Y2		Z2				
I	F		F		F				
none	0.		0.		0.				

Card Format (4E10.0)

Card 4	1	2	3	4	5	6	7	8
Variable	PSF	FAILA	FAILS	FAILM				
Type	F	F	F	F				
Default	1.0	0.0	0.0	0.0				

VARIABLE	DESCRIPTION
CID	Constrained points ID.
Xi, Yi, Zi	Coordinates of the constrained points, i=1,2.
EIDi	Shell element ID, i=1,2.
PSF	Penalty scale factor (Default=1.0).
FAILA	Axial force resultant failure value (Skip if zero).
FAILS	Shear force resultant failure value (Skip if zero).
FAILM	Moment resultant failure value (Skip if zero).

*CONSTRAINED_RIGID_BODIES

Purpose: Merge two rigid bodies. One rigid body, called slave rigid body, is merged to the other one called a master rigid body.

Card	1	2	3	4	5	6	7	8
Variable	PIDM	PIDS	IFLAG					
Type	I	I	I					
Default	none	none	0					

VARIABLE	DESCRIPTION
PIDM	Master rigid body part ID, see *PART.
PIDS	Slave rigid body part ID, see *PART.
IFLAG	This flag is meaningful *if and only if* the inertia properties of the Part ID, PIDM, are defined in PART_INERTIA. If set to unity, the center-of-gravity, the translational mass, and the inertia matrix of PIDM, will be updated to reflect the merging of rigid body PIDS. If IFLAG is defaulted to zero, the merged PIDS will not affect the properties defined in PART_INERTIA for PIDM since it is assumed the properties already account for merged parts. The inertia properties of PIDS will be computed from its nodal masses if the properties are not defined in a PART_INERTIA definition.

Remarks:

The slave rigid body is merged to the master rigid body. The inertial properties computed by LS-DYNA are based on the combination of the master rigid body plus all the rigid bodies which are slaved to it unless the inertial properties of the master rigid body are defined via the *PART_INERTIA keyword in which case those properties are used for the combination of the master and slave rigid bodies. Note that a master rigid body may have many slaves.

Independent rigid bodies must not share common nodes since each rigid body updates the motion of its nodes independently of the other rigid bodies. If common nodes exist between rigid bodies the rigid bodies sharing the nodes must be merged.

It is also possible to merge rigid bodies that are completely separated and share no common nodal points or boundaries. All actions valid for the master rigid body, e.g., constraints, given velocity, are now also valid for the newly-created rigid body.

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$  *CONSTRAINED_RIGID_BODIES
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Rigidly connect parts 35, 70, 71, and 72 to part 12.
$  All parts must be defined as rigid.
$
$  This example is used to make a single rigid body out of the five parts
$  that compose the back end of a vehicle.  This was done to save cpu time
$  and was determined to be valid because the application was a frontal
$  impact with insignificant rear end deformations.  (The cpu time saved
$  was from making the parts rigid, not from merging them - merging was
$  more of a convenience in this case for post processing, for checking
$  inertial properties, and for joining the parts.)
$
*CONSTRAINED_RIGID_BODIES
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$     pidm      pids
        12        35
        12        70
        12        71
        12        72
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

*CONSTRAINED_RIGID_BODY_STOPPERS

Purpose: Rigid body stoppers provide a convenient way of controlling the motion of rigid tooling in metalforming applications. The motion of a "master" rigid body is limited by load curves. This option will stop the motion based on a time dependent constraint. The stopper overrides prescribed motion boundary conditions (except relative displacement) operating in the same direction for both the master and slaved rigid bodies. See Figure 6.20.

Card 1	1	2	3	4	5	6	7	8
Variable	PID	LCMAX	LCMIN	PSIDMX	PSIDMN	LCVMNX	DIR	VID
Type	I	I	I	I	I	I	I	I
Default	required	0	0	0	0	0	required	0

Card 2

Variable	TB	TD						
Type	F	F						
Default	0	10^{21}						

VARIABLE	DESCRIPTION
PID	Part ID of master rigid body, see *PART.
LCMAX	Load curve ID defining the maximum coordinate or displacement as a function of time. See *DEFINE_CURVE: LT.0: Load Curve ID |LCMAX| provides an upper bound for the displacement of the rigid body EQ.0: no limitation of the maximum displacement. GT 0: Load Curve ID LCMAX provides an upper bound for the position of the rigid body center of mass

VARIABLE	DESCRIPTION
LCMIN	Load curve ID defining the minimum coordinate or displacement as a function of time. See *DEFINE_CURVE: LT.0: Load Curve ID |LCMIN| defines a lower bound for the displacement of the rigid body EQ.0: no limitation of the minimum displacement. GT.0: Load Curve ID LCMIN defines a lower bound for the position of the rigid body center of mass
PSIDMX	Optional part set ID of rigid bodies that are slaved in the maximum coordinate direction to the master rigid body. The part set definition, (see *SET_PART_COLUMN) may be used to define the closure distance (D_1 and D_2 in Figure 6.20) which activates the constraint. The constraint does not begin to act until the master rigid body stops. If the distance between the master rigid body is greater than or equal to the closure distance, the slave rigid body motion away from the master rigid body also stops. However, the slaved rigid body is free to move towards the master. If the closure distance is input as zero (0.0) then the slaved rigid body stops when the master stops.
PSIDMN	Optional part set ID of rigid bodies that are slaved in the minimum coordinate direction to the master rigid body. The part set definition, (see *SET_PART_COLUMN) may be used to define the closure distance (D_1 and D_2 in Figure 6.20) which activates the constraint. The constraint does not begin to act until the master rigid body stops. If the distance between the master rigid body is less than or equal to the closure distance, the slave rigid body motion towards the master rigid body also stops. However, the slaved rigid body is free to move away from the master. If the closure distance is input as zero (0.0) then the slaved rigid body stops when the master stops.
LCVMX	Load curve ID which defines the maximum absolute value of the velocity as a function of time that is allowed for the master rigid body. See *DEFINE_CURVE: EQ.0: no limitation on the velocity.
DIR	Direction stopper acts in: EQ.1: x-translation, EQ.2: y-translation, EQ.3: z-translation, EQ.4: arbitrary, defined by vector VID (see below), EQ.5: x-axis rotation, EQ.6: y-axis rotation, EQ.7: z-axis rotation, EQ.8: arbitrary, defined by vector VID (see below).

VARIABLE	DESCRIPTION
VID	Vector for arbitrary orientation of stopper, see *DEFINE_VECTOR.
TB	Time at which stopper is activated.
TD	Time at which stopper is deactivated.

Remarks:

The optional definition of part sets in minimum or maximum coordinate direction allows the motion to be controlled in arbitrary direction.

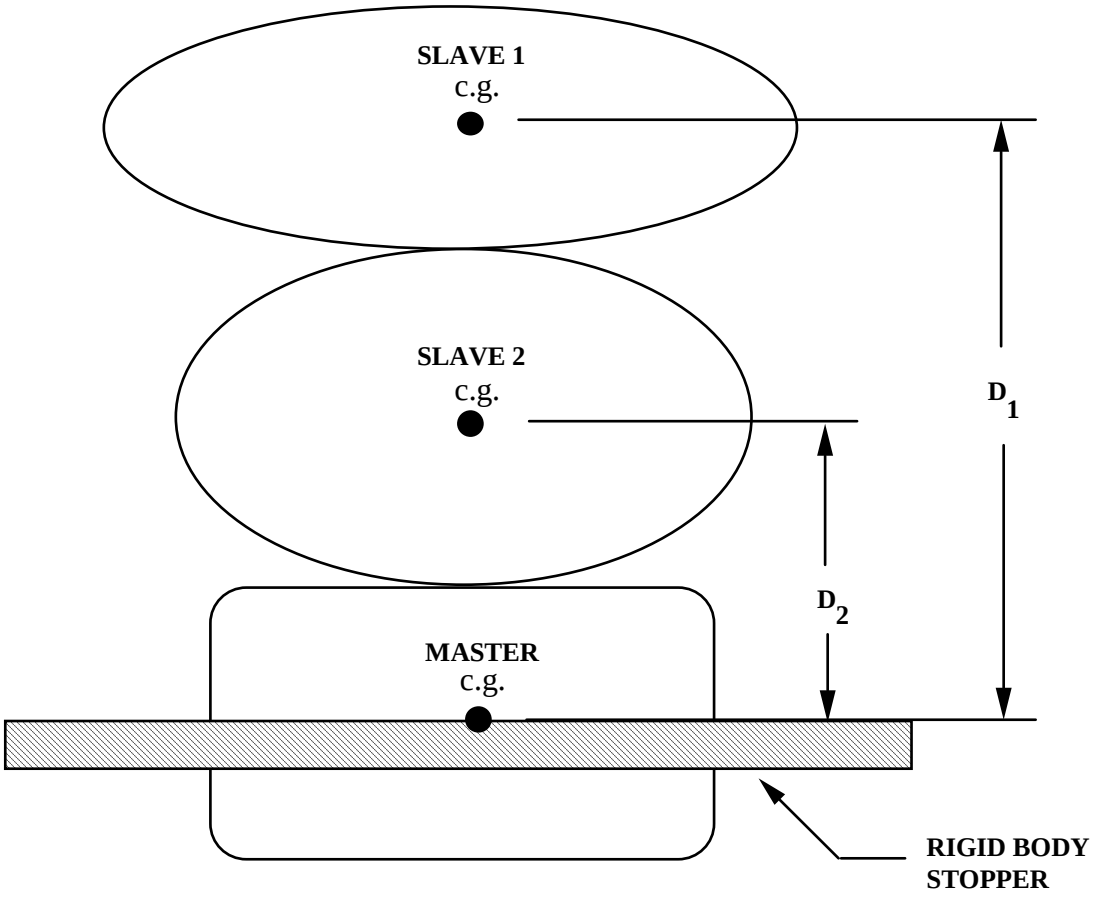


Figure 6.20 When the master rigid body reaches the rigid body stopper, the velocity component into the stopper is set to zero. Slave rigid bodies 1 and 2 also stop if the distance between their mass centers and the master rigid body is less than or equal to the input values D_1 and D_2, respectively. (c.g. + center of gravity).

***CONSTRAINED_RIVET**_{*OPTION*}

To define an ID for the rivet, the following option is available:

ID

If the ID is defined an additional card is required.

Purpose: Define massless rivets between non-contiguous nodal pairs. The nodes must not have the same coordinates. The action is such that the distance between the two nodes is kept constant throughout any motion. No failure can be specified.

ID Card - Required if the option ID is active on the keyword card.

Card 1

Variable	RID							
Type	I							
Default	0							

Card 1

Variable	N1	N2	TF					
Type	I	I	F					
Default	none	none	1.E+20					
Remarks	1		2					

VARIABLE	DESCRIPTION
RID	Optional rivet ID.
N1	Node ID
N2	Node ID
TF	Failure time for nodal constraint set.

Remarks:

1. Nodes connected by a rivet cannot be members of another constraint set that constrain the same degrees-of-freedom, a tied interface, or a rigid body, i.e., nodes cannot be subjected to multiple, independent, and possibly conflicting constraints. Also care must be taken to ensure that single point constraints applied to nodes in a constraint set do not conflict with the constraint sets constrained degrees-of-freedom.

2. When the failure time, *TF*, is reached the rivet becomes inactive and the constrained nodes may move freely.

```
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$   *CONSTRAINED_RIVET
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Connect node 382 to node 88471 with a massless rivet.
$
*CONSTRAINED_RIVET
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$       n1        n2        tf
       382     88471       0.0
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

***CONSTRAINED_SHELL_TO_SOLID**

Purpose: Define a tie between a shell edge and solid elements. Nodal rigid bodies can perform the same function and may also be used.

Card	1	2	3	4	5	6	7	8
Variable	NID	NSID						
Type	I	I						
Default	none	none						
Remarks								

VARIABLE	DESCRIPTION
NID	Shell node ID
NSID	Solid nodal set ID, see *SET_NODE_*OPTION*.

Remarks:

The shell-brick interface, an extension of the tied surface capability, ties regions of hexahedron elements to regions of shell elements. A shell node may be tied to up to nine brick nodes lying along the tangent vector to the nodal fiber. See Figure 6.21. During the calculation, the brick nodes thus constrained, must lie along the fiber but can move relative to each other in the fiber direction. The shell node stays on the fiber at the same relative spacing between the first and last brick node. The brick nodes must be input in the order in which they occur, in either the plus or minus direction, as one moves along the shell node fiber.

This feature is intended to tie four node shells to eight node shells or solids; it is not intended for tying eight node shells to eight node solids.

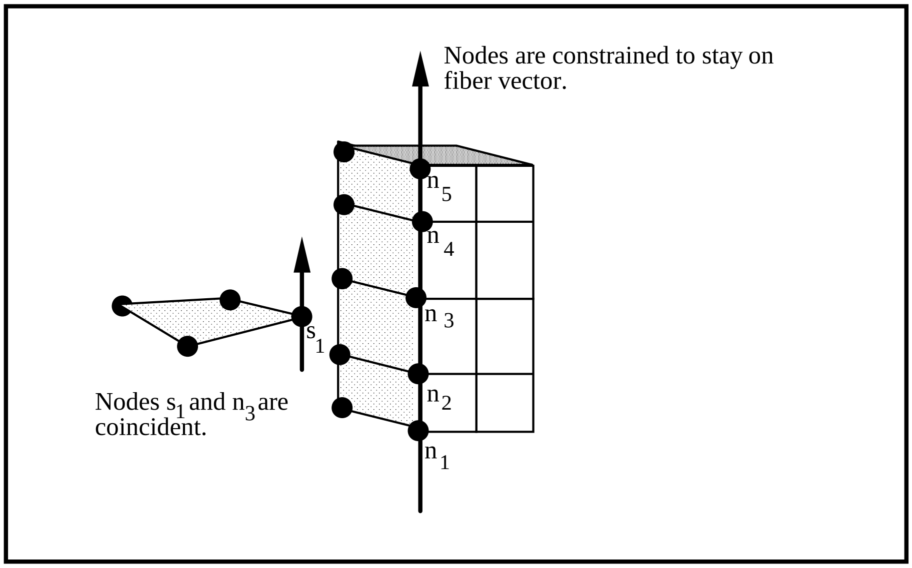


Figure 6.21 The interface between shell elements and solids ties shell node s1 to a line of nodes on the solid elements n1-n5. It is very important for the nodes to be aligned.

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$  *CONSTRAINED_SHELL_TO_SOLID
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Tie shell element, at node 329, to a solid element at node 203.
$    - nodes 329 and 203 are coincident
$
$  Additionally, define a line of nodes on the solids elements, containing
$  node 203, that must remain in the same direction as the fiber of the shell
$  containing node 329.  In other words:
$
$    - Nodes 119, 161, 203, 245 and 287 are nodes on a solid part that
$       define a line on that solid part.
$    - This line of nodes will be constrained to remain linear throughout
$       the simulation.
$    - The direction of this line will be kept the same as the fiber of the
$       of the shell containing node 329.
$
*CONSTRAINED_SHELL_TO_SOLID
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$      nid      nsid
       329         4
$
*SET_NODE_LIST
$      sid
         4
$     nid1      nid2      nid3      nid4      nid5      nid6      nid7      nid8
       119       161       203       245       287
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
```

***CONSTRAINED_SPLINE**

Purpose: Define an elastic cubic spline interpolation constraint. The displacements and slopes at the end points are continuous. The first and last nodes, which define the constraint, must be independent. The degrees-of-freedom of interior nodes may be either dependent or independent.

Card 1	1	2	3	4	5	6	7	8
Variable	SPLID	DLRATIO						
Type	I	I						
Default	0	0.10						

Cards 2, 3, 4, etc. Define one card per independent/dependent node. The first and last nodes must be independent. The next "*" card terminates this input.

Card 2…	1	2	3	4	5	6	7	8
Variable	NID	DOF						
Type	I	I						
Default	0	0						

VARIABLE	DESCRIPTION
SPLID	Spline constraint ID.
DLRATIO	Ratio of bending to torsional stiffness for an elastic tubular beam which connects the independent degrees-of-freedom. The default value is set to 0.10.
NID	Independent/dependent node ID. For explicit problems this node should not be a member of a rigid body, or elsewhere constrained in the input.

VARIABLE	DESCRIPTION
DOF	Degrees-of-freedom. The list of dependent degrees-of-freedom consists of a number with up to six digits, with each digit representing a degree of freedom. For example, the value 1356 indicates that degrees of freedom 1, 3, 5, and 6 are controlled by the constraint. The default is 123456. Digit: degree of freedom ID's: EQ.1: x EQ.2: y EQ.3: z EQ.4: rotation about x axis EQ.5: rotation about y axis EQ.6: rotation about z axis

*CONSTRAINED_SELF_PIERCING_RIVET

Purpose: Define a self-piercing rivet with failure. This model for a self-piercing rivet (SPR) includes a plastic-like damage model that reduces the force and moment resultants to zero as the rivet fails. The domain of influence is specified by a diameter, which should be approximately equal to the rivet's diameter. The location of the rivet is defined by a single node at the center of two riveted sheets. The algorithm does a normal projection from the master and slave sheets to the rivet node and locates all nodes within the user-defined diameter of influence. The numerical implementation of this rivet model was developed by L. Olovsson of Impetus Afea, based on research work on SPR point connector models originally carried out by SIMLab (NTNU) and SINTEF, see references by Porcaro, Hanssen, and *et.al*. [2006, 2006, 2007].

Card 1

Variable	MID	SID	NSID	H1	H2	D	FN0	FT0
Type	I	I	I	F	F	F	F	F
Default	none	none	none	none	none	none	none	none

Card 2

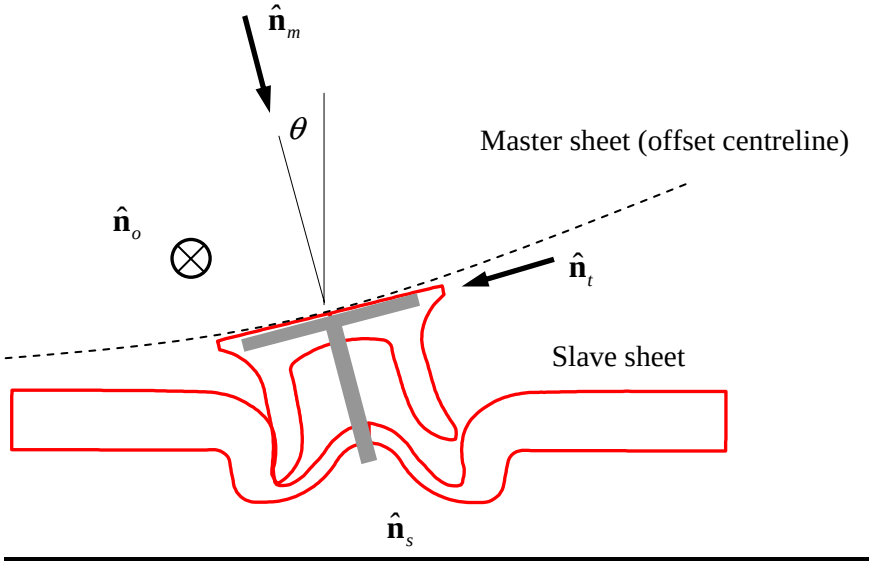
Variable	DFAIL	ALPHA0	ALPHA1	ALPHA2	ALPHA3	DENS		
Type	F	F	F	F	F	F		
Default	none	none	none	none	none	none		
Remarks								

VARIABLE	DESCRIPTION
SPRID	Self-piercing rivet ID.
MID	Master sheet Part ID
SID	Slave sheet Part ID
NSID	Node set ID of rivet location nodes.
H1	Thickness of master sheet.

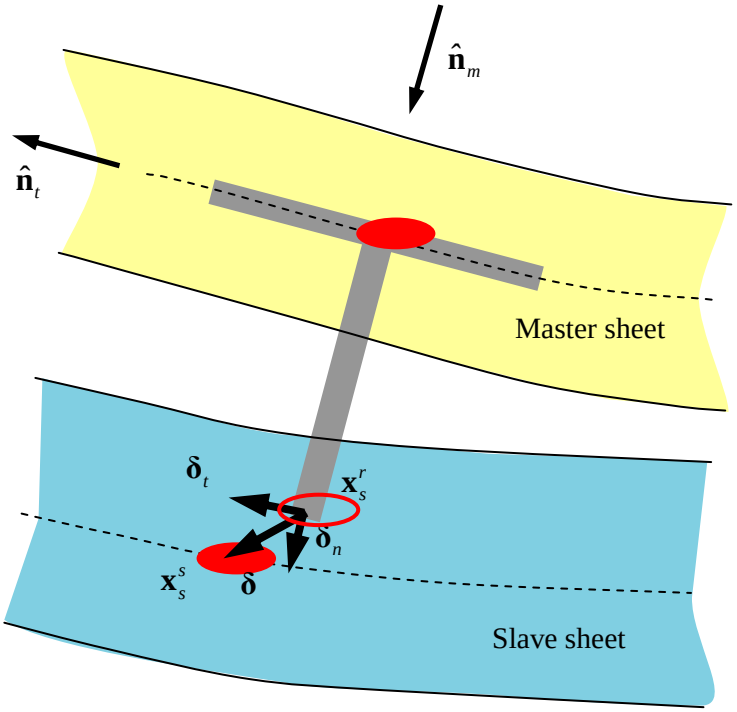
VARIABLE	DESCRIPTION
H2	Thickness of slave sheet.
D	Rivet diameter.
FN0	Maximum pure pull-out force.
FT0	Maximum pure shear force.
DFAIL	Local rivet deformation capacity in pure-pull out..
ALPHA0	Dimensionless yield surface parameter.
ALPHA1	Dimensionless yield surface parameter.
ALPHA2	Dimensionless yield surface parameter.
ALPHA3	Dimensionless yield surface parameter.
DENS	Rivet density (necessary for time step calculation)

Remarks:

Self piercing rivets are a type of fastener that is sometimes used in place of spot welds to join sheet metal of similar or dissimilar materials. The rivet penetrates the first sheet, expands to interlock with the lower sheet without penetration as shown in the figure. The strength and fatigue characteristics of self piercing rivets can meet or even exceed that of spot welds; consequently, their practicle applications are expanding.



The finite element model can approximate the behavior of this rivet including failure where the riveted sheets separate. A yield surface combined with damage ensures that the forces that develop during pullout are adequately approximated.



Referring to the above figure, the following local deformation vectors are used:

Normal stretch:

$$\boldsymbol{\delta}_n$$

Tangential stretch

$$\boldsymbol{\delta}_t$$

Total stretch

$$\boldsymbol{\delta} = \boldsymbol{\delta}_t + \boldsymbol{\delta}_n$$

The scalar measure of the normal stretch is

$$\delta_n = \|\boldsymbol{\delta}_n\|$$

The scalar measure of the tangential stretch

$$\delta_t = \|\boldsymbol{\delta}_t\|$$

So that

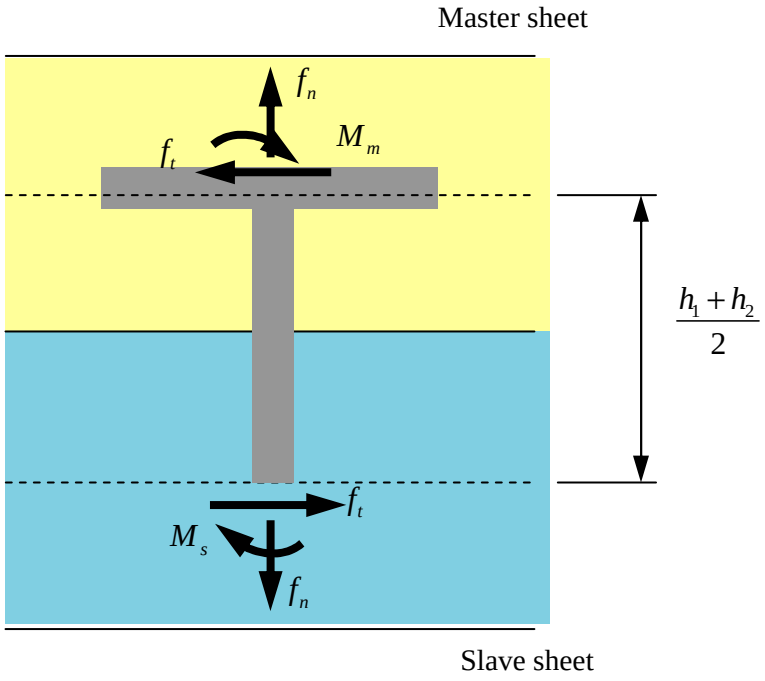
$$\delta = \left\| \boldsymbol{\delta}_t + \boldsymbol{\delta}_n \right\|$$

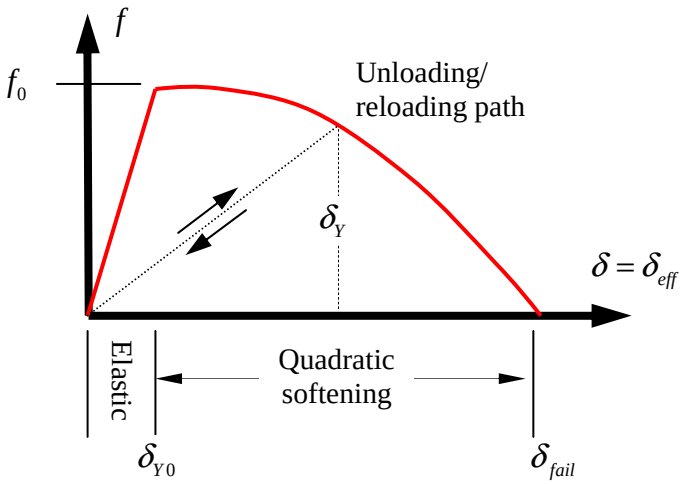
At any given time the total stretch is computed from the position vectors

$$\boldsymbol{\delta} = \mathbf{x}_s^r - \mathbf{x}_s^s$$

so that

$$\boldsymbol{\delta}_n = \boldsymbol{\delta} \bullet \hat{\mathbf{n}}_m \qquad \boldsymbol{\delta}_t = \boldsymbol{\delta} \bullet \hat{\mathbf{n}}_t$$





For general loading where both normal and tangential displacements occur we need a new measure for the effective displacement so that the damage evolution can be determined. We define the following measure for the effective displacement which can be thought of as a yield surface in displacement space:

$$\delta_{eff}(\delta_n,\delta_t,\alpha_{12},\alpha_3)=\left[\xi+\frac{1-\xi}{\alpha_{12}}\right]\sqrt{\delta_n^2+\left(\frac{\delta_t}{\alpha_3}\right)^2}$$

ξ is a parameter ranging between 0 and 1 and it scales the effective displacement as a function of the direction of the displacement vector in the δn- δt plane.

$$\xi = cos^2\, 2\bar{\theta}$$

$$\bar{\theta}=\theta+\alpha_0\theta\left(\frac{\pi}{2}-\theta\right)$$

$$\theta = arctan\left(\frac{\alpha_3\delta_n}{\delta_t}\right)$$

The yield surface is allowed to change in shape as damage develops. This is done by defining the following relationship for the shape coefficient $a12$

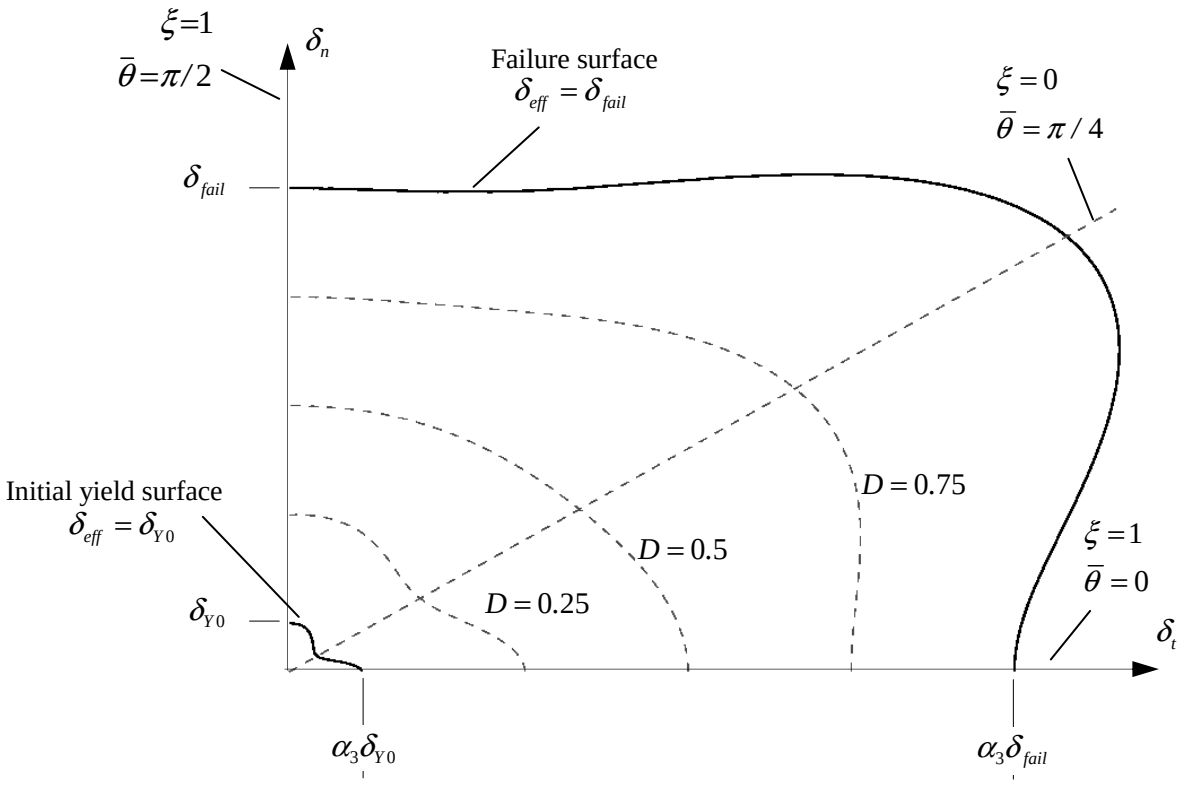
$$\alpha_{12}=(1-D)\alpha_1+D\alpha_2$$

The damage evolution is in terms of effective measures

$$\dot{D}=\begin{cases}\dot{\delta}_{eff}/\delta_{fail} & :\delta_{eff}=\delta_Y\\ 0 & :\delta_{eff}<\delta_Y\end{cases}$$

The relation between damage and maximum effective displacement is

$$\delta_Y=(0.1+0.9D)\,\delta_{fail}$$


ξ=1
$\bar{\theta}=\pi/2$
δ_n
Failure surface
$\delta_{eff}=\delta_{fail}$
$\xi=0$
$\bar{\theta}=\pi/4$
δ_{fail}
$D=0.75$
Initial yield surface
$\delta_{eff}=\delta_{Y0}$
$D=0.5$
$\xi=1$
$\bar{\theta}=0$
δ_{Y0}
$D=0.25$
δ_t
$\alpha_3\delta_{Y0}$
$\alpha_3\delta_{fail}$

***CONSTRAINED_SPOTWELD_{*OPTION*}_{*OPTION*}**

If it is desired to use a time filtered force calculation for the forced based failure criterion then the following option is available:

FILTERED_FORCE

and one additional card must be defined below. To define an ID for the spotweld the following option is available:

ID

If the ID is defined an additional card is required. The ordering of the options is arbitrary.

Purpose: Define massless spot welds between non-contiguous nodal pairs. The spot weld is a rigid beam that connects the nodal points of the nodal pairs; thus, nodal rotations and displacements are coupled. The spot welds must be connected to nodes having rotary inertias, i.e., beams or shells. If this is not the case, for example, if the nodes belong to solid elements, use the option: *CONSTRAINED_RIVET. For Implicit, this case is treated like a rivet, constraining only the displacements. Note that shell elements do not have rotary stiffness in the normal direction and, therefore, this component cannot be transmitted. Spot welded nodes must not have the same coordinates. Coincident nodes in a spot weld can be handled by the *CONSTRAINED_NODAL_RIGID_BODY option. Brittle and ductile failures can be specified. Brittle failure is based on the resultant forces acting on the weld, and ductile failure is based on the average plastic strain value of the shell elements which include the spot welded node. Spot welds, which are connected to massless nodes, are automatically deleted in the initialization phase and a warning message is printed in the MESSAG file and the D3HSP file.

Warning: The accelerations of spot welded nodes are output as zero into the various databases, but if the acceleration of spotwelded nodes are required, use either the *CONSTRAINED_ GENERALIZED_WELD or the *CONSTRAINED_NODAL_RIGID_BODY input. However, if the output interval is frequent enough accurate acceleration time histories can be obtained from the velocity time history by differentiation in the post-processing phase.

ID Card - Required if the option ID is active on the keyword card.

Card 1	1	2	3	4	5	6	7	8
Variable	WID							
Type	I							
Default	0							

Card 1	1	2	3	4	5	6	7	8
Variable	N1	N2	SN	SS	N	M	TF	EP
Type	I	I	F	F	F	F	F	F
Default	none	none	optional	optional	none	none	1.E+20	1.E+20
Remarks	1.		2.				3	4

Define if and only if the option FILTERED_FORCE is specified.

Card 2	1	2	3	4	5	6	7	8
Variable	NF	TW						
Type	I	F						
Default	none	none						
Remarks								

VARIABLE	DESCRIPTION
WID	Optional weld ID.
N1	Node ID
N2	Node ID
SN	Normal force at spotweld failure (see Remark 2 below).
SS	Shear force at spotweld failure (see Remark 2 below).
N	Exponent for normal spotweld force (see Remark 2 below).
M	Exponent for shear spotweld force (see Remark 2 below).

VARIABLE	DESCRIPTION
TF	Failure time for nodal constraint set.
EP	Effective plastic strain at failure.
NF	Number of force vectors stored for filtering.
TW	Time window for filtering.

Remarks:

1. Nodes connected by a spot weld cannot be members of another constraint set that constrain the same degrees-of-freedom, a tied interface, or a rigid body, i.e., nodes cannot be subjected to multiple, independent, and possibly conflicting constraints. Also, care must be taken to ensure that single point constraints applied to nodes in a constraint set do not conflict with the constraint sets constrained degrees-of-freedom.

2. Failure of the spot welds occurs when:

$$\left(\frac{|f_n|}{S_n}\right)^n + \left(\frac{|f_s|}{S_s}\right)^m \geq 1$$

   where f_n and f_s are the normal and shear interface force. Component f_n is nonzero for tensile values only.

3. When the failure time, *TF*, is reached the spot weld becomes inactive and the constrained nodes may move freely.

4. Spot weld failure due to plastic straining occurs when the effective nodal plastic strain exceeds the input value, ε_{fail}^{p}. This option can model the tearing out of a spotweld from the sheet metal since the plasticity is in the material that surrounds the spotweld, not the spotweld itself. A least squares algorithm is used to generate the nodal values of plastic strains at the nodes from the element integration point values. The plastic strain is integrated through the element and the average value is projected to the nodes via a least square fit. This option should only be used for the material models related to metallic plasticity and can result is slightly increased run times. Failures can include both the plastic and brittle failures.

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$   *CONSTRAINED_SPOTWELD
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Spotweld two nodes (34574 and 34383) with the approximate strength
$  of a 3/8" SAE Grade No 3 bolt.
$
*CONSTRAINED_SPOTWELD
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$       n1        n2        sn        sf         n         m        tf        ps
     34574     34383      36.0      18.0       2.0       2.0       10.       1.0
$
$
$       sn = 36.0  normal failure force is 36 kN
$       sf = 18.0  shear failure force is 18 kN
$        n = 2.0   normal failure criteria is raised to the power of 2
$        m = 2.0   shear failure criteria is raised to the power of 2
$       tf = 10.0  failure occurs at time 10 unless strain failure occurs
$       ps = 2.0   plastic strain at failure
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
```

***CONSTRAINED_TIE-BREAK**

Purpose: Define a tied shell edge to shell edge interface that can release locally as a function of plastic strain of the shells surrounding the interface nodes. A rather ductile failure is achieved.

Card	1	2	3	4	5	6	7	8
Variable	SNSID	MNSID	EPPF					
Type	I	I	F					
Default	none	none	0.					
Remarks		1, 2	3, 4					

VARIABLE	DESCRIPTION
SNSID	Slave node set ID, see *SET_NODE_*OPTION*.
MNSID	Master node set ID, see *SET_NODE_*OPTION*.
EPPF	Plastic strain at failure

Remarks:

1. Nodes in the master node set must be given in the order they appear as one moves along the edge of the surface.

2. Tie-breaks may not cross.

3. Tie-breaks may be used to tie shell edges together with a failure criterion on the joint. If the average volume-weighted effective plastic strain in the shell elements adjacent to a node exceeds the specified plastic strain at failure, the node is released. The default plastic strain at failure is defined for the entire tie-break but can be overridden in the slave node set to define a unique failure plastic strain for each node.

4. Tie-breaks may be used to simulate the effect of failure along a predetermined line, such as a seam or structural joint. When the failure criterion is reached in the adjoining elements, nodes along the slideline will begin to separate. As this effect propagates, the tie-breaks will appear to "unzip," thus simulating failure of the connection.

*CONSTRAINED_TIED_NODES_FAILURE

Purpose: Define a tied node set with failure based on plastic strain. The nodes must be coincident.

Card	1	2	3	4	5	6	7	8
Variable	NSID	EPPF	ETYPE					
Type	I	F						
Default	none	0.						
Remarks	1, 2, 3, 4							

VARIABLE	DESCRIPTION
NSID	Nodal set ID, see *SET_NODE_*OPTION*.
EPPF	Plastic strain at failure
ETYPE	Element type for nodal group: EQ.0: shell, EQ.1: solid element

Remarks:

1. This feature applies to solid and shell elements using plasticity material models, and to solid elements using the honeycomb material *MAT_HONEYCOMB (EPPF=plastic volume strain). The specified nodes are tied together until the average volume weighted plastic strain exceeds the specified value. Entire regions of individual shell elements may be tied together unlike the tie-breaking shell slidelines. The tied nodes are coincident until failure. When the volume weighted average of the failure value is reached for a group of constrained nodes, the nodes of the elements that exceed the failure value are released to simulate the formation of a crack.

2. To use this feature to simulate failure, each shell element in the failure region should be generated with unique node numbers that are coincident in space with those of adjacent elements. Rather than merging these coincident nodes, the *CONSTRAINED_TIED_NODES_FAILURE option ties the nodal points together. As plastic strain develops and exceeds the failure strain, cracks will form and propagate through the mesh.

3. Entire regions of individual shell elements may be tied together, unlike the *CONSTRAINED_TIE-BREAK option. This latter option is recommended when the location of failure is known, e.g., as in the plastic covers which hide airbags in automotive structures.

4. When using surfaces of shell elements defined using the *CONSTRAINED_TIED_NODES_FAILURE option in contact, it is best to defined each node in the surface as a slave node with the NODE_TO_SURFACE contact options. If this is not possible, the automatic contact algorithms beginning with *CONTACT_AUTOMATIC_... all of which include thickness offsets are recommended.

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$   *CONSTRAINED_TIED_NODES_FAILURE
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Tie shell elements together at the nodes specified in nodal set 101.  The
$  constraint will be broken when the plastic strain at the nodes exceeds 0.085.
$
$  In this example, four shell elements come together at a common point.
$  The four corners of the shells are tied together with failure as opposed
$  to the more common method of merging the nodes in the pre-processing stage.
$
*CONSTRAINED_TIED_NODES_FAILURE
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$     nsid      eppf
       101     0.085
$
$
*SET_NODE_LIST
$      sid
       101
$     nid1      nid2      nid3      nid4      nid5      nid6      nid7      nid8
       775       778       896       897
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

*CONTACT

The keyword *CONTACT provides a way of treating interaction between disjoint parts. Different types of contact may be defined:

CONTACT_OPTION1*_{*OPTION2*}_{*OPTION3*}_{*OPTION4*}_{*OPTION5*}

***CONTACT_AUTO_MOVE**

***CONTACT_COUPLING**

***CONTACT_ENTITY**

CONTACT_GEBOD_OPTION*

***CONTACT_GUIDED_CABLE**

***CONTACT_INTERIOR**

***CONTACT_RIGID_SURFACE**

***CONTACT_1D**

CONTACT_2D_OPTION1*_{*OPTION2*}_{*OPTION3*}

The first, *CONTACT_..., is the general 3D contact algorithms. The second, *CONTACT_ COUPLING, provides a means of coupling to deformable surfaces to MADYMO. The third, *CONTACT_ENTITY, treats contact using mathematical functions to describe the surface geometry for the master surface. The fourth, *CONTACT_GEBOD is a specialized form of the contact entity for use with the rigid body dummies (see *COMPONENT_GEBOD). The fifth, *CONTACT_INTERIOR, is under development and is used with soft foams where element inversion is sometimes a problem. Contact between layers of brick elements is treated to eliminate negative volumes. The sixth, *CONTACT_RIGID_SURFACE is for modeling road surfaces for durability and NVH calculations. The seventh, *CONTACT_1D, remains in LS-DYNA for historical reasons, and is sometimes still used to model rebars which run along edges of brick elements. The last, *CONTACT_2D, is the general 2D contact algorithm based on those used previously in LS-DYNA2D.

CONTACT_OPTION1***_{***OPTION2***}_{***OPTION3***}_{***OPTION4***}_{***OPTION5***}**

Purpose: Define a contact interface.

OPTION1 specifies the contact type. Not all options are implemented for implicit solutions. A list of available contact options is given in Remark 4:

AIRBAG_SINGLE_SURFACE
AUTOMATIC_BEAMS_TO_SURFACE
AUTOMATIC_GENERAL
AUTOMATIC_GENERAL_INTERIOR
AUTOMATIC_NODES_TO_SURFACE
AUTOMATIC_NODES_TO_SURFACE_SMOOTH
AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE
AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE_TIEBREAK
AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE_SMOOTH
AUTOMATIC_SINGLE_SURFACE
AUTOMATIC_SINGLE_SURFACE_MORTAR
AUTOMATIC_SINGLE_SURFACE_SMOOTH
AUTOMATIC_SURFACE_TO_SURFACE
AUTOMATIC_SURFACE_TO_SURFACE_MORTAR
AUTOMATIC_SURFACE_TO_SURFACE_MORTAR_TIED
AUTOMATIC_SURFACE_TO_SURFACE_TIEBREAK
AUTOMATIC_SURFACE_TO_SURFACE_SMOOTH
CONSTRAINT_NODES_TO_SURFACE
CONSTRAINT_SURFACE_TO_SURFACE
DRAWBEAD
ERODING_NODES_TO_SURFACE
ERODING_SINGLE_SURFACE
ERODING_SURFACE_TO_SURFACE
FORCE_TRANSDUCER_CONSTRAINT
FORCE_TRANSDUCER_PENALTY
FORMING_NODES_TO_SURFACE
FORMING_NODES_TO_SURFACE_SMOOTH
FORMING_ONE_WAY_SURFACE_TO_SURFACE
FORMING_SURFACE_TO_SURFACE_MORTAR
FORMING_ONE_WAY_SURFACE_TO_SURFACE_SMOOTH
FORMING_SURFACE_TO_SURFACE

FORMING_SURFACE_TO_SURFACE_SMOOTH
NODES_TO_SURFACE
NODES_TO_SURFACE_INTERFERENCE
NODES_TO_SURFACE_SMOOTH
ONE_WAY_SURFACE_TO_SURFACE
ONE_WAY_SURFACE_TO_SURFACE_INTERFERENCE
ONE_WAY_SURFACE_TO_SURFACE_SMOOTH
RIGID_NODES_TO_RIGID_BODY
RIGID_BODY_ONE_WAY_TO_RIGID_BODY
RIGID_BODY_TWO_WAY_TO_RIGID_BODY
SINGLE_EDGE
SINGLE_SURFACE
SLIDING_ONLY
SLIDING_ONLY_PENALTY
SPOTWELD
SPOTWELD_WITH_TORSION
SURFACE_TO_SURFACE
SURFACE_TO_SURFACE_INTERFERENCE
SURFACE_TO_SURFACE_SMOOTH
SURFACE_TO_SURFACE_CONTRACTION_JOINT
TIEBREAK_NODES_TO_SURFACE
TIEBREAK_NODES_ONLY
TIEBREAK_SURFACE_TO_SURFACE
TIED_NODES_TO_SURFACE
TIED_SHELL_EDGE_TO_SURFACE
TIED_SURFACE_TO_SURFACE
TIED_SURFACE_TO_SURFACE_FAILURE

OPTION2 specifies a thermal contact and is defined by either:

THERMAL

THERMAL_FRICTION

Only the SURFACE_TO_SURFACE contact type may be used with this option.

OPTION3 specifies that the first card to read defines the heading and ID number of contact interface and takes the single option:

ID

OPTION4 specifies that offsets may be used with the tied contacts types. If one of these three offset options is set, then offsets are permitted for these contact types, and, if not, the nodes are projected back to the contact surface during the initialization phase and a constraint formulation is used. Note that in a constraint formulation the nodes of rigid bodies are not permitted in the definition.

OFFSET

Contact types TIED_NODES_TO_SURFACE, TIED_SHELL_EDGE_TO_SURFACE, and TIED_SURFACE_TO_SURFACE may be used with this option. The OFFSET option switches the formulation from a constraint type formulation to one that is penalty based where the force and moment (if applicable) resultants are transferred discrete spring elements between the slave nodes and master segments. For the TIED_SHELL_EDGE_TO_SURFACE contact the BEAM_OFFSET option may be preferred. Rigid bodies can be used with this option. *The nodal points in the TIED_NODES_TO_SURFACE option and the TIED_SURFACE_TO_SURFACE may not be connected to structural nodes, i.e., nodes with rotational degrees-of-freedom, since the rotational degrees-of-freedom are not affected, which will lead to an instability since the translational motions due to rotation are imposed on the slave nodes.*

BEAM_OFFSET

This option applies only to contact type TIED_SHELL_EDGE_TO_SURFACE. If this option is set, then offsets are permitted for this contact type. The BEAM_OFFSET option switches the formulation from a constraint type formulation to one that is penalty based. Beam like springs are used to transfer force and moment resultants between the slave nodes and the master segments. Rigid bodies can be used with this option.

CONSTRAINED_OFFSET

Contact types TIED_NODES_TO_SURFACE, TIED_SHELL_EDGE_TO_SURFACE, and TIED_SURFACE_TO_SURFACE may be used with this option. If this option is set, then offsets are permitted for these contact types. The CONSTRAINED_OFFSET option is a constraint type formulation. *The nodal points in the TIED_NODES_TO_SURFACE option and the TIED_SURFACE_TO_SURFACE may not be connected to structural nodes, i.e., nodes with rotational degrees-of-freedom, since the rotational degrees-of-freedom are not affected, which will lead to an instability since the translational motions due to rotation are imposed on the slave nodes.*

OPTION5 gives extra options specifically for MPP implementation.

MPP

Remarks:

1. *OPTION1*, *OPTION2*, *OPTION3* and *OPTION4* may appear in any order in the keyword command line. The data must be in the order specified below.

2. *OPTION1* is mandatory.

3. *OPTION2*, *OPTION3* and *OPTION4* are optional.

4. The following contact types are available for implicit calculations:

SURFACE_TO_SURFACE
SURFACE_TO_SURFACE_SMOOTH
SURFACE_TO_SURFACE_CONTRACTION_JOINT
NODES_TO_SURFACE
NODES_TO_SURFACE_SMOOTH
ONE_WAY_SURFACE_TO_SURFACE
ONE_WAY_SURFACE_TO_SURFACE_SMOOTH
FORMING_SURFACE_TO_SURFACE
FORMING_SURFACE_TO_SURFACE_SMOOTH
FORMING_NODES_TO_SURFACE
FORMING_NODES_TO_SURFACE_SMOOTH
FORMING_ONE_WAY_SURFACE_TO_SURFACE
FORMING_SURFACE_TO_SURFACE_MORTAR
FORMING_ONE_WAY_SURFACE_TO_SURFACE_SMOOTH
AUTOMATIC_BEAMS_TO_SURFACE
AUTOMATIC_SURFACE_TO_SURFACE
AUTOMATIC_SURFACE_TO_SURFACE_MORTAR
AUTOMATIC_SURFACE_TO_SURFACE_MORTAR_TIED
AUTOMATIC_SURFACE_TO_SURFACE_SMOOTH
AUTOMATIC_NODES_TO_SURFACE
AUTOMATIC_NODES_TO_SURFACE_SMOOTH
AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE
AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE_SMOOTH
AUTOMATIC_SINGLE_SURFACE
AUTOMATIC_SINGLE_SURFACE_MORTAR
AUTOMATIC_SINGLE_SURFACE_SMOOTH
TIED_SURFACE_TO_SURFACE
TIED_NODES_TO_SURFACE
TIED_SHELL_EDGE_TO_SURFACE
TIED_SURFACE_TO_SURFACE_OFFSET
TIED_NODES_TO_SURFACE_OFFSET
TIED_SHELL_EDGE_TO_SURFACE_OFFSET
TIED_SURFACE_TO_SURFACE_BEAM_OFFSET
TIED_NODES_TO_SURFACE_BEAM_OFFSET

TIED_SHELL_EDGE_TO_SURFACE_BEAM_OFFSET
TIED_SURFACE_TO_SURFACE_CONSTRAINED_OFFSET
TIED_NODES_TO_SURFACE_CONSTRAINED_OFFSET
TIED_SHELL_EDGE_TO_SURFACE_CONSTRAINED_OFFSET
2D_AUTOMATIC_SURFACE_TO_SURFACE

5. For smooth contact, a smooth curve-fitted surface is used to represent the master segment, so that it can provide a more accurate representation of the actual surface, reduce the contact noise, and produce smoother results with coarse mesh. All the smooth contacts are available for MPP, only the forming smooth contacts are available for SMP.

DISCUSSION AND EXAMPLES:

A brief discussion on the contact types and a few examples are provided at the end of this section. A theoretical discussion is provided in the LS-DYNA Theory Manual.

Card ordering is important in this section:

- **Card for the ID option is inserted here; otherwise, do not define this card.**

  Define the ID and heading card first.

- **Card for the _MPP option is inserted here; otherwise, do not define this card.**

- **Cards 1 to 3 are mandatory for all contact types.**

- **Card 4 is mandatory for the following contact types:**

  ***CONTACT_CONSTRAINT_type**
  ***CONTACT_DRAWBEAD**
  ***CONTACT_ERODING_type**
  ***CONTACT_..._INTERFERENCE**
  ***CONTACT_RIGID_type**
  ***CONTACT_TIEBREAK_type**
  ***CONTACT_..._CONTRACTION_JOINT_type**

  Each of these types have different Card 4 formats. These card formats are presented in this manual after the optional cards specified above but, if used, Card 4 needs to be specified in your dyna deck before the optional cards.

- **Card for the THERMAL option is inserted here; otherwise, do not define this card.**

  Additional parameters are required for thermal contact and are defined on this card.

- **Optional Card A**

  Additional contact parameters that may be user specified. Default values have evolved over time to become pretty good values for most circumstances.

- **Optional Card B**

  Additional contact parameters that may be user specified. Default values have evolved over time to become pretty good values for most circumstances. If Optional Card B is used, then Optional Card A is mandatory (use a blank line if no changes are desired for Card A parameters).

The following card is read if and only if the ID option is specified.

Optional 1 2-8

Variable	CID	HEADING
Type	I	A70

The contact ID is needed during full deck restarts for contact initialization. If the contact ID is undefined, the default ID is determined by the sequence of the contact definitions, i.e., the first contact definition has an ID of 1, the second, 2, and so forth. In a full deck restart without contact IDs, for a successful run no contact interfaces can be deleted and those which are added must be placed after the last definition in the previous run. The ID and heading is picked up by some of the peripheral LS-DYNA codes to aid in post-processing.

VARIABLE	DESCRIPTION
CID	Contact interface ID. This must be a unique number.
HEADING	Interface descriptor. It is suggested that unique descriptions be used.

*CONTACT…_…MPP

Card 1	1	2	3	4	5	6	7	8
Variable	IGNORE	BUCKET	LCBUCKET	NS2TRACK	INITITER	PARMAX		CPARM8
Type	I	I	I	I	I	F		I
Default	0	200	none	3	2	1.0005		0

VARIABLE	DESCRIPTION
IGNORE	This is the same as the "ignore initial penetrations" option on the *CONTROL_CONTACT Optional Card C entry 2 and can also be specified in the normal contact control cards. It predates both of those, and is not really needed anymore since both are honored by the MPP code. That is, if any of the three are on, initial penetrations are tracked.
BUCKET	Bucketsort frequency. This field is the only way to specify the bucketsort frequency for the MPP code. The BSORT option on Optional Card A is ignored.
LCBUCKET	Loadcurve for bucketsort frequency. The normal input for this is ignored by MPP.
NS2TRACK	Number of potential contacts to track for each slave node. The normal input for this (DEPTH on Optional Card A) is ignored.
INITITER	Number of iterations to perform when trying to eliminate initial penetrations.
PARMAX	The parametric extension distance for contact segments. The MAXPAR parameter on Optional Card A is not used.
CPARM8	Exclude beam to beam contact from the same PID for AUTOMATIC_GENERAL. EQ.0: disable (default) EQ.1: enable

The following card is read if and only if "&" is defined in column 1 of the first field.

Optional	1	2	3	4	5	6	7	8
Variable		CHKSEGS	PENSF	GRPABLE				
Type		I	F	I				
Default		0	1.0	0				

VARIABLE	DESCRIPTION
CHKSEGS	If this value is non-zero, then the node to surface and surface to surface contacts will perform a special check at time 0 for elements that are inverted (or nearly so), and remove them from contact. These poorly formed elements have been known to occur on the tooling in metalforming problems, which allows these problems to run. It should not normally be needed for reasonable meshes.
PENSF	This option is used together with IGNORE for 3D forging problems. If non-zero, the IGNORED penetration distance is multiplied by this value each cycle, effectively pushing the slave node back out to the surface. This is useful for nodes that might get generated below the master surface during 3D remeshing. Care should be exercised, as energy may be generated and stability may be effected for values lower than 0.95. A value in the range of 0.98 to 0.99 or higher (but < 1.0) is recommended.
GRPABLE	This option is experimental at this time. It activates a different contact algorithm available for SINGLE_SURFACE, NODE_TO_SURFACE, and SURFACE_TO_SURFACE contacts. The new algorithm does not support all options as of yet, and is still under development. It can be significantly faster and scale better than the normal algorithm.

Card 1 is mandatory for all contact types.

Card 1	1	2	3	4	5	6	7	8
Variable	SSID	MSID	SSTYP	MSTYP	SBOXID	MBOXID	SPR	MPR
Type	I	I	I	I	I	I	I	I
Default	none	none	none	none			0	0
Remarks	1	2			optional	optional	0=off	0=off

VARIABLE	DESCRIPTION
SSID	Slave segment, node set ID, part set ID, part ID, or shell element set ID, see *SET_SEGMENT, *SET_NODE_*OPTION*, *PART, *SET_PART or *SET_SHELL_*OPTION*. For eroding contact use either a part ID or a part set ID. EQ.0: all part IDs are included for single surface contact, automatic single surface, and eroding single surface.
MSID	Master segment set ID, part set ID, part ID, or shell element set ID, see *SET_SEGMENT, *SET_NODE_*OPTION*, *PART, *SET_PART, or *SET_SHELL_*OPTION*: EQ.0: for single surface contact, automatic single surface, and eroding single surface.
SSTYP	Slave segment or node set type. The type must correlate with the number specified for SSID: EQ.0: segment set ID for surface-to-surface contact, EQ.1: shell element set ID for surface-to-surface contact, EQ.2: part set ID, EQ.3: part ID, EQ.4: node set ID for node to surface contact, EQ.5: include all for single surface definition. EQ.6: part set ID for exempted parts. All non-exempted parts are included in the contact. For *AUTOMATIC_BEAMS_TO_SURFACE contact either a part set ID or a part ID can be specified.

VARIABLE	DESCRIPTION
MSTYP	Master segment set type. The type must correlate with the number specified for MSID: EQ.0: segment set ID, EQ.1: shell element set ID, EQ.2: part set ID, EQ.3: part ID.
SBOXID	Include in contact definition only those slave nodes/segments within box SBOXID (corresponding to BOXID in *DEFINE_BOX), or if SBOXID is negative, only those slave nodes/segments within contact volume |SBOXID| (corresponding to CVID in *DEFINE_CONTACT_VOLUME). A positive value of SBOXID can be used only if SSTYP is set to 2 or 3, i.e., SSID is a part ID or part set ID.
MBOXID	Include in contact definition only those master segments within box MBOXID (corresponding to BOXID in *DEFINE_BOX), or if MBOXID is negative, only those master segments within contact volume |MBOXID| (corresponding to CVID in *DEFINE_CONTACT_VOLUME). A positive value of MBOXID can be used only if MSTYP is set to 2 or 3, i.e., MSID is a part ID or part set ID.
SPR	Include the slave side in the *DATABASE_NCFORC and the *DATABASE _BINARY_INTFOR interface force files: EQ.1: slave side forces included.
MPR	Include the master side in the *DATABASE_NCFORC and the *DATABASE_BINARY_INTFOR interface force files: EQ.1: master side forces included.

Remarks:

1. Giving a slave set ID equal to zero is valid only for the single surface contact algorithms, i.e., the options SINGLE_SURFACE, and the AUTOMATIC_, AIRBAG_, and ERODING_SINGLE_SURFACE options.

2. A master set ID is not defined for the single surface contact algorithms (including AUTOMATIC_GENERAL). A master set ID is optional for FORCE_ TRANSDUCERS. If a master set is defined for the FORCE_TRANSDUCER option, only those force that develop between and master and slave surfaces are considered. *The master surface option is only implemented for the _PENALTY option and works only with the AUTOMATIC_SINGLE_SURFACE contact types.*

Card 2 is mandatory for all contact types.

Card 2	1	2	3	4	5	6	7	8
Variable	FS	FD	DC	VC	VDC	PENCHK	BT	DT
Type	F	F	F	F	F	I	F	F
Default	0.	0.	0.	0.	0.	0	0.	1.0E20
Remarks								

VARIABLE **DESCRIPTION**

FS Static coefficient of friction if FS is >0 and not equal to 2. The frictional coefficient is assumed to be dependent on the relative velocity v_{rel} of the surfaces in contact $\mu_c = FD + (FS - FD)e^{-DC\cdot|v_{rel}|}$. The two other possibilities are:

EQ.-2: If the frictional coefficients defined in the *DEFINE_ FRICTION section are to be used, set FS to the negative number, -2.0.

EQ.-1: If the frictional coefficients defined in the *PART section are to be used, set FS to the negative number, -1.0.

WARNING: Please note that the FS=-1.0 and FS=-2.0 options apply only to contact types:

SINGLE_SURFACE,
AUTOMATIC_GENERAL,
AUTOMATIC_SINGLE_SURFACE,
AUTOMATIC_NODES_TO_SURFACE,
AUTOMATIC_SURFACE_TO_SUFACE,
AUTOMATIC_ONE_WAY_SURFACE_TO_SU…,
ERODING_SINGLE_SURFACE.

EQ.2: For contact type ONE_WAY_SURFACE_TO_SURFACE, the dynamic coefficient of friction points to the table, see DEFINE_TABLE (The table ID is give by FD below.), giving the coefficient of friction as a function of the relative velocity and pressure. This option must be used in combination with the thickness offset option. See Figure 7.1.

FD Dynamic coefficient of friction. The frictional coefficient is assumed to be dependent on the relative velocity v_{rel} of the surfaces in contact $\mu_c = FD + (FS - FD)e^{-DC\cdot|v_{rel}|}$. Give table ID if FS=2.

VARIABLE	DESCRIPTION
	Note: For the special contact option "TIED_SURFACE_TO_SURFACE_FAILURE" only, the variables FS and FD act as failure stresses, i.e., failure occurs if $\left[\frac{\max(0.0,\sigma_{normal})}{FS}\right]^2+\left[\frac{\sigma_{shear}}{FD}\right]^2-1>0$ where σ_{normal} and σ_{shear} are the interface normal and shear stresses.
FS	Normal tensile stress at failure
FD	Shear stress at failure
DC	Exponential decay coefficient. The frictional coefficient is assumed to be dependent on the relative velocity v_{rel} of the surfaces in contact $\mu_c = FD + (FS - FD)e^{-DC\cdot\lvert v_{rel}\rvert}$.
VC	Coefficient for viscous friction. This is necessary to limit the friction force to a maximum. A limiting force is computed $F_{lim} = VC \cdot A_{cont}$. A_{cont} being the area of the segment contacted by the node in contact. The suggested value for *VC* is to use the yield stress in shear $VC = \frac{\sigma_o}{\sqrt{3}}$ where σ_o is the yield stress of the contacted material.
VDC	Viscous damping coefficient in percent of critical. In order to avoid undesirable oscillation in contact, e.g., for sheet forming simulation, a contact damping perpendicular to the contacting surfaces is applied. Damping coefficient $\xi = \frac{VDC}{100}\xi_{wd,}$ eg VDC = 20. ξ_{crit} is determined in the following fashion by LS-DYNA. $\xi_{crit} = 2mw; \quad m = min(m_{slave}, m_{master})$ *mass of master resp. slave node* $w = \sqrt{k \cdot \frac{m_{slave}+m_{master}}{m_{slave}\cdot m_{master}}}$ k interface stiffness
PENCHK	Small penetration in contact search option. If the slave node penetrates more than the segment thickness times the factor XPENE, see *CONTROL_CONTACT, the penetration is ignored and the slave node is set free. The thickness is taken as the shell thickness if the segment belongs to a shell element or it is taken as 1/20 of its shortest diagonal if the segment belongs to a solid element. This option applies to the surface-to-surface contact algorithms: See Table 7.1 for contact types and more details.

VARIABLE	DESCRIPTION
	EQ.0: check is turned off, EQ.1: check is turned on, EQ.2: check is on but shortest diagonal is used.
BT	Birth time (contact surface becomes active at this time). EQ.0: Birth time is inactive, i.e., contact is always active LT.0: Birth time, |BT|, is active during dynamic relaxation, and after dynamic relaxation contact is always active GT.0: If DT>0, birth time applies both during and after dynamic relaxation.
DT	Death time (contact surface is deactivated at this time). LT.0: Birth time, |BT|, is active after dynamic relaxation is completed and DT=|DT|. During dynamic relaxation, the contact is inactive. EQ.0: DT defaults to 1.E+20.

Card 3 is mandatory for all contact types.

Card 3	1	2	3	4	5	6	7	8
Variable	SFS	SFM	SST	MST	SFST	SFMT	FSF	VSF
Type	F	F	F	F	F	F	F	F
Default	1.	1.	element thickness	element thickness	1.	1.	1.	1.

VARIABLE	DESCRIPTION
SFS	Scale factor on default slave penalty stiffness when SOFT=0 or SOFT=2, see also *CONTROL_ CONTACT.
SFM	Scale factor on default master penalty stiffness when SOFT=0 or SOFT=2, see also *CONTROL_CONTACT.
SST	Optional thickness for slave surface (overrides true thickness). This option applies only to contact with shell elements. True thickness is the element thickness of the shell elements. For the *CONTACT_TIED_.. options, SST and MST below can be defined as negative values, which will cause the determination of whether or not a node is tied to depend only on the separation distance relative to the absolute value of these thicknesses. More information is given under General Remarks on *CONTACT following Optional Card C.
MST	Optional thickness for master surface (overrides true thickness). This option applies only to contact with shell elements. True thickness is the element thickness of the shell elements. For the TIED options see SST above.
SFST	Scale factor for slave surface thickness (scales true thickness). This option applies only to contact with shell elements. True thickness is the element thickness of the shell elements.
SFMT	Scale factor for master surface thickness (scales true thickness). This option applies only to contact with shell elements. True thickness is the element thickness of the shell elements.
FSF	Coulomb friction scale factor. The Coulomb friction value is scaled as $\mu_{sc} = FSF \cdot \mu_c$, see above.
VSF	Viscous friction scale factor. If this factor is defined then the limiting force becomes: $F_{lim} = VSF \cdot VC \cdot A_{cont}$, see above.

Remarks:

The variables FSF and VSF above can be overridden segment by segment on the *SET_SEGMENT or *SET_SHELL_*OPTION* cards for the **slave surface only** as A3 and A4, and for the **master surface only** as A1 and A2. See *SET_SEGMENT and *SET_SHELL_*OPTION*.

This Card 4 is mandatory for:

***CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_TIEBREAK_**{*OPTION*}

***CONTACT_AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE_TIEBREAK _**{*OPTION*}

For the response variable OPTION below, if set to 9 three damping constants can be defined for the various failure modes. To do this, set the option to

DAMPING

Card 4

1	2	3	4	5	6	7	8
Variable	OPTION	NFLS	SFLS	PARAM	ERATEN	ERATES	CT2CN	CN
Type	I	F	F	F	F	F	F	F
Default	required	required	required	opt=2,6,.,11	opt=7,9.11	opt=7,9.11	opt=9..11	

Define the following card if and only if OPTION=9 and the _DAMPING option is set on the keyword card.

optional

Variable	DMP_1	DMP_2	DMP_3				
Type	F	F	F				
Default	0.0	0.0	0.0				

VARIABLE	DESCRIPTION
OPTION	Response: EQ.-3: see 3, moments are transferred. SMP only. EQ.-2: see 2, moments are transferred. SMP only. EQ.-1: see 1, moments are transferred. SMP only.

VARIABLE	DESCRIPTION
	EQ.1: slave nodes in contact and which come into contact will permanently stick. Tangential motion is inhibited. EQ.2: tiebreak is active for nodes which are initially in contact. Until failure, tangential motion is inhibited. If PARAM is set to unity, (1.0) shell thickness offsets are ignored, and the orientation of the shell surfaces is required such that the outward normals point to the opposing contact surface. EQ.3: as 1 above but with failure after sticking. EQ.4: tiebreak is active for nodes which are initially in contact but tangential motion with frictional sliding is permitted. EQ.5: tiebreak is active for nodes which are initially in contact. Stress is limited by the yield condition described in Remark 5 below. Damage is a function of the crack width opening. The damage function is defined by a load curve which starts at unity for a crack width of zero and decays in some way to zero at a given value of the crack opening. This interface can be used to represent deformable glue bonds. EQ.6: This option is for use with solids and thick shells only. Tiebreak is active for nodes which are initially in contact. Failure stress must be defined for tiebreak to occur. After the failure stress tiebreak criterion is met, damage is a linear function of the distance C between points initially in contact. When the distance is equal to PARAM damage is fully developed and interface failure occurs. After failure, this contact option behaves as a surface-to-surface contact. EQ.7: Dycoss Discrete Crack Model. "_ONE_WAY_SURFACE_TO_SURFACE_TIEBREAK" definition is recommended for this option. See Remark 7. EQ.8: This is similar to OPTION=6 but works with offset shell elements. "_ONE_WAY_SURFACE_TO_SURFACE_TIEBREAK" definition is recommended for this option. EQ.9: Extension of OPTION=7. Discrete Crack Model with power law and B-K damage models. "_ONE_WAY_SURFACE_TO_SURFACE_TIEBREAK" definition is recommended for this option. See Remark 8. EQ.10: This is similar to OPTION=7 but works with offset shell elements. "_ONE_WAY_SURFACE_TO_SURFACE_TIEBREAK" definition is recommended for this option. EQ.11: This is similar to OPTION=9 but works with offset shell elements. "_ONE_WAY_SURFACE_TO_SURFACE_TIEBREAK" definition is recommended for this option.
NFLS	Normal failure stress for OPTION=2, 3, 4, 6, 7, 8, 9, 10 or 11. For OPTION=5 NFLS becomes the plastic yield stress as defined in Remark 5. For OPTION=9 or 11 and NFLS<0, a load curve ID=-NFLS is referenced defining normal failure stress as a function of element size. See remarks.

VARIABLE	DESCRIPTION
SFLS	Shear failure stress for OPTION=2, 3, 6, 7, 8, 9, 10 or 11. For OPTION=4, SFLS is a frictional stress limit if PARAM=1. This frictional stress limit is independent of the normal force at the tie. For OPTION=5 SFLS becomes the load curve ID of the damage model. For OPTION=9 or 11 and SFLS<0, a load curve ID=-SFLS is referenced defining shear failure stress as a function of element size. See remarks.
PARAM	For OPTION=2, setting PARAM=1 causes the shell thickness offsets to be ignored. For OPTION=4, setting PARAM=1 causes SFLS to be a frictional stress limit. For OPTION=6 or 8, PARAM is the critical distance, CCRIT, at which the interface failure is complete. For OPTION=7 or 10 PARAM is the friction angle in degrees. For OPTION=9 or 11, it is the exponent in the damage model. A positive value invokes the power law, while a negative one, the B-K model. See MAT_138 for additional details.
ERATEN	For OPTION=7,9,10,11 only. Normal energy release rate (stress*length) used in damage calculation, see Lemmen and Meijer [2001].
ERATES	For OPTION=7,9,10,11 only. Shear energy release rate (stress*length) used in damage calculation, see Lemmen and Meijer [2001].
CT2CN	The ratio of the tangential stiffness to the normal stiffness for OPTION=9,11. The default is 1.0.
CN	Normal stiffness (stress/length) for OPTION=9,11. If CN is not given explicitly, penalty stiffness is used (default). This optional stiffness should be used with care, since contact stability can get affected. A warning message with a recommended time step is given initially.
DMP_1	Mode I damping force per unit velocity per unit area.
DMP_2	Mode II damping force per unit velocity per unit area.
DMP_3	Mode III damping force per unit velocity per unit area.

Remarks:

1. After failure, this contact option behaves as a surface-to-surface contact with thickness offsets. After failure, no interface tension is possible.
2. The soft constraint option with SOFT=2 is not implemented for the tiebreak option.
3. For OPTION = 2, 3, and 6 the tiebreak failure criterion has normal and shear components:

$$\left(\frac{|\sigma_n|}{NFLS}\right)^2+\left(\frac{|\sigma_s|}{SFLS}\right)^2\geq 1$$

4. For OPTION = 4, the tiebreak failure criterion has only a normal stress component:

$$\frac{|\sigma_n|}{NFLS} \geq 1$$

5. For OPTION = 5, the stress is limited by a perfectly plastic yield condition. For ties in tension, the yield condition is

$$\frac{\sqrt{\sigma_n^2 + 3|\sigma_s|^2}}{NLFS} \leq 1$$

For ties in compression, the yield condition is

$$\frac{\sqrt{3|\sigma_s|^2}}{NLFS} \leq 1$$

The stress is also scaled by the damage function which is obtained from the load curve. For ties in tension, both normal and shear stress are scaled. For ties in compression, only shear stress is scaled.

6. For OPTION = 6 or 8, damage initiates when the stress meets the failure criterion. The stress is then scaled by the damage function. Assuming no load reversals, the energy released due to the failure of the interface is approximately 0.5*S*CCRIT, where

$$S = \sqrt{\max(\sigma_n, 0)^2 + |\sigma_s|^2}$$

at initiation of damage. This interface may be used for simulating crack propagation. For the energy release to be correct, the contact penalty stiffness must be much larger than

$$\frac{MIN(NFLF, SFLS)}{CCRIT}.$$

7. OPTION = 7 or 10 is the Dycoss Discrete Crack Model as described in Lemmen and Meijer [2001]. The relation for the crack initiation is given as

$$\left(\frac{\max(\sigma_n, 0)}{NFLS}\right)^2 + \left(\frac{\sigma_s}{SFLS\left(1 - \sin(PARAM)\min(0, \sigma_n)\right)}\right) = 1$$

8. OPTION = 9 or 11 is an extension of the Dycoss Discrete Crack Model based on the fracture model in the cohesive material model *MAT_COHESIVE_MIXED_MODE, where the model is described in detail. Failure stresses/peak tractions NFLS and/or SFLS can be defined as functions of characteristic element length (square root of master segment area) via load curve. This option is useful to get nearly the same global responses (e.g. load-displacement curve) with coarse meshes compared to a fine mesh solution. In general, lower peak tractions are needed for coarser meshes. See also *MAT_138.

This Card 4 is mandatory for:

***CONTACT_CONSTRAINT_NODES_TO_SURFACE**

***CONTACT_CONSTRAINT_SURFACE_TO_SURFACE**

Card 4	1	2	3	4	5	6	7	8
Variable	KPF							
Type	F							
Default	0.0							

VARIABLE	DESCRIPTION
KPF	Kinematic partition factor for constraint: EQ. 0.0: fully symmetric treatment. EQ.1.0: one way treatment with slave nodes constrained to master surface. Only the slave nodes are checked against contact. EQ.-1.0: one way treatment with master nodes constrained to slave surface. Only the master nodes are checked against contact.

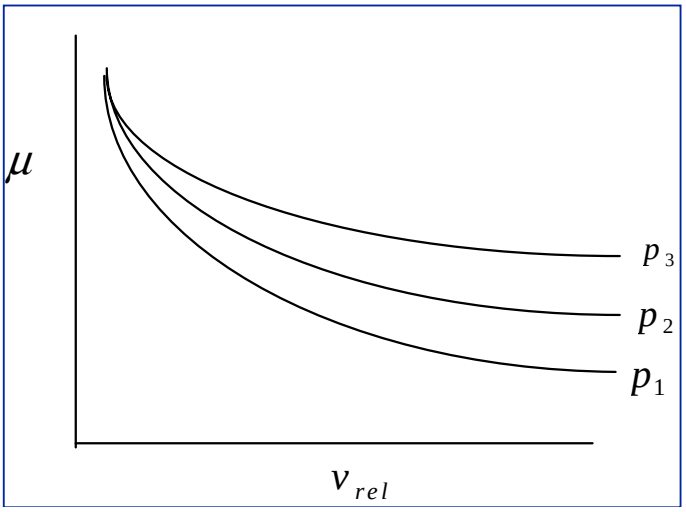


Figure 7.1. Friction coefficient, μ, can be a function of relative velocity and pressure. Specify a flag for the static coefficient of friction, FS, and a table ID for the dynamic coefficient. This option only works with ONE_WAY_SURFACE_TO_ SURFACE with thickness offsets.

This Card 4 is mandatory for:

***CONTACT_DRAWBEAD**

***CONTACT_DRAWBEAD_INITIALIZE**

Card 4	1	2	3	4	5	6	7	8
Variable	LCIDRF	LCIDNF	DBDTH	DFSCL	NUMINT	DBPID	ELOFF	
Type	I	I	F	F	I	I	I	
Default	required	none	0.0	1.0	0	0	0	

If the option INITIALIZE is active, then define the following card to initialize the plastic strain and thickness of elements that pass under the drawbead.

Optional	1	2	3	4	5	6	7	8
Variable	LCEPS	TSCALE	LCEPS2	OFFSET				
Type	I	F	I	F				
Default	required	1.0	optional	optional				

VARIABLE	DESCRIPTION
LCIDRF	If LCIDRF is positive then it defines the load curve ID giving the bending component of the restraining force, $F_{bending}$, per unit draw bead length as a function of displacement, δ, see Figure 7.2. This force is due to the bending and unbending of the blank as it moves through the draw bead. The total restraining force is the sum of the bending and friction components. If LCIDRF is negative then the absolute value gives the load curve ID defining max bead force versus normalized draw bead length. The abscissa values are between zero and 1 and is the normalized draw bead length. The ordinate gives the maximum allowed draw bead, retaining force when the bead is in the fully closed position. If the draw bead is not fully closed linear interpolation is used to compute the draw bead force.

VARIABLE	DESCRIPTION
LCIDNF	Load curve ID giving the normal force per unit draw bead length as a function of displacement, δ, see Figure 7.2. This force is due to the bending of the blank into the draw bead as the binder closes on the die and represents a limiting value. The normal force begins to develop when the distance between the die and binder is less than the draw bead depth. *As the binder and die close on the blank this force should diminish or reach a plateau,* see the Remarks section.
DBDTH	Draw bead depth, see Figure 7.2. Necessary to determine correct δ displacement from contact displacements.
DFSCL	Scale factor for load curve. Default=1.0. This factor scales load curve ID, LCIDRF above.
NUMINT	Number of equally spaced integration points along the draw bead: EQ.0: Internally calculated based on element size of elements that interact with draw bead. This is necessary for the correct calculation of the restraining forces. More integration points may increase the accuracy since the force is applied more evenly along the bead.
DBPID	Optional part ID for the automatically generated truss elements for the draw bead display in the post-processor. If undefined LS-DYNA assigns a unique part ID.
ELOFF	Option to specify and element ID offset for the truss elements that are automatically generated for the draw bead display. If undefined LS-DYNA chooses a unique offset.
LCEPS	Load curve ID defining the plastic strain versus the parametric coordinate through the shell thickness. The parametric coordinate should be defined in the interval between -1 and 1 inclusive. The value of plastic strain at the integration point is interpolated from this load curve. If the plastic strain at an integration point exceeds the value of the load curve at the time initialization occurs, the plastic strain at the point will remain unchanged.
TSCALE	Scale factor that multiplies the shell thickness as the shell element moves under the draw bead.
LCEPS2	Optional load curve ID defining the plastic strain versus the parametric coordinate through the shell thickness, which is used after an element has traveled a distance equal to OFFSET. The parametric coordinate should be defined in the interval between -1 and 1 inclusive. The value of plastic strain at the integration point is interpolated from this load curve. If the plastic strain at an integration point exceeds the value of

VARIABLE	DESCRIPTION
	the load curve at the time initialization occurs, the plastic strain at the point will remain unchanged. Input parameters LCEPS2 and OFFSET provides a way to model the case where a material moves under two draw beads. In this latter case the curve would be the sum of the plastic strains generate by moving under two consecutive beads.
OFFSET	If the center of an element has moved a distance equal to OFFSET, the load curve ID, LCEPS2 is used to reinitialize the plastic strain. The TSCALE scale factor is also applied.

Remarks:

The draw bead is defined three ways:

1. A *consecutive* list of *slave* nodes that lie along the bead.
2. A part ID of a beam that lies along the draw bead.
3. A part set ID of beams that lie along the draw bead.

For straight draw beads only two nodes or a single beam need to be defined, i.e., one at each end, but for curved beads sufficient nodes or beams are required to define the curvature of the bead geometry. When beams are used to define the bead, with the exception of the first and last node, each node must connect with two beam elements. This requirement means that the number of slave nodes equals the number of beam elements plus one. The integration points along the bead are equally spaced and are independent of the nodal spacing used in the definition of the draw bead. By using the capability of tying extra nodes to rigid bodies (see *CONSTRAINED_ EXTRA_NODES or *CONSTRAINED_RIGID_BODIES) the draw bead nodal points do not need to belong to the element connectivities of the die and binder. The blank makes up the master surface. IT IS HIGHLY RECOMMENDED TO DEFINE A BOXID AROUND THE DRAWBEAD TO LIMIT THE SIZE OF THE MASTER SURFACE CONSIDERED FOR THE DRAW BEAD. THIS WILL SUBSTANTIALLY REDUCE COST AND MEMORY REQUIREMENTS.

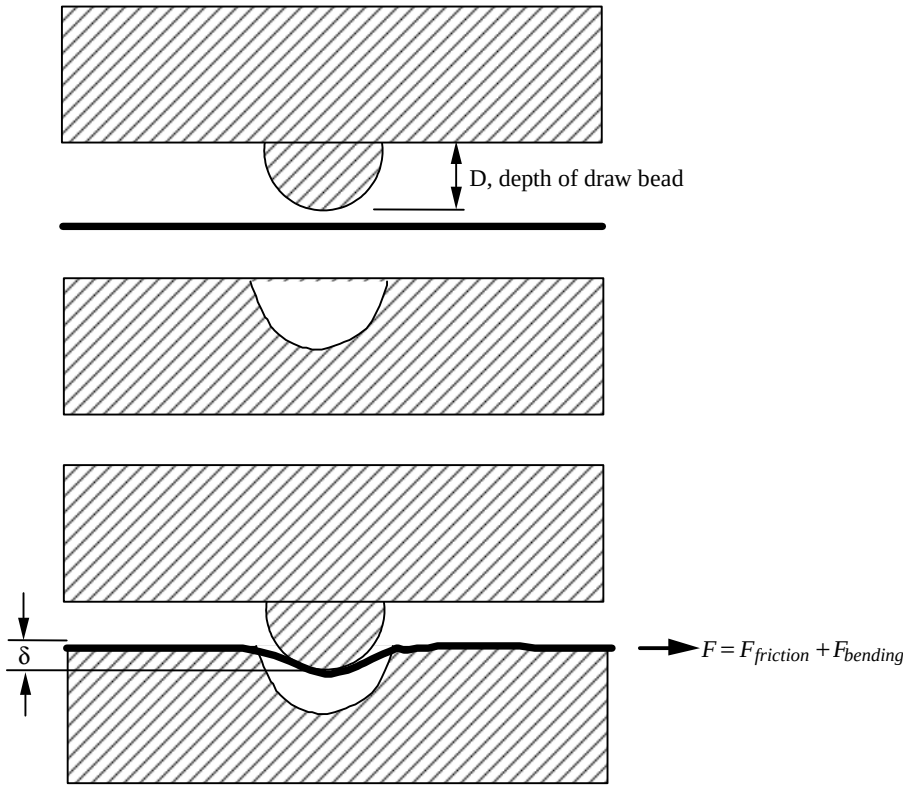


Figure 7.2.

This Card 4 is mandatory for:

***CONTACT_ERODING_NODES_TO_SURFACE**

***CONTACT_ERODING_SINGLE_SURFACE**

***CONTACT_ERODING_SURFACE_TO_SURFACE**

Card 4	1	2	3	4	5	6	7	8
Variable	ISYM	EROSOP	IADJ					
Type	I	I	I					
Default	0	0	0					

VARIABLE	DESCRIPTION
ISYM	Symmetry plane option: EQ.0: off, EQ.1: do not include faces with normal boundary constraints (e.g., segments of brick elements on a symmetry plane). This option is important to retain the correct boundary conditions in the model with symmetry.
EROSOP	Erosion/Interior node option: EQ.0: only exterior boundary information is saved, EQ.1: storage is allocated so that eroding contact can occur. Otherwise, no contact is assumed after erosion of the corresponding element.
IADJ	Adjacent material treatment for solid elements: EQ.0: solid element faces are included only for free boundaries, EQ.1: solid element faces are included if they are on the boundary of the material subset. This option also allows the erosion within a body and the subsequent treatment of contact.

Remarks:

Eroding contact may control the timestep (see ECDT in *CONTROL_CONTACT). For ERODING_NODES_TO_SURFACE, define the slave side using a node set, not a part ID or part set ID.

This Card 4 is mandatory for:

***CONTACT_NODES_TO_SURFACE_INTERFERENCE**

***CONTACT_ONE_WAY_SURFACE_TO_SURFACE_INTERFERENCE**

***CONTACT_SURFACE_TO_SURFACE_INTERFERENCE**

Purpose: This contact option provides a means of modeling parts which are shrink fitted together and are, therefore, prestressed in the initial configuration. This option turns off the nodal interpenetration checks (which changes the geometry by moving the nodes to eliminate the interpenetration) at the start of the simulation and allows the contact forces to develop to remove the interpenetrations. The load curves defined in this section scale the interface stiffness constants such that the stiffness can increase slowly from zero to a final value with effect that the interface forces also increase gradually to remove the overlaps.

Card 4	1	2	3	4	5	6	7	8
Variable	LCID1	LCID2						
Type	I	I						
Default	0	0						

VARIABLE	DESCRIPTION
LCID1	Load curve ID which scales the interface stiffness during dynamic relaxation. This curve must originate at (0,0) at time=0 and gradually increase.
LCID2	Load curve ID which scales the interface stiffness during the transient calculation. This curve generally has a constant value of unity for the duration of the calculation if LCID1 is defined. If LCID1=0, this curve must originate at (0,0) at time=0 and gradually increase to a constant value.

Remarks:

Extreme caution must be used with this option. First, shell thickness offsets are taken into account for deformable shell elements. Furthermore, SEGMENT ORIENTATION FOR SHELL ELEMENTS AND INTERPENETRATION CHECKS ARE SKIPPED. Therefore, it is necessary in the problem setup to ensure that all contact segments which belong to shell elements are properly oriented, i.e., the outward normal vector of the segment based on the right hand rule relative to the segment numbering, must point to the opposing contact surface; consequently, automatic contact generation should be avoided for parts composed of shell elements unless automatic generation is used on the slave side of a nodes to surface interface.

This Card 4 is mandatory for:

***CONTACT_RIGID_NODES_TO_RIGID_BODY**

***CONTACT_RIGID_BODY_ONE_WAY_TO_RIGID_BODY**

***CONTACT_RIGID_BODY_TWO_WAY_TO_RIGID_BODY**

Card 4	1	2	3	4	5	6	7	8
Variable	LCID	FCM	US		LCDC	DSF	UNLCID	
Type	I	I	F		I	F	I	
Default	required	required	from LCID		optional	0.0	optional	

VARIABLE	DESCRIPTION
LCID	Load curve ID giving force versus penetration behavior for RIGID_ contact. See also the definition of FCM below.
FCM	Force calculation method for RIGID_contact: EQ.1: Load curve gives total normal force on surface versus maximum penetration of any node (RIGID_BODY_ONE_WAY only). EQ.2: Load curve gives normal force on each node versus penetration of node through the surface (all RIGID_contact types). EQ.3: Load curve gives normal pressure versus penetration of node through the surface (RIGID_BODY_TWO_WAY and RIGID_BODY_ONE_WAY only). EQ.4: Load curve gives total normal force versus maximum soft penetration. In this case the force will be followed based on the original penetration point. (RIGID_BODY_ONE_WAY only).
US	Unloading stiffness for RIGID_contact. The default is to unload along the loading curve. This should be equal to or greater than the maximum slope used in the loading curve.
LCDC	Load curve ID giving damping coefficient (DC) versus penetration velocity. The damping force FD is then: FD = DSF * DC * velocity.
DSF	Damping scaling factor.
UNLCID	Optional load curve ID giving force versus penetration behavior for RIGID_BODY_ONE_WAY contact. This option requires the definition of the unloading stiffness, US. See Figure 7.3.

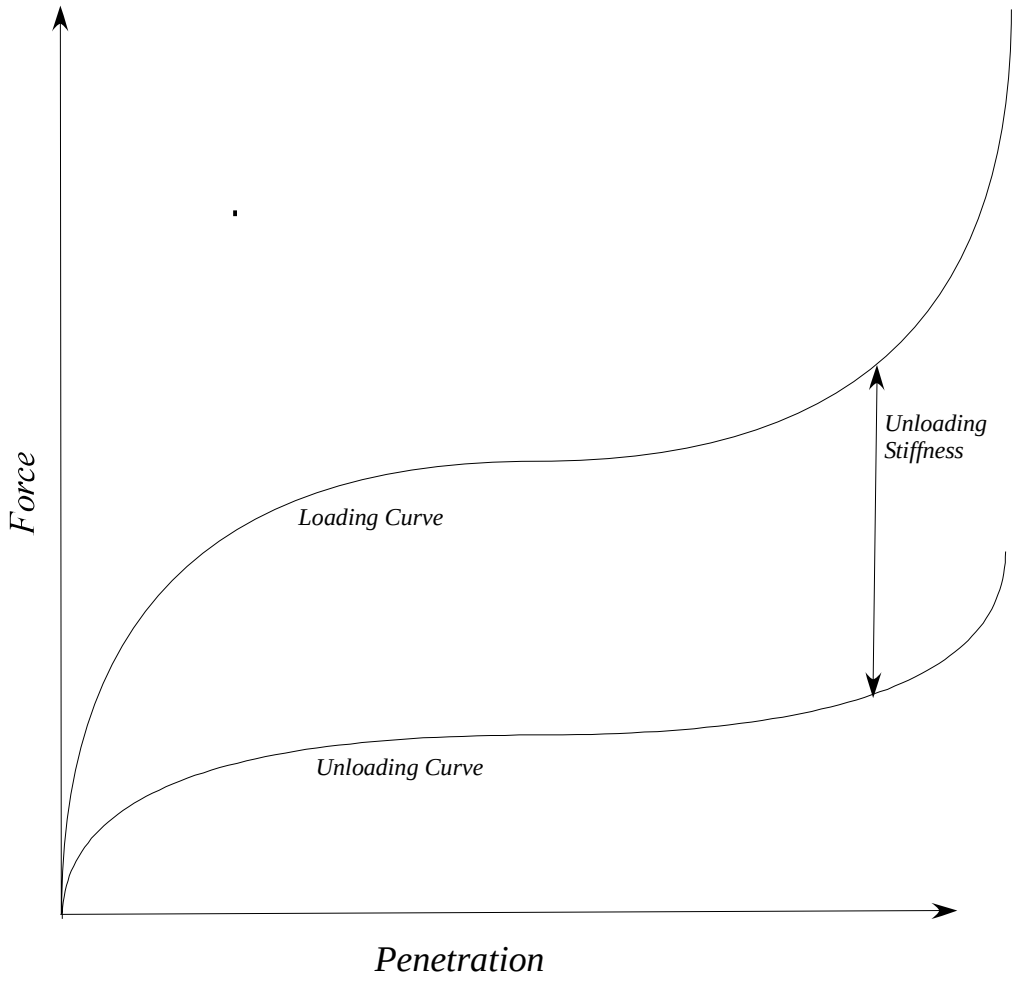


Figure 7.3. Behavior if an unloading curve is defined.

This Card 4 is mandatory for:

***CONTACT_TIEBREAK_NODES_TO_SURFACE and**

***CONTACT_TIEBREAK_NODES_ONLY**

Card 4	1	2	3	4	5	6	7	8
Variable	NFLF	SFLF	NEN	MES				
Type	F	F	F	F				
Default	required	required	2.	2.				

VARIABLE	DESCRIPTION
NFLF	Normal failure force. Only tensile failure, i.e., tensile normal forces, will be considered in the failure criterion.
SFLF	Shear failure force
NEN	Exponent for normal force
MES	Exponent for shear force. Failure criterion: $\left(\frac{\lvert f_n \rvert}{NFLF}\right)^{NEN} + \left(\frac{\lvert f_s \rvert}{SFLF}\right)^{MES} \geq 1.$ Failure is assumed if the left side is larger than 1. f_n and f_s are the normal and shear interface force.

Remarks:

These attributes can be overridden node by node on the *SET_NODE_option cards.

Both NFLF and SFLF must be defined. If failure in only tension or shear is required then set the other failure force to a large value (1E+10).

After failure, the contact_tiebreak_nodes_to_surface behaves as a nodes-to-surface contact with no thickness offsets (no interface tension possible) whereas the contact_tiebreak_nodes_only stops acting altogether. Prior to failure, the two contact types behave identically.

This Card 4 is mandatory for:

***CONTACT_TIEBREAK_SURFACE_TO_SURFACE and**

***CONTACT_TIEBREAK_SURFACE_TO_SURFACE_ONLY**

Card 4	1	2	3	4	5	6	7	8
Variable	NFLS	SFLS	TBLCID	THKOFF				
Type	F	F	I	I				
Default	required	required	0	0				

VARIABLE	DESCRIPTION
NFLS	Tensile failure stress. See remark below.
SFLS	Shear failure stress. Failure criterion: $\left(\frac{\lvert\sigma_n\rvert}{NFLS}\right)^2+\left(\frac{\lvert\sigma_s\rvert}{SFLS}\right)^2\geq 1.$
TBLCID	Optional load curve number defining the resisting stress versus gap opening for the post failure response. This can be used to model the failure of adhesives.
THKOFF	Thickness offsets are considered if THKOFF=1. If shell offsets are included in the meshed geometry, this option is highly recommended since segment orientation can be arbitrary and the contact surfaces can be disjoint.

Remarks:

The failure attributes can be overridden segment by segment on the *SET_SEGMENT or *SET_SHELL_option cards for the **slave surface** as A1 and A2. These variables do not apply to the master surface. Both NFLS and SFLS must be defined. If failure in only tension or shear is required then set the other failure stress to a large value (1E+10). When used with shells, contact segment normals are used to establish the tension direction (as opposed to compression). Compressive stress does not contribute to the failure equation.

After failure, *CONTACT_TIEBREAK_SURFACE_TO_SURFACE behaves as a surface-to-surface contact with no thickness offsets.

After failure, *CONTACT_TIEBREAK_SURFACE_TO_SURFACE_ONLY stops acting altogether. Until failure, it ties the slave nodes to the master nodes.

This Card 4 is mandatory for:

***CONTACT_SURFACE_TO_SURFACE_CONTRACTION_JOINT**

Purpose: This contact option turns on the contraction joint model designed to simulate the effects of sinusoidal joint surfaces (shear keys) in the contraction joints of arch dams and other concrete structures. The sinusoidal functions for the shear keys are defined according to the following three methods [Solberg and Noble 2002]:

Method 1: $\hat{g} = g - A(1-\cos(B(s_2 - s_1)))$

Method 2: $\hat{g} = g - 2A\left|\sin(B(s_2 - s_1)/2)\right|$

Method 3: (default) $\hat{g} = g - A\cos(Bs_2) + A\cos(Bs_1)$

Where g is gap function for contact surface, $\hat{g}$ is gap function for the joint surface. A is key amplitude parameter, and B is key frequency parameter. s_1 and s_2 are referential surfaces:

$$s_1 = \mathbf{X}_{surface1} * \mathbf{T}_{key}$$
$$s_2 = \mathbf{X}_{surface2} * \mathbf{T}_{key}$$
$$\mathbf{T}_{key} = \mathbf{T}_{slide} \times \mathbf{n}$$

Where $\mathbf{T}_{slide}$ is the free sliding direction of the keys, $\mathbf{n}$ is the surface normal in reference.

Card 4

1	2	3	4	5	6	7	8
Variable	MTCJ	ALPHA	BETA	TSVX	TSVY	TSVZ		
Type	I	F	F	F	F	F		
Default	0	0.0	0.0	0.0	0.0	0.0		

VARIABLE	DESCRIPTION
MTCJ	The method option for the gap function, $\hat{g}$
ALPHA	Key amplitude parameter A
BETA	Key frequency parameter B
TSVX	X component of the free sliding direction $\mathbf{T}_{slide}$
TSVY	Y component of the free sliding direction $\mathbf{T}_{slide}$
TSVZ	Z component of the free sliding direction $\mathbf{T}_{slide}$

This Card is mandatory for the THERMAL option, i.e.,:

Reminder: If Card 4 is required, then it must go before this optional card. (Card 4 is required for certain contact types - see earlier in this section for the list, later in this section for details of Card 4.)

***CONTACT_ ..._THERMAL_.....**

Optional	1	2	3	4	5	6	7	8
Variable	K	FRAD	H0	LMIN	LMAX	CHLM	BC_FLG	ALGO
Type	F	F	F	F	F	F	I	I
Default	none	none	none	none	none	1.0	0	0

VARIABLE **DESCRIPTION**

K Thermal conductivity of fluid between the contact surfaces. If a gap with a thickness l_{gap} exists between the contact surfaces, then the conductance due to thermal conductivity between the contact surfaces is

$$h_{cond} = \frac{K}{l_{gap}}$$

Note that LS- DYNA calculates l_{gap} based on deformation.

FRAD Radiation factor between the contact surfaces.

$$f_{rad} = \frac{\sigma}{\frac{1}{\varepsilon_1} + \frac{1}{\varepsilon_2} - 1}$$

Where: σ = Stefan Boltzman constant
ε_1 = emissivity of master surface
ε_2 = emissivity of slave surface

LS-DYNA calculates a radiant heat transfer conductance

$$h_{rad} = f_{rad}\left(T_m + T_s\right)\left(T_m^2 + T_s^2\right)$$

VARIABLE	DESCRIPTION
H0	Heat transfer conductance for closed gaps. Use this heat transfer conductance for gaps in the range $0 \le l_{gap} \le l_{\min}$
LMIN	Minimum gap ($l_{\min}$), use the heat transfer conductance defined (H0) for gap thicknesses less than this value. If $l_{\min} < 0$, then abs($l_{\min}$) is a load curve number defining $l_{\min}$ vs. time.
LMAX	No thermal contact if gap is greater than this value ($l_{\max}$).
CHLM	Is a multiplier used on the element characteristic distance for the search routine. The characteristic length is the largest interface surface element diagonal. EQ.0: Default set to 1.0
BC_FLAG	Thermal boundary condition flag EQ.0: thermal boundary conditions are **on** when parts are in contact EQ.1: thermal boundary conditions are **off** when parts are in contact
ALGO	Contact algorithm type. EQ.0: two way contact, both surfaces change temperature due to contact EQ.1: one way contact, master surface does not change temperature due to contact. Slave surface does change temperature.

Remarks:

In summary:

$h = h_0$, if the gap thickness is $0 \le l_{gap} \le l_{\min}$

$h = h_{cond} + h_{rad}$, if the gap thickness is $l_{\min} \le l_{gap} \le l_{\max}$

$h = 0$, if the gap thickness is $l_{gap} > l_{\max}$

***CONTACT_ …_THERMAL_FRICTION_ …**

WARNING: This is only implemented for the keyword *CONTACT_SURFACE_TO_SURFACE_THERMAL_FRICTION. Check with LSTC for information on other contact types.

This card is required if the FRICTION suffix is added to THERMAL. The keyword *LOAD_SURFACE_STRESS must be used so that LS-DYNA will calculate contact pressure which is needed in the h(P) formulas. The blank (or work piece) must be defined as the slave surface in a metal forming model.

Purpose:

1. Used to define the mechanical static and dynamic friction coefficients as a function of temperature.

2. Used to define the thermal contact conductance as a function of temperature and pressure.

Card 1

1	2	3	4	5	6	7	8
Variable	LCFST	LCFDT	FORMULA	a	b	c	d	LCH
Type	I	I	I	I	I	I	I	I
Default	0	0	0	0	0.	0	0	0

Optional Card (Required if FORMULA is a negative number, use as many cards as necessary to define -FORMULA variables)

Card 1

1	2	3	4	5	6	7	8
Variable	UC1	UC2	UC3	UC4	UC5	UC6	UC7	UC8
Type	F	F	F	F	F	F	F	F
Default	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0

VARIABLE	DESCRIPTION
LCFST	Load curve number for static coefficient of friction as a function of temperature. The load curve value multiplies the coefficient value FS.
LCFDT	Load curve number for dynamic coefficient of friction as a function of temperature. The load curve value multiplies the coefficient value FD.
FORMULA	Formula that defines the contact heat conductance as a function of temperature and pressure.

Note that the following coefficients take on different definitions based on their use.

a	Load curve number for the "a" coefficient used in the formula.
b	Load curve number for the "b" coefficient used in the formula.
c	Load curve number for the "c" coefficient used in the formula.
d	Load curve number for the "d" coefficient used in the formula.
LCH	Load curve number for h. If this is defined, it will take precedence over any other defintions. GT.0: function versus time, GT.nlcur: general function(time,temp,pressure,gap) LT.0: function versus temperature.

Remarks:

FORMULA = 1 $h(P)$ is defined by load curve "a"

"a" defines a load curve for the contact conductance as a function of interface pressure.

FORMULA = 2 $h(P) = a + bP + cP^2 + dP^3$

Although defined by load curves, the coefficients a, b, c, and d are typically constants for use in this formula. The load curves are functions of temperature.

FORMULA = 3 $$h(P) = \frac{\pi k_{gas}}{4\lambda}\left[1. + 85\left(\frac{P}{\sigma}\right)^{0.8}\right] = \frac{a}{b}\left[1. + 85\left(\frac{P}{c}\right)^{0.8}\right]$$

The above formula is from [Shvets and Dyban 1964].

"a" defines a load curve for the thermal conductivity $\left(k_{gas}\right)$ of the gas in the gap as a function of temperature.

"b" defines a load curve for the parameter grouping $\pi/4\lambda$. Therefore, this load curve should be set to a constant value. λ is the surface roughness.

"c" defines a stress metric for deformation (e.g., yield) as a function of temperature.

FORMULA = 4 $$h(P) = a\left[1 - \exp\left(-b\frac{P}{c}\right)\right]^d$$

The above formula is from [Li and Sellars 1996].

"a" defines a load curve as a function of temperature.

"b" defines a load curve as a function of temperature.

"c" defines a stress metric for deformation (e.g., yield) as a function of temperature.

"d" defines a load curve as a function of temperature.

FORMULA = 5 $h(gap)$ is defined by load curve "a"

"a" defines a load curve for the contact conductance as a function of the interface gap.

FORMULA<0 This is equivalent to defining the keyword *USER_INTERFACE_CONDUCTIVITY and the user subroutine `usrhcon` will be called for this contact interface for defining the contact heat transfer coefficient.

Optional Card A

Reminder: If Card 4 is required, then it must go before this optional card. (Card 4 is required for certain contact types - see earlier in this section for the list, later in this section for details of Card 4.)

Optional Card A	1	2	3	4	5	6	7	8
Variable	SOFT	SOFSCL	LCIDAB	MAXPAR	SBOPT	DEPTH	BSORT	FRCFRQ
Type	I	F	I	F	F	I	I	I
Default	0	.1	0	1.025.	0.	2	10-100	1
Remarks			type a13					

VARIABLE	DESCRIPTION
SOFT	Soft constraint option: EQ.0: penalty formulation, EQ.1: soft constraint formulation, EQ.2: segment-based contact. EQ.4: constraint approach for FORMING contact option. The soft constraint may be necessary if the material constants of the elements which make up the surfaces in contact have a wide variation in the elastic bulk moduli. In the soft constraint option, the interface stiffness is based on the nodal mass and the global time step size. This method of computing the interface stiffness will typically give much higher stiffness value than would be obtained by using the bulk modulus; therefore, this method the preferred approach when soft foam materials interact with metals. See the remark below for the segment-based penalty formulation.
SOFSCL	Scale factor for constraint forces of soft constraint option (default=.10). Values greater than .5 for single surface contact and 1.0 for a one-way treatment are inadmissible.
LCIDAB	Load curve ID defining airbag thickness as a function of time for type a13 contact (*CONTACT_AIRBAG_SINGLE_SURFACE).

VARIABLE	DESCRIPTION
MAXPAR	Maximum parametric coordinate in segment search (values 1.025 and 1.20 recommended). Larger values can increase cost. If zero, the default is set to 1.025 for most contact options. Other defaults are: EQ.1.006:SPOTWELD, EQ.1.006:TIED_SHELL_..._CONSTRAINED_OFFSET, EQ.1.006:TIED_SHELL_..._OFFSET, EQ.1.006:TIED_SHELL_..._:BEAM_OFFSET, EQ.1.100:AUTOMATIC_GENERAL This factor allows an increase in the size of the segments which may be useful at sharp corners. For the SPOTWELD and ..._OFFSET options larger values can sometimes lead to numerical instabilities; however, a larger value is sometimes necessary to ensure that all nodes of interest are tied.
SBOPT	Segment-based contact options (SOFT=2). EQ.0: defaults to 2. EQ.1: pinball edge-edge contact (not recommended) EQ.2: assume planer segments (default) EQ.3: warped segment checking EQ.4: sliding option EQ.5: do options 3 and 4
DEPTH	Search depth in automatic contact. Value of 1 is sufficiently accurate for most crash applications and is much less expensive. LS-DYNA for improved accuracy sets this value to 2. If zero, the default is set to 2. LT.0: |DEPTH| is the load curve ID defining searching depth versus time. See remarks below for segment-based contact options controlled by DEPTH.
BSORT	Number of cycles between bucket sorts. Values of 25 and 100 are recommended for contact types 4 and 13 (SINGLE_SURFACE), respectively. Values of 10-15 are okay for the surface to surface and node to surface contact. If zero, LS-DYNA determines the interval. LT.0: |BSORT| load curve ID defining bucket sorting frequency versus time.
FRCFRQ	Number of cycles between contact force updates for penalty contact formulations. This option can provide a significant speed-up of the contact treatment. If used, values exceeding 3 or 4 are dangerous. Considerable care must be exercised when using this option, as this option assumes that contact does not change FRCFRG cycles. EQ.0: FRCFRG is set to 1 and force calculations are performed each cycle-strongly recommended.

Remarks:

Setting SOFT=1 or 2 on optional contact card A will cause the contact stiffness to be determined based on stability considerations, taking into account the time step and nodal masses. This approach is generally more effective for contact between materials of dissimilar stiffness or dissimilar mesh densities.

SOFT=2 is for general shell and solid element contact. This option is available for all SURFACE_TO_SURFACE, ONE_WAY_SURFACE_TO_SURFACE, and SINGLE_ SURFACE options including eroding and airbag contact. When the AUTOMATIC option is used, orientation of shell segment normals is automatic. When the AUTOMATIC option is not used, the segment or element orientations are used as input. The segment-based penalty formulation contact algorithm checks for segments vs. segment penetration rather than node vs. segment. After penetrating segments are found, an automatic judgment is made as to which is the master segment, and penalty forces are applied normal to that segment. The user may override this automatic judgment by using the ONE_WAY options in which case the master segment normals are used as input by the user. All parameters on the first three cards are active except for VC, and VSF. On optional card A, some parameters have different meanings than they do for the default contact.

For SOFT=2, the SBOPT parameter on optional card A controls several options. The pinball edge-to-edge checking is not recommended and is included only for back compatibility. For edge-to-edge checking setting DEPTH=5 is recommended instead (see below). The warped segment option more accurately checks for penetration of warped surfaces. The sliding option uses neighbor segment information to improve sliding behavior. It is primarily useful for preventing segments from incorrectly catching nodes on a sliding surface.

For SOFT=2, the DEPTH parameter controls several additional options for segment based contact. When DEPTH=2 (default), surface penetrations measured at nodes are checked. When DEPTH=3, surface penetration may also be measured at the edge. This option is useful mainly for airbags. When DEPTH=5, both surface penetrations and edge-to-edge penetration is checked. When DEPTH=13, the penetration checking is the same as for DEPTH=3, but the code has been tuned to better conserve energy. When DEPTH=23, the penetration checking is similar to DEPTH=3, but a new algorithm is used to try to improve robustness. The airbag contact has two additional options, DEPTH=1 and 4. DEPTH=4 activates additional airbag logic that uses neighbor segment information when judging if contact is between interior or exterior airbag surfaces. This option is not recommended and is maintained only for backward compatibility. Setting DEPTH=1 suppresses all airbag logic.

For SOFT=2 contact, the MAXPAR has a totally different use. Positive values of MAXPAR are ignored. If a negative value is input for MAXPAR, the absolute value of MAXPAR will be used as an assumed time step for scaling the contact stiffness. This option is useful for maintaining consistent contact behavior of an airbag deployment when a validated airbag is inserted into an automobile model. For the new run, setting MAXPAR=the negative of the solution time step of the validated run will cause the airbag contact stiffness to be unchanged in the new run even if the solution time step of the new run is smaller.

For SOFT=2 contact, only the ISYM, I2D3D, SLDTHK, and SLDSTF parameters are active on optional card B.

Optional Card B

Reminder: If Optional Card B is used, then Optional Card A must be defined. (Optional Card A may be a blank line).

Optional Card B	1	2	3	4	5	6	7	8
Variable	PENMAX	THKOPT	SHLTHK	SNLOG	ISYM	I2D3D	SLDTHK	SLDSTF
Type	F	I	I	I	I	I	F	F
Default	0	0	0	0	0	0	0	0
Remarks		Old types 3, 5, 10	Old types 3, 5, 10					

VARIABLE **DESCRIPTION**

PENMAX — Maximum penetration distance for old type 3, 5, 8, 9, and 10 contact or the segment thickness multiplied by PENMAX defines the maximum penetration allowed (as a multiple of the segment thickness) for contact types a 3, a 5, a10, 13, 15, and 26. (see discussion at end of section, including Table 7.1):

EQ.0.0 for old type contacts 3, 5, and 10: Use small penetration search and value calculated from thickness and XPENE, see *CONTROL_CONTACT.

EQ.0.0 for contact types a 3, a 5, a10, 13, and 15: Default is 0.4, or 40 percent of the segment thickness

EQ.0.0 for contact type26: Default is 200.0 times the segment thickness

THKOPT — Thickness option for contact types 3, 5, and 10:

EQ.0: default is taken from control card, *CONTROL_CONTACT,

EQ.1: thickness offsets are included,

EQ.2: thickness offsets are not included (old way).

SHLTHK — Define if and only if THKOPT above equals 1. Shell thickness considered in type surface to surface and node to surface type contact options, where options 1 and 2 below activate the new contact algorithms. The thickness offsets are always included in single surface and constraint method contact types:

VARIABLE	DESCRIPTION
	EQ.0: thickness is not considered, EQ.1: thickness is considered but rigid bodies are excluded, EQ.2: thickness is considered including rigid bodies.
SNLOG	Disable shooting node logic in thickness offset contact. With the shooting node logic enabled, the first cycle that a slave node penetrates a master segment, that node is moved back to the master surface without applying any contact force. EQ.0: logic is enabled (default), EQ.1: logic is skipped (sometimes recommended for metalforming calculations or for contact involving foam materials).
ISYM	Symmetry plane option: EQ.0: off, EQ.1: do not include faces with normal boundary constraints (e.g., segments of brick elements on a symmetry plane). This option is important to retain the correct boundary conditions in the model with symmetry. For the _ERODING_ contacts this option may also be defined on card 4.
I2D3D	Segment searching option: EQ.0: search 2D elements (shells) before 3D elements (solids, thick shells) when locating segments. EQ.1: search 3D (solids, thick shells) elements before 2D elements (shells) when locating segments.
SLDTHK	Optional solid element thickness. A nonzero positive value will activate the contact thickness offsets in the contact algorithms where offsets apply. The contact treatment will then be equivalent to the case where null shell elements are used to cover the brick elements. The contact stiffness parameter below, SLDSTF, may also be used to override the default value.
SLDSTF	Optional solid element stiffness. A nonzero positive value overrides the bulk modulus taken from the material model referenced by the solid element.

Optional Card C

Reminder: If Optional Card C is used, then Optional Cards A and B must be defined. (Optional Cards A and B may be blank lines).

Optional Card C	1	2	3	4	5	6	7	8
Variable	IGAP	IGNORE	DPRFAC	DTSTIF	FNLSCL	DNLSCL	FLANGL	CID_RCF
Type	I	I	F	F	F	F	F	I
Default	1	0	0	0	0	0	0	0
Remarks			1	2	3	3		

VARIABLE	DESCRIPTION
IGAP	Flag to improve implicit convergence behavior at the expense of creating some sticking if parts attempt to separate. Outputted contact forces may also be too small or zero. (IMPLICIT ONLY) EQ.1: apply method to improve convergence (DEFAULT) EQ.2: do not apply method
IGNORE	Ignore initial penetrations in the *CONTACT_AUTOMATIC options. EQ.0: Take the default value from the fourth card of the CONTROL_CONTACT input. EQ.1: Allow initial penetrations to exist by tracking the initial penetrations. EQ.2: Allow initial penetrations to exist by tracking the initial penetrations. However, penetration warning messages are printed with the original coordinates and the recommended coordinates of each slave node given.
DPRFAC	Depth of penetration reduction factor for SOFT=2 contact. EQ.0.0: Initial penetrations are always ignored. GT.0.0: Initial penetrations are penalized over time. LE.1.0: |DPRFAC| is the load curve ID defining DPRFAC versus time.
DTSTIF	Time step used in stiffness calculation for SOFT=1 and SOFT=2 contact. EQ.0.0: Use the initial value that is used for time integration. GT.0.0: Use the value specified. LT.-0.01 and GT. -1.0: use a moving average of the solution time step. (SOFT=2 only) LE.-1.0 |DTSTIF| is the ID of a curve that defines DTSTIF vs. time.

VARIABLE	DESCRIPTION
FNLSCL	Scale factor for nonlinear force scaling.
DNLSCL	Distance for nonlinear force scaling.
FLANGL	Angle tolerance in radians for feature lines option in smooth contact. EQ.0.0: No feature line is considered for surface fitting in smooth contact. GT.0.0: Any edge with angle between two contact segments bigger than this angle will be treated as feature line during surface fitting in smooth contact.
CID_RCF	Coordinate system ID to output RCFORC force resultants in a local system.

Remarks:

1. DPRFAC is used only by segment based contact (SOFT=2). By default, SOFT=2 contact measures the initial penetration between segment pairs that are found to be in contact and subtracts the measured value from the total penetration for as long as a pair of segments remains in contact. The penalty force is proportional to this modified value. This approach prevents shooting nodes, but may allow unacceptable penetration. DPRFAC can be used to decrease the measured value over time until the full penetration is penalized. Setting DPRFAC=0.01 will cause ~1% reduction in the measured value each cycle. The maximum allowable value for DPRFAC is 0.1. A small value such as 0.001 is recommended. DPRFAC does not apply to initial penetrations at the start of the calculation, only those that are measured at later times. This prevents nonphysical movement and energy growth at the start of the calculation.

   The anticipated use for the load curve option is to allow the initial penetrations to be reduced at the end of a calculation if the final geometry is to be used for a subsequent analysis. To achieve this, load curve should have a y-value of zero until a time near the end of the analysis and then ramp up to a positive value such as 0.01 near the end of the analysis.

2. DTSTIF is used only by the SOFT=1 and SOFT=2 contact options. By default when the SOFT option is active, the contact uses the initial solution time step to scale the contact stiffness. If the user sets DTSIFF larger than the initial solution time step, the larger value will be used. A smaller value will be ignored. Because the square of the time step appears in the denominator of the stiffness calculation, a larger value reduces the contact stiffness. This option could be used when one component of a larger model has been analyzed independently and validated. When the component is inserted into the larger model, the larger model may run at a smaller time step due to higher mesh frequencies. In the full model analysis, setting DTSTIF equal to the component analysis time step for

the contact interface that treats the component will cause consistent contact stiffness between the analyses.

The load curve option allows a variable contact stiffness. This should be done with care as energy will not be conserved. A special case of the load curve option is when |DTSTIF| = LCTM on *CONTROL_CONTACT. LCTM sets an upper bound on the solution time step. For |DTSTIF| = LCTM, the contact stiffness time step value will track LCTM whenever the LCTM value is less than the initial solution time step. If the LCTM value is greater, the initial solution time step is used. This option could be used to stiffen the contact at the end of an analysis. To achieve this, the LCTM curve should be defined such that it is larger than the solution time step until near the end of the analysis. Then the LCTM curve should ramp down below the solution time step causing it to decrease and the contact to stiffen. A load curve value of 0.1 of the calculated solution time step will cause penetrations to reduce by about 99%. To prevent shooting nodes, the rate at which the contact stiffness increases is automatically limited. Therefore, to achieve 99% reduction, the solution should be run for perhaps 1000 cycles with a small time step.

For segment based contact (SOFT=2), setting DTSTIF less than or equal to -0.01 and greater than -1.0, causes the contact stiffness will be be updated based on the current solution time step. Varying the contact stiffness during a simulation can cause energy growth so this option should be used with care when extra stiffness is needed to prevent penetration and the solution time step has dropped from the initial Because quick changes in contact stiffness can cause shooting nodes, using a moving average of the solution time step can prevent this. The value of DTSTIF determines the number of terms in the moving average where n=100*(-DTSTIF) such that n=1 for DTSTIF=-0.01 and n=100 for DTSTIF=-0.999. Setting DTSTIF=-1.0 triggers the load curve option described in the previous paragraph, so DTSIF cannot be smaller than -0.999 for this option.

3. FNLSCL=*f* and DNLSCL=*d* scale the contact stiffness nonlinearly to provide smoother initial contact and a larger contact force as the depth of penetration exceeds DNLSCL. The stiffness *k* is scaled by the relation $k = k \bullet . f\sqrt{\delta / d}$ where δ is the depth of penetration, making the penalty force proportional to the 3/2 power of the penetration depth. Adding a small amount of surface damping (e.g., VDC=10) is advised with this option.

Optional Card D

Reminder: If Optional Card D is used, then Optional Cards A, B and C must be defined. (Optional Cards A, B and C may be blank lines).

Optional Card D	1	2	3	4	5	6	7	8
Variable	Q2TRI	DTPCHK	SFNBR				TIEDID	
Type	I	F	F				I	
Default	0	0	0				0	
Remarks	1	2	3				4	

VARIABLE **DESCRIPTION**

Q2TRI — Option to split quadrilateral contact segments into two triangles (only available when SOFT=2).
- EQ.0: Off (default).
- EQ.1: On for all slave shell segments.
- EQ.2: On for all master shell segments.
- EQ.3: On for all shell segments.
- EQ.4: On for all shell segments of material type 34.

DTPCHK — Time interval between shell penetration reports (only available for segment based contact)
- EQ.0.0: Off (default).
- GT.0.0: Check and report segment penetrations at time intervals equal to DTPCHK

SFNBR — Scale factor for neighbor segment contact (only available for segment based contact)
- EQ.0.0: Off (default).
- GT.0.0: Check neighbor segments for contact

TIEDID — Incremental displacement update for tied contacts. (See remark 4 below.)
- EQ.0: Off (default).
- EQ.1: On

Remarks:

1 Setting Q2TRI to a nonzero value causes quadrilateral shell segments to be spilt into two triangles. The contact segments only are split. The elements are not changed. This option is only available for segment based contact which is activated by setting SOFT=2.

2 Setting DTPCHK to a positive value causes a penetration check to be done periodically with the interval equal to DTPCHK. The check looks for shell segments that are penetrating the mid-plane of another shell segment. It does not report on penetration of thickness offsets. The penetrating pairs are reported to the messag file or files for MPP. If at least one penetration is found, the total number of pairs is reported to the screen output. This option is only available for segment based contact which is activated by setting SOFT=2.

3 Setting SFNBR to a positive value turns on neighbor segment contact checking in segment based contact. This is helpful when a mesh folds as can happen with compression folding of an airbag. Only shell element segments are checked.

4 There have been several issues with tied OFFSET contacts and AUTOMATIC_ TIEBREAK contacts with offsets creating numerical round-off noise in stationary parts. By computing the interface displacements incrementally rather than using total displacements, the round-off errors that occur in single precision are eliminated. The incremental approach is available for the following contact types:

TIED_SURFACE_TO_SURFACE_OFFSET
TIED_SURFACE_TO_SURFACE_PENALTY
TIED_NODES_TO_SURFACE_OFFSET
TIED_NODES_TO_SURFACE_PENALTY
TIED_SHELL_EDGE_TO_SURFACE_PENALTY
AUTOMATIC_TIEBREAK

General Remarks on *CONTACT:

1. Modeling airbag interactions with structures and occupants using the actual fabric thickness, which is approximate 0.30 mm, may result in a contact breakdown that leads to inconsistent occupant behavior between different machines. Based on our experience, using a two-way automatic type contact definition, i.e., AUTOMATIC_SURFACE_TO_SURFACE, between any airbag to structure/occupant interaction and setting the airbag fabric contact thickness to at least 10 times the actual fabric thickness has helped improved contact behavior and eliminates the machine inconsistencies. Due to a large stiffness difference between the airbag and the interacting materials, the soft constraint option (SOFT=1) or the segment based pinball option (SOFT=2) is recommended. It must be noted that with the above contact definition, only the airbag materials should be included in any *AIRBAG_SINGLE_SURFACE definitions to avoid duplicate contact treatment that can lead to numerical instabilities.

2. TIED_NODES_TO_SURFACE
TIED_SHELL_EDGE_TO_SURFACE
TIED_SHELL_EDGE_TO_SURFACE_CONSTRAINED_OFFSET
SPOTWELD
SPOTWELD_WITH_TORSION
TIED_SURFACE_TO_SURFACE

   These contact definitions are based on constraint equations and will not work with rigid bodies. However, tied interfaces with the offset option can be used with rigid bodies, i.e.,

   TIED_NODES_TO_SURFACE_OFFSET
TIED_SHELL_EDGE_TO_SURFACE_OFFSET
TIED_SHELL_EDGE_TO_SURFACE_BEAM_OFFSET
TIED_SURFACE_TO_SURFACE_OFFSET

   Also, it may sometimes be advantageous to use the CONSTRAINED_EXTRA_NODE_*OPTION* instead for tying deformable nodes to rigid bodies since in this latter case the tied nodes may be an arbitrary distance away from the rigid body.

   Tying will only work if the surfaces are near each other. The criteria used to determine whether a slave node is tied down is that it must be “close”. For shell elements “close” is defined as distance, δ, less than:

$$\delta_1 = 0.60 * (thickness\_slave\_node + thickness\_master\_segment)$$

$$\delta_2 = 0.05 * \min(master\_segment\_diagonals)$$

$$\delta = \max(\delta_1, \delta_2)$$

   If a node is further away it will not be tied and a warning message will be printed. For solid elements the slave node thickness is zero and the segment thickness is the element volume divided by the segment area; otherwise, the same procedure is used.

If there is a large difference in element areas between the master and slave side, the distance, δ_2, may be too large and may cause the unexpected projection of nodes that should not be tied. This can occur during calculation when adaptive remeshing is used. To avoid this difficulty the slave and master thickness can be specified as negative values on Card 3 in which case

$$\delta = abs(\delta_1)$$

3. The contact algorithm for tying spot welds with torsion, SPOTWELD_WITH_TORSION, must be used with care. Parts that are tied by this option should be subjected to stiffness proportional damping of approximately ten percent, i.e., input a coefficient of 0.10. This can be defined for each part on the *DAMPING_PART_STIFFNESS input. Stability problems may arise with this option if damping is not used.

4. CONSTRAINT_NODES_TO_SURFACE
CONSTRAINT_SURFACE_TO_SURFACE

   These contact definitions must be used with care. The surface and the nodes which are constrained to a surface are not allowed to be used in any other CONSTRAINT_... contact definition. If, however, contact has to be defined from both sides as in sheet metalforming, one of these contact definitions can be a CONSTRAINT_ type; the other one could be a standard penalty type such as SURFACE_TO_SURFACE or NODES_TO_SURFACE.

5. AIRBAG_SINGLE_SURFACE
AUTOMATIC_GENERAL
AUTOMATIC_GENERAL_INTERIOR
AUTOMATIC_NODES_TO_SURFACE
AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE
AUTOMATIC_SINGLE_SURFACE
AUTOMATIC_SURFACE_TO_SURFACE
SINGLE_SURFACE

   These contact definitions require thickness to be taken into account for rigid bodies modeled with shell elements. Therefore, care should be taken to ensure that realistic thicknesses are specified for the rigid body shells. A thickness that is too small may result in loss of contact and an unrealistically large thickness may result in a degradation in speed during the bucket sorts as well as nonphysical behavior. The SHLTHK option on the *CONTROL_CONTACT card is ignored for these contact types.

6. Two methods are used in LS-DYNA for projecting the contact surface to account for shell thicknesses. The choice of methods can influence the accuracy and cost of the calculation. Segment based projection is used in contact types:

   AIRBAG_SINGLE_SURFACE
AUTOMATIC_GENERAL
AUTOMATIC_NODES_TO_SURFACE
AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE

AUTOMATIC_SINGLE_SURFACE
AUTOMATIC_SURFACE_TO_SURFACE
FORMING_NODES_TO_SURFACE
FORMING_ONE_WAY_SURFACE_TO_SURFACE
FORMING_SURFACE_TO_SURFACE

The remaining contact types use nodal normal projections if projections are used. The main advantage of nodal projections is that a continuous contact surface is obtained which is much more accurate in applications such as metal forming. The disadvantages of nodal projections are the higher costs due to the nodal normal calculations, difficulties in treating T-intersections and other geometric complications, and the need for consistent orientation of contact surface segments. The contact type:

SINGLE_SURFACE

uses nodal normal projections and consequently is slower than the alternatives.

7. FORCE_TRANSDUCER_PENALTY
FORCE_TRANSDUCER_CONSTRAINT

This contact allows the total contact forces applied by all contacts to be picked up. This contact does not apply any force to the model and will have no effect on the solution. Only the slave set and slave set type need be defined for this contact type. Generally, only the first three cards are defined. The force transducer option, _PENALTY, works with penalty type contact algorithms only, i.e., it does not work with the CONSTRAINT or TIED options. For these latter options, use the _CONSTRAINT option. *If the interactions between two surfaces are needed, a master surface should be defined. In this case, only the contact forces applied between the slave and master surfaces are kept. The master surface option is only implemented for the _PENALTY option and works only with the AUTOMATIC contact types.*

8. FORMING_... These contacts are mainly used for metal forming applications. A connected mesh is not required for the master (tooling) side but the orientation of the mesh **must** be in the same direction. These contact types are based on the AUTOMATIC type contacts and consequently the performance is better than the original two surface contacts.
9. The mortar contact, invoked by appending the suffix MORTAR to either FORMING_SURFACE_TO_SURFACE, AUTOMATIC_SURFACE_TO_SURFACE or AUTOMATIC_SINGLE_SURFACE is a segment to segment penalty based contact. For two segments on each side of the contact interface that are overlapping and penetrating, a consistent nodal force assembly taking into account the individual shape functions of the segments is performed, see Figure 7.3a for an illustration. In this respect the results with this contact may be more accurate, especially when considering contact with elements of higher order. By appending the suffix TIED to the CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_MORTAR keyword, this is treated as a tied contact interface. This contact is intended for implicit analysis in particular but is nevertheless supported for explicit analysis as well.

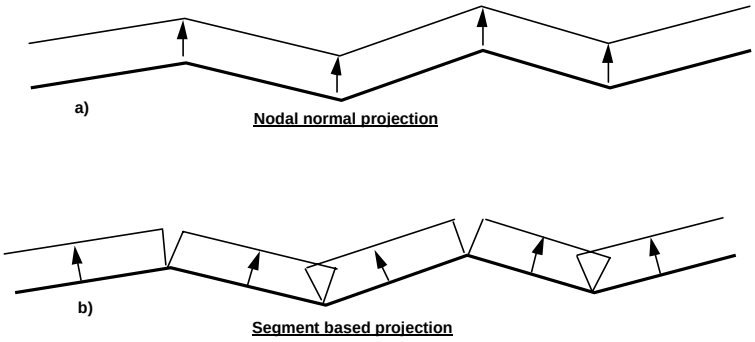


Figure 7.3. Nodal normal and segment based projection is used in the contact options.

INTERFACE TYPE ID	PENCHK	ELEMENT TYPE	FORMULA FOR RELEASE OF PENETRATING NODAL POINT
1, 2, 6, 7	———	———	———————----------
3, 5, 8, 9, 10	0	solid	d=PENMAX if and only if PENMAX>0
(without thickness)			d=1.e+10 if PENMAX=0
		shell	d=PENMAX if and only if PENMAX>0
			d=1.e+10 if PENMAX=0
	1	solid	d=XPENE*thickness of solid element
		shell	d=XPENE*thickness of shell element
	2	solid	d=0.05*minimum diagonal length
		shell	d=0.05*minimum diagonal length
3, 5, 10 (thickness)	———	solid	d=XPENE*thickness of solid element
17, and 18		shell	d=XPENE*thickness of shell element
a3, a5, a10, 13, 15	———	solid	d=PENMAX*thickness of solid element
			[default: PENMAX=0.5]
		shell	d=PENMAX*(slave thickness+master thickness) [default: PENMAX=0.4]
4	———	solid	d=0.5*thickness of solid element
		shell	d=0.4*(slave thickness+master thickness)
26	———	solid	d=PENMAX*thickness of solid element
			[default: PENMAX=200.0]
		shell	d=PENMAX*(slave thickness+master thickness) [default: PENMAX=200.]

Table 7.1. Criterion for node release for nodal points which have penetrated too far. Larger penalty stiffnesses are recommended for the contact interface which allows nodes to be released. For node-to-surface type contacts (5, 5a) the element thicknesses which contain the node determines the nodal thickness. The parameter is defined on the *CONTROL_CONTACT input.

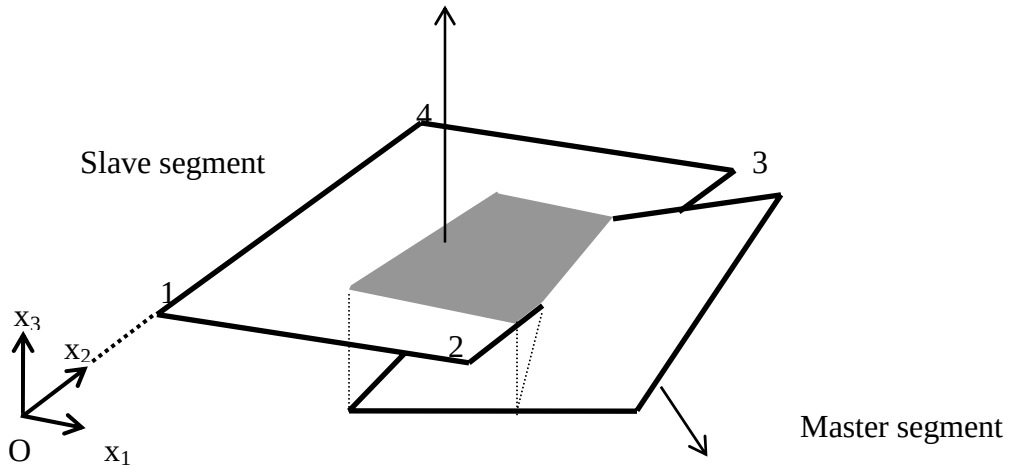


Figure 7.3a . Illustration of Mortar segment to segment contact

The keyword options for the contact type and the corresponding Version 92X, 93X, 94X, 95X type numbers are:

STRUCTURED INPUT TYPE ID	KEYWORD NAME
a13	AIRBAG_SINGLE_SURFACE
26	AUTOMATIC_GENERAL
i26	AUTOMATIC_GENERAL_INTERIOR
a 5	AUTOMATIC_NODES_TO_SURFACE
a 5	AUTOMATIC_NODES_TO_SURFACE_TIEBREAK
a10	AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE
13	AUTOMATIC_SINGLE_SURFACE
a 3	AUTOMATIC_SURFACE_TO_SURFACE
a 3	AUTOMATIC_SURFACE_TO_SURFACE_TIEBREAK
18	CONSTRAINT_NODES_TO_SURFACE
17	CONSTRAINT_SURFACE_TO_SURFACE
23	DRAWBEAD
16	ERODING_NODES_TO_SURFACE
14	ERODING_SURFACE_TO_SURFACE
15	ERODING_SINGLE_SURFACE
27	FORCE_TRANSDUCER_CONSTRAINT
25	FORCE_TRANSDUCER_PENALTY
m 5	FORMING_NODES_TO_SURFACE
m10	FORMING_ONE_WAY_SURFACE_TO_SURFACE
m 3	FORMING_SURFACE_TO_SURFACE
5	NODES_TO_SURFACE
5	NODES_TO_SURFACE_INTERFERENCE
10	ONE_WAY_SURFACE_TO_SURFACE
20	RIGID_NODES_TO_RIGID_BODY
21	RIGID_BODY_ONE_WAY_TO_RIGID_BODY
19	RIGID_BODY_TWO_WAY_TO_RIGID_BODY
22	SINGLE_EDGE
4	SINGLE_SURFACE
1	SLIDING_ONLY
p 1	SLIDING_ONLY_PENALTY
3	SURFACE_TO_SURFACE
3	SURFACE_TO_SURFACE_INTERFERENCE

STRUCTURED INPUT TYPE ID	KEYWORD NAME
8	TIEBREAK_NODES_TO_SURFACE
9	TIEBREAK_SURFACE_TO_SURFACE
6	TIED_NODES_TO_SURFACE
o 6	TIED_NODES_TO_SURFACE_OFFSET
7	TIED_SHELL_EDGE_TO_SURFACE
7	SPOTWELD
s 7	SPOTWELD_WITH_TORSION
2	TIED_SURFACE_TO_SURFACE
o 2	TIED_SURFACE_TO_SURFACE_OFFSET

CONTACT EXAMPLES

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$   *CONTACT_NODES_TO_SURFACE
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Make a simple contact that prevents the nodes in part 2 from
$  penetrating the segments in part 3.
$
*CONTACT_NODES_TO_SURFACE
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$     ssid      msid     sstyp     mstyp    sboxid    mboxid       spr       mpr
         2         3         3         3
$
$       fs        fd        dc        vc       vdc    penchk        bt        dt

$
$      sfs       sfm       sst       mst      sfst      sfmt       fsf       vsf

$
$    sstype, mstype = 3  id's specified in ssid and msid are parts
$      ssid = 2  use slave nodes in part 2
$      msid = 3  use master segments in part 3
$
$  Use defaults for all parameters.
$
$$$$  Optional Cards A and B not specified (default values will be used).
$
$
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$   *CONTACT_SINGLE_SURFACE
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Create a single surface contact between four parts: 28, 97, 88 and 92
$     - create a part set with set ID = 5, list the four parts
$     - in the *CONTACT_SINGLE_SURFACE definition specify:
$          sstyp = 2  which means the value for ssid is a part set
$          ssid  = 5  use part set 5 for defining the contact surfaces
$
$  Additional contact specifications described below.
$
*CONTACT_SINGLE_SURFACE
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$     ssid      msid     sstyp     mstyp    sboxid    mboxid       spr       mpr
         5                   2
$       fs        fd        dc        vc       vdc    penchk        bt        dt
      0.08      0.05        10                  20                           40.0
$      sfs       sfm       sst       mst      sfst      sfmt       fsf       vsf

$
$       fs = 0.08  static coefficient of friction equals 0.08
$       fd = 0.05  dynamic coefficient of friction equals 0.05
$       dc =   10  exponential decay coefficient, helps specify the transition
$                     from a static slide to a very dynamic slide
$      vdc =   20  viscous damping of 20% critical (damps out nodal
$                     oscillations due to the contact)
$       dt = 40.0  contact will deactivate at 40 ms (assuming time unit is ms)
$
$$$$  Optional Cards A and B not specified (default values will be used).
$
$
*SET_PART_LIST
$      sid
         5
$     pid1      pid2      pid3      pid4
        28        97        88        92
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$   *CONTACT_DRAWBEAD
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Define a draw bead contact:
$    - the draw bead is to be made from the nodes specified in node set 2
$    - the master segments are to be those found in the box defined by box 2
$        that are in part 18
$    - include slave and master forces in interface file (spr, mpr = 1)
$
*CONTACT_DRAWBEAD
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$     ssid      msid     sstyp     mstyp    sboxid    mboxid       spr       mpr
         2        18         4         3                 2         1         1
$
$       fs        fd        dc        vc       vdc    penchk        bt        dt
      0.10
$
$      sfs       sfm       sst       mst      sfst      sfmt       fsf       vsf

$
$$$$ Card 4 required because it's a drawbead contact
$
$  lcdidrf    lcidnf     dbdth     dfscl    numint
         3             0.17436       2.0
$
$  lcdidrf =       3  load curve 3 specifies the bending component of the
$                       restraining force per unit draw bead length
$    dbdth = 0.17436  draw bead depth
$    dfscl =     2.0  scale load curve 3 (lcdidrf) by 2
$
$$$$  Optional Cards A and B not specified (default values will be used).
$
$
*DEFINE_BOX
$    boxid       xmn       xmx       ymn       ymx       zmn       zmx
         2 0.000E+00 6.000E+00 6.000E+00 1.000E+02-1.000E+03 1.000E+03
$
*SET_NODE_LIST
$      sid       da1       da2       da3       da4
         2
$     nid1      nid2      nid3      nid4      nid5      nid6      nid7      nid8
      2580      2581      2582      2583      2584      2585      2586      2587
      2588      2589      2590
$
*DEFINE_CURVE
$     lcid      sidr      scla      sclo      offa      offo
         3
$                a                 o
$            DEPTH          FORC/LGTH
          0.000E+00          0.000E+00
          1.200E-01          1.300E+02
          1.500E-01          2.000E+02
          1.800E-01          5.000E+02
```

*CONTACT_AUTO_MOVE

Purpose: Move the master surface in a contact definition to close an initial gap between the slave and master surfaces. The contact surfaces will then start in contact thereby saving calculational cost. The master surface in metalforming applications will typically be the punch and the blank will be the slave surface.

Define one card. Card Format (I0)

Cards 1	1	2	3	4	5	6	7	8
Variable	ID	CID	VID	LCID	ATIME			
Type	I	I	I	I	F			
Default	required	required	required	0	0.0			

VARIABLE	DESCRIPTION
ID	ID for this auto positioning input.
CID	Contact ID.
VID	Vector ID for a vector oriented in the direction of the movement of the master surface. See *DEFINE_VECTOR. The origin of this vector is unimportant since the direction cosines of the vector are computed and used.
LCID	Optional load curve ID defining velocity versus time. The load curve will be adjusted to account for the movement of the master surface. The load curve should be defined by four points, and its shape should resemble a trapezoid with the longest parallel side along the abscissa. The abscissa is adjusted (shortened) in the flat part of the curve where the velocity is constant to account for the movement.
ATIME	Activation time. At this time the master surface is moved.

*CONTACT_COUPLING

Purpose: Define a coupling surface for MADYMO to couple LS-DYNA with deformable and rigid parts within MADYMO. In this interface, MADYMO computes the contact forces acting on the coupling surface, and LS-DYNA uses these forces in the update of the motion of the coupling surface for the next time step. Contact coupling can be used with other coupling options in LS-DYNA.

Card 1	1	2	3	4	5	6	7	8
Variable	ID							
Type	I							
Default	required							

Cards 2, 3, 4, ... Define as cards as necessary. The next "*" card terminates this input.

Cards 2,3,..	1	2	3	4	5	6	7	8
Variable	SID	STYPE						
Type	I	I						
Default	required	0						

VARIABLE	DESCRIPTION
SID	Set ID for coupling. See Remark 1 below.
STYPE	Set type: EQ.0: part set EQ.1: shell element set EQ.2: solid element set EQ.3: thick shell element set

Remarks:

1. Only one coupling surface can be defined. If additional surfaces are defined, the coupling information will be added to the first definition.

2. The units and orientation can be converted by using the CONTROL_COUPLING keyword. It is not necessary to use the same system of units in MADYMO and in LS-DYNA if unit conversion factors are defined.

*CONTACT_ENTITY

Purpose: Define a contact entity. Geometric contact entities treat the impact between a deformable body defined as a set of slave nodes or nodes in a shell part set and a rigid body. The shape of the rigid body is determined by attaching geometric entities. Contact is treated between these geometric entities and the slave nodes using a penalty formulation. The penalty stiffness is optionally maximized within the constraint of the Courant criterion. As an alternative, a finite element mesh made with shells can be used as geometric entity. Also, axisymmetric entities with arbitrary shape made with multi-linear polygons are possible. The latter is particularly useful for metalforming simulations.

WARNING: If the problem being simulated involves dynamic motion of the entity, care should be taken to insure that the inertial properties of the entity are correct. It may be necessary to use the *PART_INERTIA option to specify these properties.

Define 5 cards for the contact entity definition below.

Card 1	1	2	3	4	5	6	7	8
Variable	PID	GEOTYP	SSID	SSTYP	SF	DF	CF	INTORD
Type	I	I	I	I	F	F	F	I
Default	required	required	required	0	1.	0.	0.	0

VARIABLE	DESCRIPTION
PID	Part ID of the rigid body to which the geometric entity is attached, see *PART.
GEOTYP	Type of geometric entity: EQ.1: plane, EQ.2: sphere, EQ.3: cylinder, EQ.4: ellipsoid, EQ.5: torus, EQ.6: CAL3D/MADYMO Plane, see Appendix I, EQ.7: CAL3D/MADYMO Ellipsoid, see Appendix I, EQ.8: VDA surface, see Appendix L, EQ.9: rigid body finite element mesh (shells only), EQ.10: finite plane, EQ.11: load curve defining line as surface profile of axisymmetric rigid bodies.

VARIABLE	DESCRIPTION
SSID	Slave set ID, see *SET_NODE_*OPTION*, *PART, or *SET_PART.
SSTYP	Slave set type: EQ.0: node set, EQ.1: part ID, EQ.2: part set ID.
SF	Penalty scale factor. Useful to scale maximized penalty.
DF	Damping option, see description for *CONTACT_*OPTION*: EQ.0: no damping, GT.0: viscous damping in percent of critical, e.g., 20 for 20% damping, EQ.-n: |n| is the load curve ID giving the damping force versus relative normal velocity (see remark 1 below).
CF	Coulomb friction coefficient. Assumed to be constant.
INTORD	Integration order (slaved materials only). This option is not available with entity types 8 and 9 where only nodes are checked: EQ.0: check nodes only, EQ.1: 1 point integration over segments, EQ.2: 2×2 integration, EQ.3: 3×3 integration, EQ.4: 4×4 integration, EQ.5: 5×5 integration. This option allows a check of the penetration of the rigid body into the deformable (slaved) material. Then virtual nodes at the location of the integration points are checked.

Remarks:

1. The optional load curves that are defined for damping versus relative normal velocity and for force versus normal penetration should be defined in the positive quadrant. The sign for the damping force depends on the direction of the relative velocity and the treatment is symmetric if the damping curve is in the positive quadrant. If the damping force is defined in the negative and positive quadrants, the sign of the relative velocity is used in the table look-up.

Card 2	1	2	3	4	5	6	7	8
Variable	BT	DT	SO	GO				
Type	F	F	I	I				
Default	0.	1.E+20	0	0				

VARIABLE	DESCRIPTION
BT	Birth time
DT	Death time
SO	Flag to use penalty stiffness as in surface-to-surface contact: EQ.0: contact entity stiffness formulation, EQ.1: surface to surface contact method, EQ.-n: |n| is the load curve ID giving the force versus the normal penetration.
GO	Flag for mesh generation of the contact entity for entity types 1-5 and 10-11. This is used for visualization in post-processing only. EQ.0: mesh is not generated, EQ.1: mesh is generated.

Card 3	1	2	3	4	5	6	7	8
Variable	XC	YC	ZC	AX	AY	AZ		
Type	F	F	F	F	F	F		
Default	0.	0.	0.	0.	0.	0		

Card 4

Variable	BX	BY	BZ					
Type	F	F	F					
Default	0.	0.	0.					

VARIABLE	DESCRIPTION
XC	x-center, x_c, see remarks below.
YC	y-center, y_c, see remarks below.
ZC	z-center, z_c. See remarks below.
AX	x-direction for local axis A, A_x, see remarks below.
AY	y-direction for local axis A, A_y, see remarks below.
AZ	z-direction for local axis A, A_z, see remarks below.
BX	x-direction for local axis B, B_x, see remarks below.
BY	y-direction for local axis B, B_y, see remarks below.
BZ	z-direction for local axis B, B_z, see remarks below.

Remarks:

1. The coordinates, (x_c, y_c, z_c), are the positions of the local origin of the geometric entity in global coordinates. The entity's local A-axis is determined by the vector (A_x, A_y, A_z) and the local B-axis by the vector (B_x, B_y, B_z).

2. Cards 3 and 4 define a local to global transformation. The geometric contact entities are defined in a local system and transformed into the global system. For the ellipsoid, this is necessary because it has a restricted definition for the local position. For the plane, sphere, and cylinder, the entities can be defined in the global system and the transformation becomes $(x_c, y_c, z_c)=(0,0,0)$, $(A_x, A_y, A_z)=(1,0,0)$, and $(B_x, B_y, B_z)=(0,1,0)$.

Card 5	1	2	3	4	5	6	7	8
Variable	INOUT	G1	G2	G3	G4	G5	G6	G7
Type	I	F	F	F	F	F	F	F
Default	0	0.	0.	0.	0.	0.	0.	0.

VARIABLE	DESCRIPTION
INOUT	In-out flag. Allows contact from the inside or the outside (default) of the entity: EQ.0: slave nodes exist outside of the entity, EQ.1: slave nodes exist inside the entity.
G1	Entity coefficient g_1 (CAL3D/MADYMO plane or ellipse number) for coupled analysis (see Appendix I).
G2	Entity coefficient g_2, see remarks below.
G3	Entity coefficient g_3, see remarks below.
G4	Entity coefficient g_4, see remarks below.
G5	Entity coefficient g_5, see remarks below.
G6	Entity coefficient g_6, see remarks below.
G7	Entity coefficient g_7, see remarks below.

Remarks:

Figures 7.4a and 7.4b show the definitions of the geometric contact entities. The relationships between the entity coefficients and the Figure 7.4a and 7.4b variables are as follows (please note that (P_x,P_y,P_z) is a position vector and that (Q_x,Q_y,Q_z) is a direction vector):

GEOTYP = 1: g1 = Px, g2 = Py, g3 = Pz, g4 = Qx, g5 = Qy, g6 = Qz, g7 = L

If automatic generation is used, a square plane of length L on each edge is generated which represents the infinite plane. If generation is inactive, then g7 may be ignored.

GEOTYP = 2: g1 = Px, g2 = Py, g3 = Pz, g4 = r

GEOTYP = 3: g1 = Px, g2 = Py, g3 = Pz, g4 = Qx, g5 = Qy, g6 = Qz, g7 = r

If automatic generation is used, a cylinder of length $\sqrt{Qx^2 + Qy^2 + Qz^2}$ and radius r is generated which represents the infinite cylinder.

GEOTYP = 4: g1 = Px, g2 = Py, g3 = Pz, g4 = a, g5 = b, g6 = c, g7 = *n* (order of the ellipsoid)

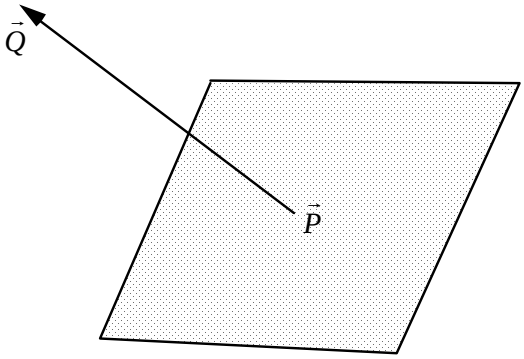
GEOTYP = 5:
g1 = Radius of torus
g2 = r
g3 = number of elements along minor circumference
g4 = number of elements along major circumference

GEOTYP = 8:
g1 = Blank thickness (option to override true thickness)
g2 = Scale factor for true thickness (optional)
g3 = Load curve ID defining thickness versus time. (optional)

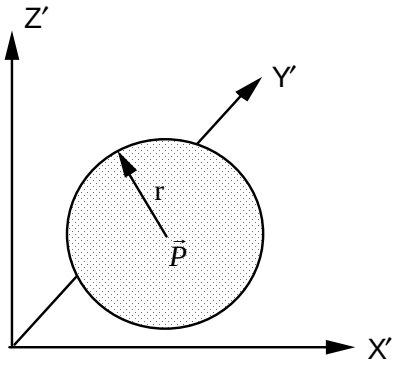
GEOTYP = 9:
g1 = Shell thickness (option to override true thickness).
NOTE: **The shell thickness specification is necessary if the slave surface is generated from solid elements.**
g2 = Scale factor for true thickness (optional)
g3 = Load curve ID defining thickness versus time. (optional)

GEOTYP =10:
g1 = Length of edge along X′ axis
g2 = Length of edge along Y′ axis

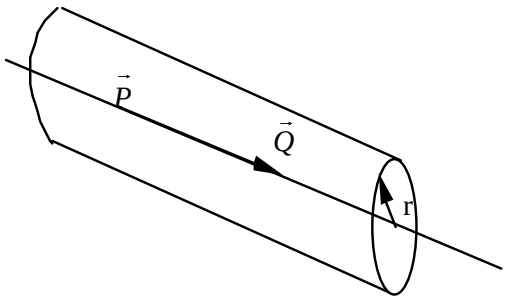
GEOTYP=11:
g1 =Load curve ID defining axisymmetric surface profile about Z′-axis. Load curves defined by the keywords *DEFINE_CURVE or *DEFINE_CURVE_ ENTITY can be used.
g2 = Number of elements along circumference
EQ.0: default set to 10
g3 = Number of elements along axis
EQ.0: default set to 20
EQ.-1: the elements are generated from the points on the load curve
g4 = Number of sub divisions on load curve used to calculate contact
EQ:0 default set to 1000



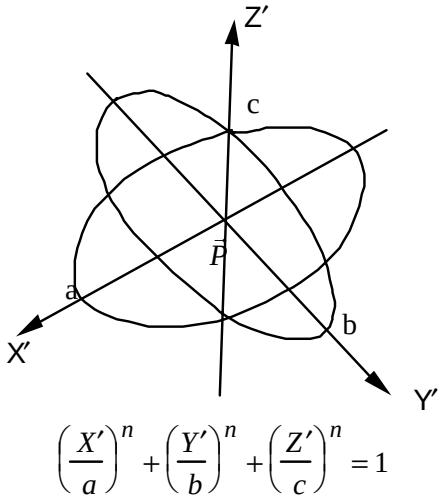
IGTYPE= 1: Infinite Plane



IGTYPE= 2: Sphere



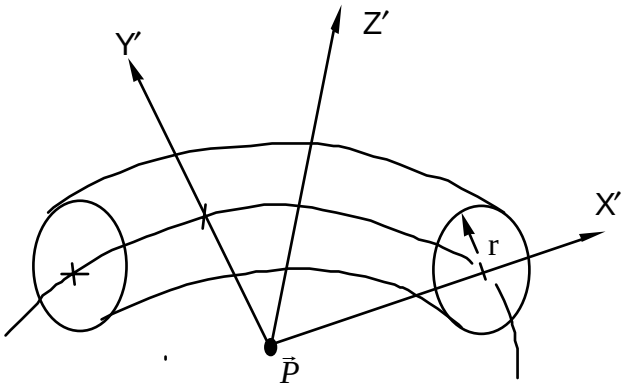
IGTYPE= 3: Infinite Cylinder



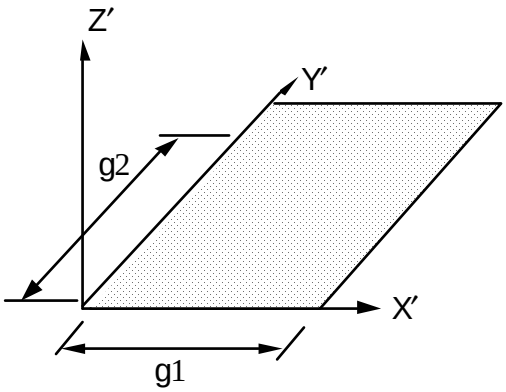
$$\left(\frac{X'}{a}\right)^n + \left(\frac{Y'}{b}\right)^n + \left(\frac{Z'}{c}\right)^n = 1$$

IGTYPE= 4: Hyperellipsoid

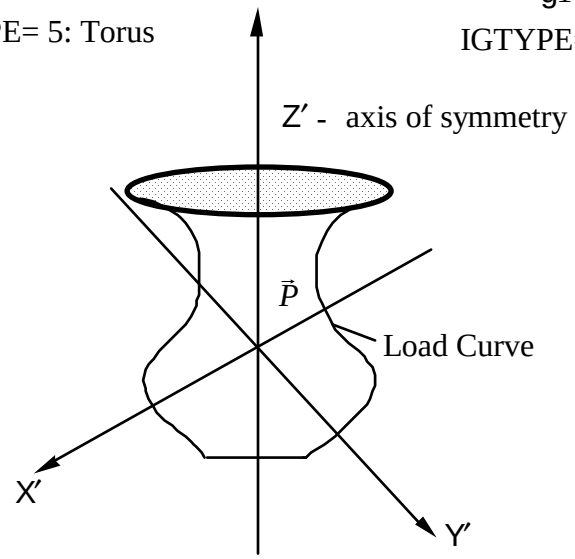
Figure 7.4a. Contact Entities.



IGTYPE= 5: Torus



IGTYPE= 10: Finite Plane



IGTYPE= 11:Load Curve

Figure 7.4b. Contact Entities.

CONTACT_GEBOD_*OPTION

Purpose: Define contact interaction between the segment of a GEBOD dummy and parts or nodes of the finite element model. This implementation follows that of the contact entity, however, it is specialized for the dummies. Forces may be output using the *DATABASE_GCEOUT command. See *COMPONENT_GEBOD and Appendix N for further details.

Conventional *CONTACT_*OPTION* treatment (surface-to-surface, nodes-to-surface, etc.) can also be applied to the segments of a dummy. To use this approach it is first necessary to determine part ID assignments by running the model through LS-DYNA's initialization phase.

The following options are available and refer to the ellipsoids which comprise the dummy. Options involving **HAND** are not applicable for the child dummy since its lower arm and hand share a common ellipsoid.

LOWER_TORSO

MIDDLE_TORSO

UPPER_TORSO

NECK

HEAD

LEFT_SHOULDER

RIGHT_SHOULDER

LEFT_UPPER_ARM

RIGHT_UPPER_ARM

LEFT_LOWER_ARM

RIGHT_LOWER_ARM

LEFT_HAND

RIGHT_HAND

LEFT_UPPER_LEG

RIGHT_UPPER_LEG

LEFT_LOWER_LEG

RIGHT_LOWER_LEG

LEFT_FOOT

RIGHT_FOOT

Card 1	1	2	3	4	5	6	7	8
Variable	DID	SSID	SSTYP	SF	DF	CF	INTORD	
Type	I	I	I	F	F	F	I	
Default	required	required	required	1.	20.	0.5	0	

VARIABLE	DESCRIPTION
DID	Dummy ID, see *COMPONENT_GEBOD_*OPTION*.
SSID	Slave set ID, see *SET_NODE_*OPTION*, *PART, or *SET_PART.
SSTYP	Slave set type: EQ.0: node set, EQ.1: part ID, EQ.2: part set ID.
SF	Penalty scale factor. Useful to scale maximized penalty.
DF	Damping option, see description for *CONTACT_*OPTION*: EQ.0: no damping, GT.0: viscous damping in percent of critical, e.g., 20 for 20% damping, EQ.-n: |n| is the load curve ID giving the damping force versus relative normal velocity (see Remark 1 below).
CF	Coulomb friction coefficient (see Remark 2 below). Assumed to be constant.
INTORD	Integration order (slaved materials only). EQ.0: check nodes only, EQ.1: 1 point integration over segments, EQ.2: 2×2 integration, EQ.3: 3×3 integration, EQ.4: 4×4 integration, EQ.5: 5×5 integration. This option allows a check of the penetration of the dummy segment into the deformable (slaved) material. Then virtual nodes at the location of the integration points are checked.

Card 2	1	2	3	4	5	6	7	8
Variable	BT	DT	SO					
Type	F	F	I					
Default	0.	1.E+20	0					

VARIABLE	DESCRIPTION
BT	Birth time
DT	Death time
SO	Flag to use penalty stiffness as in surface-to-surface contact: EQ.0: contact entity stiffness formulation, EQ.1: surface to surface contact method, EQ.-n: |n| is the load curve ID giving the force versus the normal penetration.

Remarks:

1. The optional load curves that are defined for damping versus relative normal velocity and for force versus normal penetration should be defined in the positive quadrant. The sign for the damping force depends on the direction of the relative velocity and the treatment is symmetric if the damping curve is in the positive quadrant. If the damping force is defined in the negative and positive quadrants, the sign of the relative velocity is used in the table look-up.

2. Insofar as these ellipsoidal contact surfaces are continuous and smooth it may be necessary to specify Coulomb friction values larger than those typically used with faceted contact surfaces.

***CONTACT_GUIDED_CABLE_**{*OPTION1*}_{*OPTION2*}

Purpose: Define a sliding contact that guides 1D elements, such as springs, trusses, and beams, through a list of nodes. Each node in the node set is permitted contact with a single 1D element. If for some reason, a node is in contact with multiple 1D elements, one guided contact definition must be used for each contact. The ordering of the nodal points and 1D elements in the input is arbitrary.

OPTION1 specifies that a part set ID is given with the single option:

<BLANK>

SET

If not used a part ID is assumed.

OPTION2 specifies that the first card to read defines the heading and ID number of the contact interface and takes the single option:

ID

The following card is read if and only if the ID option is specified.

Optional 1 2-8

Variable	CID	HEADING
Type	I	A70

VARIABLE	DESCRIPTION
CID	Contact interface ID. This must be a unique number.
HEADING	Interface descriptor. It is suggested that unique descriptions be used.

Card 1	1	2	3	4	5	6	7	8
Variable	NSID	PID/PSID	SOFT	SSFAC	FRIC			
Type	I	I	I	F	F			
Default	none	none	0	1.0	none			

VARIABLE	DESCRIPTION
NSID	Node set ID that guides the 1D elements.
PID/PSID	Part ID or part set ID if SET is included in the keyword line.
SOFT	Flag for soft constraint option. Set to 1 for soft constraint.
SSFAC	Stiffness scale factor for penalty stiffness value. The default value is unity. This applies to SOFT set to 0 and 1.
FRIC	Contact friction.

*CONTACT_INTERIOR

Purpose: Define interior contact for foam hexahedral and tetrahedral elements. Frequently, when foam materials are compressed under high pressure, the solid elements used to discretize these materials may invert leading to negative volumes and error terminations. In order to keep these elements from inverting, it is possible to consider interior contacts within the foam between layers of interior surfaces made up of the faces of the solid elements. Since these interior surfaces are generated automatically, the part (material) ID's for the materials of interest are defined here, prior to the interface definitions. ONLY ONE PART SET ID CAN BE DEFINED.

Card 1	1	2	3	4	5	6	7	8
Variable	PSID							
Type	I							
Default	none							

VARIABLE	DESCRIPTION
PSID	Part set ID including all parts for which interior contact is desired.

Four attributes should be defined for the part set:

Attribute 1: PSF, penalty scale factor (Default=1.00).

Attribute 2: Activation factor, F_a (Default=0.10). When the crushing of the element reaches F_a times the initial thickness the contact algorithm begins to act.

Attribute 3: ED, Optional modulus for interior contact stiffness.

Attribute 4: TYPE, Formulation for interior contact.
EQ.1.0: Default, recommended for uniform compression
EQ.2.0: Designed to control the combined modes of shear and compression. Works for type 1 brick formulation and type 10 tetrahedron formulation.

Define the part set with the *SET_PART_COLUMN option to specify independent attribute values for each part in the part set,.

Remarks:

The interior penalty is determined by the formula:

$$K = \frac{SLSFAC \cdot PSF \cdot Volume^{\frac{2}{3}} \cdot E}{Min.Thickness}$$

where SLSFAC is the value specified on the *CONTROL_CONTACT card , volume is the volume of the brick element, E is a constitutive modulus, and min. thickness is approximately the thickness of the solid element through its thinnest dimension. If ED, is defined above the interior penalty is then given instead by:

$$K = \frac{Volume^{\frac{2}{3}} \cdot ED}{Min.Thickness}$$

where the scaling factors are ignored. Generally, ED should be taken as the locking modulus specified for the foam constitutive model.

Caution should be observed when using this option since if the time step size is too large an instability may result. The time step size is not affected by the use of interior contact.

*CONTACT_RIGID_SURFACE

Purpose: Define rigid surface contact. The purpose of rigid surface contact is to model large rigid surfaces, e.g., road surfaces, with nodal points and segments that require little storage and are written out at the beginning of the binary databases. The rigid surface motion, which can be optionally prescribed, is defined by a displacement vector which is written with each output state. The nodal points defining the rigid surface must be defined in the *NODE_RIGID_SURFACE section of this manual. These rigid nodal points do not contribute degrees-of-freedom.

Card 1

Variable	CID	PSID	BOXID	SSID	FS	FD	DC	VC
Type	I	I	I	I	F	F	F	F
Default	none	none	0	none	0.	0.	0.	0.

Card 2

Variable	LCIDX	LCIDY	LCIDZ	FSLCID	FDLCID			
Type	I	I	I	I	I			
Default	0	0	0	0	0			

Card 3

Variable	SFS	STTHK	SFTHK	XPENE	BSORT	CTYPE		
Type	F	F	F	F	I	I		
Default	1.0	0.0	1.0	4.0	10	0		

VARIABLE	DESCRIPTION
CID	Contact interface ID. This must be a unique number.
PSID	Part set ID of all parts that may contact the rigid surface. See *SET_PART.
BOXID	Include only nodes of the part set that are within the specified box, see *DEFINE_BOX, in contact. If BOXID is zero, all nodes from the part set, PSID, will be included in the contact.
SSID	Segment set ID defining the rigid surface. See *SET_SEGMENT.
FS	Static coefficient of friction. The frictional coefficient is assumed to be dependent on the relative velocity v_{rel} of the surfaces in contact $\mu_c = FD + (FS - FD)e^{-DC\cdot\lvert v_{rel}\rvert}$. If FSLCID is defined, see below, then FS is overwritten by the value from the load curve.
FD	Dynamic coefficient of friction. The frictional coefficient is assumed to be dependent on the relative velocity v_{rel} of the surfaces in contact $\mu_c = FD + (FS - FD)e^{-DC\cdot\lvert v_{rel}\rvert}$. If FDLCID is defined, see below, then FD is overwritten by the value from the load curve.
DC	Exponential decay coefficient. The frictional coefficient is assumed to be dependent on the relative velocity v_{rel} of the surfaces in contact $\mu_c = FD + (FS - FD)e^{-DC\cdot\lvert v_{rel}\rvert}$.
VC	Coefficient for viscous friction. This is necessary to limit the friction force to a maximum. A limiting force is computed $F_{lim} = VC \cdot A_{cont}$. A_{cont} being the area of the segment contacted by the node in contact. The suggested value for VC is to use the yield stress in shear $VC = \frac{\sigma_o}{\sqrt{3}}$ where σ_o is the yield stress of the contacted material.
LCIDX	Load curve ID defining x-direction motion. If zero, there is no motion in the x-coordinate system.
LCIDY	Load curve ID defining y-direction motion. If zero, there is no motion in the y-coordinate system.
LCIDZ	Load curve ID defining z-direction motion. If zero, there is no motion in the z-coordinate system.
FSLCID	Load curve ID defining the static coefficient of friction as a function of interface pressure. This option applies to shell segments only.

VARIABLE	DESCRIPTION
FDLCID	Load curve ID defining the dynamic coefficient of friction as a function of interface pressure. This option applies to shell segments only.
SFS	Scale factor on default slave penalty stiffness, see also *CONTROL_CONTACT.
STTHK	Optional thickness for slave surface (overrides true thickness). This option applies to contact with shell, solid, and beam elements. True thickness is the element thickness of the shell elements. Thickness offsets are not used for solid element unless this option is specified.
SFTHK	Scale factor for slave surface thickness (scales true thickness). This option applies only to contact with shell elements. True thickness is the element thickness of the shell elements.
XPENE	Contact surface maximum penetration check multiplier. If the penetration of a node through the rigid surface exceeds the product of XPENE and the slave node thickness, the node is set free. EQ.0: default is set to 4.0.
BSORT	Number of cycles between bucket sorts. The default value is set to 10 but can be much larger, e.g., 50-100, for fully connected surfaces.
CTYPE	The contact formulation. The default, CTYPE=0, is equivalent to the ONE_WAY_SURFACE_TO_SURFACE formulation, and CTYPE=1 is a penalty formulation. If the slave surface belongs to a rigid body, CTYPE=1 must be used.

Remarks:

Thickness offsets do not apply to the rigid surface. There is no orientation requirement for the segments in the rigid surface, and the surface may be assembled from disjoint, but contiguous, arbitrarily oriented meshes. With disjoint meshes, the global searches must be done frequently, about every 10 cycles, to ensure a smooth movement of a slave node between mesh patches. For fully connected meshes this frequency interval can be safely set to 50-200 steps between searches.

The modified binary database (D3PLOT) contains the road surface information prior to the state data. This information contains:

NPDS	=	Total number of rigid surface points in problem.
NRSC	=	Total number of rigid surface contact segments summed over all definitions.
NSID	=	Number of rigid surface definitions.
NVELQ	=	Number of words at the end of each binary output state defining the rigid surface motion. This equals 6 x NSID if any rigid surface moves or zero if all rigid surfaces are stationary.

PIDS = An array equal in length to NPDS. This array defines the ID for each point in the road surface.

XC = An array equal in length to 3 x NPDS. This array defines the global x, y, and z coordinates of each point.

For each road surface define the following NSID sets of data.

ID = Rigid surface ID.

NS = Number of segments in rigid surface.

IXRS = An array equal in length to 4 x NS. This is the connectivity of the rigid surface in the internal numbering system.

At the end of each state, 6 x NVELQ words of information are written. For each road surface the x, y, and z displacements and velocities are written. If the road surface is fixed, a null vector should be output. Skip this section if NVELQ=0. LS-PREPOST currently displays rigid surfaces and animates their motion.

*CONTACT_1D

Purpose: Define one-dimensional slide lines for rebar in concrete.

Card 1	1	2	3	4	5	6	7	8
Variable	NSIDS	NSIDM	ERR	SIGC	GB	SMAX	EXP	
Type	I	I	F	F	F	F	F	
Default	none	none	0.	0.	0.	0.	0.	

VARIABLE	DESCRIPTION
NSIDS	Nodal set ID for the slave nodes, see *SET_NODE.
NSIDM	Nodal set ID for the master nodes, see *SET_NODE.
ERR	External radius of rebar
SIGC	Compressive strength of concrete
GB	Bond shear modulus
SMAX	Maximum shear strain
EXP	Exponent in damage curve

Remarks:

With this option the concrete is defined with solid elements and the rebar with truss elements, each with their own unique set of nodal points. A string of consecutive nodes, called slave nodes, related to the truss elements may slide along a string of consecutive nodes, called master nodes, related to the solid elements. The sliding commences after the rebar debonds.

The bond between the rebar and concrete is assumed to be elastic perfectly plastic. The maximum allowable slip strain is given as:

$$u_{max} = SMAX \cdot e^{-EXP \cdot D}$$

where D is the damage parameter $D_{n+1} = D_n + \Delta u$. The shear force, acting on area A_S, at time *n+1* is given as:

$$f_{n+1} = min\left(f_n - GB \cdot A_s \cdot \Delta u, GB \cdot A_s \cdot u_{max} \right)$$

CONTACT_2D**_*OPTION1*_**{OPTION2***}_{***OPTION3***}**

Purpose: Define a 2-dimensional contact or slide line. This option is to be used with 2D solid and shell elements using the plane_stress, plane_strain or axisymmetric formulations, see *SECTION_SHELL.

OPTION1 specifies the contact type. The following options should be used with deformable materials only (i.e., not rigid):

SLIDING_ONLY

TIED_SLIDING

SLIDING_VOIDS

since these methods are based on the imposition of constraints. The constraint methods may be used with rigid bodies if the rigid body is the master surface and all rigid body motions are prescribed. The following options may be used with rigid materials as well:

PENALTY_FRICTION

PENALTY

AUTOMATIC_SINGLE_SURFACE

AUTOMATIC_SURFACE_TO_SURFACE

AUTOMATIC_NODE_TO_SURFACE

AUTOMATIC_SURFACE_IN_CONTINUUM

AUTOMATIC_TIED

AUTOMATIC_TIED_ONE_WAY

OPTION2 specifies a thermal contact and takes the single option:

THERMAL

Only the AUTOMATIC types: SINGLE_SURFACE, SURFACE_TO_SURFACE, and NODE_ TO_SURFACE may be used with this option.

OPTION3 specifies that the first card to read defines the title and ID number of contact interface and takes the single option:

TITLE

Note: *OPTION2* **and** *OPTION3* **may appear in any order.**

At present, the contact ID number and title are ignored by LS-DYNA but are included for extension in the near future. The title card is picked up by some of the peripheral LS-DYNA codes to aid in post-processing.

Single surface contact in two dimensions is accomplished by the AUTOMATIC_ SURFACE_TO_SURFACE option when the master surface part set is set to zero. The SINGLE_SURFACE option in version 940 has been removed.

Read the following card here if and only if the option **TITLE** is specified:

Optional 1 2

Variable	CID	NAME
Type	I	A70

For all options except the AUTOMATIC options, define the following two cards.

Card 1 1 2 3 4 5 6 7 8

Variable	SSID	MSID	TBIRTH	TDEATH				
Type	I	I	F	F				
Default	none	none	0.	1.e20				

Card 2

Variable	EXT_PAS	THETA1	THETA2	TOL_IG	PEN	TOLOFF	FRCSCL	ONEWAY
Type	I	F	F	F	F	F	F	F
Default	none	none	none	0.001	0.1	0.025	0.010	0.0

For the PENALTY_FRICTION option define the following additional card

Card 3	1	2	3	4	5	6	7	8
Variable	FRIC	FRIC_L	FRIC_H	FRIC_S				
Type	F	F	F	F				

VARIABLE	DESCRIPTION
SSID	Nodal set ID for the slave nodes, see *SET_NODE. **The slave surface must be to the left of the master surface.**
MSID	Nodal set ID for the master nodes, see *SET_NODE.
TBIRTH	Birth time for contact.
TDEATH	Death time for contact.
EXT_PAS	Slideline extension bypass option. EQ.0: extensions are use EQ.1: extensions are not used
THETA1	Angle in degrees of slideline extension at first master node. EQ.0: extension remains tangent to first master segment.
THETA2	Angle in degrees of slideline extension at last master node. EQ.0: extension remains tangent to last master segment.
TOL_IG	Tolerance for determining initial gaps. EQ.0.0: default set to 0.001
PEN	Scale factor or penalty. EQ.0.0: default set to 0.10
TOLOFF	Tolerance for stiffness insertion for implicit solution only. The contact stiffness is inserted when a node approaches a segment a distance equal to the segment length multiplied by TOLOFF. The stiffness is increased as the node moves closer with the full stiffness being used when the nodal point finally makes contact. EQ.0.0: default set to 0.025.
FRCSCL	Scale factor for the interface friction. EQ.0.0: default set to 0.010

VARIABLE	DESCRIPTION
ONEWAY	Flag for one way treatment. If set to 1.0 the nodal points on the slave surface are constrained to the master surface. This option is generally recommended if the master surface is rigid. EQ.1.0: activate one way treatment.
FRIC	Coefficient of friction
FRIC_L	Coefficient of friction at low velocity.
FRIC_H	Coefficient of friction at high velocity.
FRIC_S	Friction factor for shear.

For the AUTOMATIC options define the following two cards:

Card 1

Variable	SIDS	SIDM	SFACT	FREQ	FS	FD	DC	
Type	I	I	F	I	F	F	F	
Default	none	none	1.0	50	0.	0.	0.	
Remarks	1,2	1,2						

Card 2

Variable	TBIRTH	TDEATH	SOS	SOM	NDS	NDM	COF	INIT
Type	F	F	F	F	I	I	I	I
Default	0.	1.e20	1.0	1.0	0	0	0	0
Remarks			3	3	4	5		8

This Card is mandatory for the THERMAL option, i.e.,:
***CONTACT_AUTOMATIC_..._THERMAL_.....**

Optional

Variable	K	RAD	H	LMIN	LMAX	CHLM	BC_FLAG	
Type	F	F	F	F	F	F	I	
Default	none	none	none	none	none	1.0	0	

Optional Card A

Card A	1	2	3	4	5	6	7	8
Variable	VC	VDC	IPF	SLIDE	ISTIFF	TIEDGAP		
Type	F	F	I	I	I	R		
Default	0.	10.0	0	0	0			
Remarks				10	11	12		

VARIABLE	DESCRIPTION
SIDS	Set ID to define the slave surface. If SIDS>0, a part set is assumed, see *SET_PART. If SIDS<0, a node set with ID equal to the absolute value of SIDS is assumed, see *SET_NODE.
SIDM	Set ID to define the master surface. If SIDM>0, a part set is assumed, see *SET_PART. If SIDM<0, a node set with ID equal to the absolute value of SIDM is assumed, see *SET_NODE. Do not define for single surface contact.
SFACT	Scale factor for the penalty force stiffness.
FREQ	Search frequency. The number of timesteps between bucket sorts. For implicit contact this parameter is ignored and the search frequency is 1. EQ.0: default set to 50.
FS	Static coefficient of friction. The frictional coefficient is assumed to be dependent on the relative velocity v_{rel} of the surfaces in contact according to the relationship given by: $\mu_c = FD + (FS - FD)e^{-DC \cdot 1 v_{rel}^1}$.
FD	Dynamic coefficient of friction. The frictional coefficient is assumed to be dependent on the relative velocity v_{rel} of the surfaces in contact $\mu_c = FD + (FS - FD)e^{-DC \cdot \lvert v_{rel} \rvert}$.
DC	Exponential decay coefficient. The frictional coefficient is assumed to be dependent on the relative velocity v_{rel} of the surfaces in contact $\mu_c = FD + (FS - FD)e^{-DC \cdot \lvert v_{rel} \rvert}$.

VARIABLE	DESCRIPTION
TBIRTH	Birth time for contact.
TDEATH	Death time for contact.
SOS	Surface offset from midline for 2D shells of slave surface EQ.0.0: default to 1. GT.0.0: scale factor applied to actual thickness LT.0.0: absolute value is used as the offset
SOM	Surface offset from midline for 2D shells of master surface EQ.0: default to 1. GT.0: scale factor applied to actual thickness LT.0: absolute value is used as the offset
NDS	Normal direction flag for 2D shells of slave surface EQ.0: Normal direction is determined automatically EQ.1: Normal direction is in the positive direction EQ.-1: Normal direction is in the negative direction
NDM	Normal direction flag for 2D shells of master surface EQ.0: Normal direction is determined automatically EQ.1: Normal direction is in the positive direction EQ.-1: Normal direction is in the negative direction
COF	Closing/Opening flag for implicit contact EQ.0: Recommended for most problem where gaps are only closing. EQ.1: Recommended when gaps are opening to avoid sticking.
INIT	Special processing during initialization EQ.0: No special processing. EQ.1: Forming option.
K	Thermal conductivity (k) of fluid between the slide surfaces. If a gap with a thickness l_{gap} exists between the slide surfaces, then the conductance due to thermal conductivity between the slide surfaces is $h_{cond} = \frac{k}{l_{gap}}$ Note that LS- DYNA calculates l_{gap} based on deformation.

VARIABLE	DESCRIPTION
RAD	Radiation factor, f, between the slide surfaces. A radiant-heat-transfer coefficient (h_{rad}) is calculated (see *BOUNDARY_RADIATION). If a gap exists between the slide surfaces, then the contact conductance is calculated by $$h = h_{cond} + h_{rad}$$
H	Heat transfer conductance (h_{cont}) for closed gaps. Use this heat transfer conductance for gaps in the range $$0 \le l_{gap} \le l_{\min}$$ where $l_{\min}$ is GCRIT defined below.
LMIN	Critical gap $(l_{\min})$, use the heat transfer conductance defined (HTC) for gap thicknesses less than this value.
LMAX	No thermal contact if gap is greater than this value $(l_{\max})$.
CHLM	Is a multiplier used on the element characteristic distance for the search routine. The characteristic length is the largest interface surface element diagonal. EQ.0: Default set to 1.0
BC_FLAG	Thermal boundary condition flag EQ.0: thermal boundary conditions are **on** when parts are in contact EQ.1: thermal boundary conditions are **off** when parts are in contact
VC	Coefficient for viscous friction. This is used to limit the friction force to a maximum. A limiting force is computed $F_{lim} = VC \cdot A_{cont}$. A_{cont} being the area of contacted between segments. The suggested value for *VC* is to use the yield stress in shear: $$VC = \frac{\sigma_o}{\sqrt{3}}$$ where σ_o is the yield stress of the contacted material.
VDC	Viscous damping coefficient in percent of critical for explicit contact.
IPF	Initial penetration flag for explicit contact. EQ.0: Allow initial penetrations to remain EQ.1: Push apart initially penetrated surfaces

VARIABLE	DESCRIPTION
SLIDE	Sliding option. EQ.0: Off EQ.1: On
ISTIFF	Stiffness scaling option. EQ.0: Use default option. EQ.1: Scale stiffness using segment masses and explicit time step (default for explicit contact) EQ.2: Scale stiffness using segment stiffness and dimensions (default for implicit contact)
TIEDGAP	Search gap for tied contacts. EQ.0: Default, use 1% of the master segment length GT.0: Use the input value LT.0: Use n% of the master segment length where n=|TIEDGAP|

Remarks:

Remarks 1 through 12 pertain to 2D_AUTOMATIC contact.

1. For AUTOMATIC_SURFACE_TO_SURFACE, AUTOMATIC_SINGLE_SURFACE contact and AUTOMATIC_NODE_TO_SURFACE contact, penetration of 2D shell elements and external faces of 2D continuum elements is prevented by penalty forces. Parts in the slave part set are checked for contact with parts in the master part set. Self contact is checked for any part in both sets. If the slave part set is omitted, all parts are checked for contact. If the master part set is omitted, it is assumed to be identical to the slave part set.

2. For AUTOMATIC_SURFACE_IN_CONTINUUM contact, penalty forces prevent the flow of slave element material (the continuum) through the master surfaces. Flow of the continuum tangent to the surface is permitted. Only 2D solid parts are permitted in the slave part set. Both 2D solid and 2D shell parts are permitted in the master part set. Neither the slave part set ID or the master part set ID may be omitted.

3. By default, the true thickness of 2D shell elements is taken into account for AUTOMATIC_SURFACE_TO_SURFACE and AUTOMATIC_NODE_TO_SURFACE contact. The user can override the true thickness by using SOS and SOM. If the surface offset is reduced to a small value, the automatic normal direction algorithm may fail, so it is best to specify the normal direction using NDS or NDM. Thickness of 2D shell elements is not considered for AUTOMATIC_SURFACE_IN_CONTINUUM contact.

4. By default, the normal direction of 2D shell elements is evaluated automatically for AUTOMATIC_SURFACE_TO_SURFACE and AUTOMATIC_NODE_TO_SURFACE contact. The user can override the automatic algorithm using NDS or NDM and contact will occur with the positive or negative face of the element.

5. For SURFACE_IN_CONTINUUM contact, flow through 2D shell elements is prevented in both directions by default. If NDM is set to ±1, flow in the direction of the normal is permitted.

6 When using AUTOMATIC_SURFACE_IN_CONTINUUM contact, there is no need to mesh the continuum around the structure because contact is not with continuum nodes but with material in the interior of the continuum elements. The algorithm works well for Eulerian or ALE elements since the structure does not interfere with remeshing. However, a structure will usually not penetrate the surface of an ALE continuum since the nodes are Lagrangian normal to the surface. Therefore, if using an ALE fluid, the structure should be initially immersed in the fluid and remain immersed throughout the calculation. Penetrating the surface of an Eulerian continuum is not a problem.

7. For all types of 2D_AUTOMATIC contact, eroding materials are treated by default. At present, subcycling is not possible.

8. Currently only one special initialization option is available. The forming option is intended for implicit solutions of thin solid parts when back side segments may interfere with the solution. It automatically removes back side segments during initialization. If slave or master surfaces are defined by node sets, the forming option should be turned off.

9. For the thermal option:

$$h = h_{cont}, \text{ if the gap thickness is } 0 \le l_{gap} \le l_{\min}$$
$$h = h_{cond} + h_{rad}, \text{ if the gap thickness is } l_{\min} \le l_{gap} \le l_{\max}$$
$$h = 0, \text{ if the gap thickness is } l_{gap} > l_{\max}$$

10. When turned on, the sliding option activates additional logic intended to improve sliding when surfaces in contact have kinks or corners. This option is off by default.

11. The ISTIFF option allows control of the equation used in calculating the penalty stiffness. For backward compatibility, the default values are different for implicit and explicit solutions. When ISTIFF=1 is used, the explicit time step appears in the stiffness equation regardless if the calculation is implicit or explicit.

12. AUTOMATIC_TIED_ONE_WAY contact creates two degree of freedom translational kinematic constraints to nodes on the slave surface which are initially located on or near master segments. AUTOMATIC_TIED contact creates kinematic constraints between slave nodes and master segments, and also creates penalty constraints between master nodes and slave segments. With either contact option, a kinematic constraint may be switched to penalty if there is a conflict with another constraint. The TIEDGAP parameter determines the maximum normal distance from a segment to a node for a constraint to be formed. Nodes will not be moved to eliminate an initial gap, and the initial gap will be maintained throughout the calculation.

The remaining discussion applies to the SLIDING_ONLY, TIED_SLIDING, SLIDING_ VOIDS, PENALTY_FRICTION, and PENALTY options. These options were adopted from LS-DYNA2D and originated in the public domain version of DYNA2D from the Lawrence

Livermore National Laboratory. The AUTOMATIC contact options are generally recommended excepted for the TIED option.

Consider two slideline surfaces in contact. It is necessary to designate one as a slave surface and the other as a master surface. Nodal points defining the slave surface are called slave nodes, and similarly, nodes defining the master surface are called master nodes. Each slave-master surface combination is referred to as a slideline.

Many potential problems with the algorithm can be avoided by observing the following precautions:

- Metallic materials should contain the master surface along high explosive-metal interfaces.

- Sliding only type slidelines are appropriate along high explosive-metal interfaces. The penalty formulation is not recommended along such interfaces.

- If one surface is more finely zoned, it should be used as the slave surface. If penalty slidelines are used, PENALTY and PENALTY_FRICTION, the slave-master distinction is irrelevant.

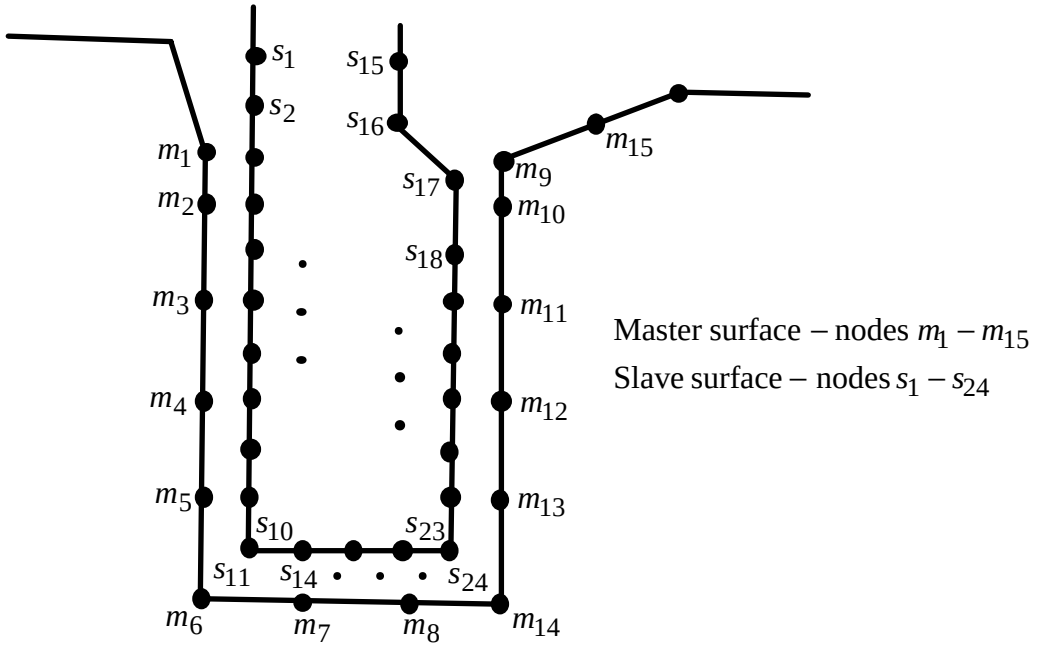
- A slave node may have more than one master segment, and may be included as a member of a master segment if a slideline intersection is defined.

- Angles in the master side of a slideline that approach 90° must be avoided.

  Whenever such angles exist in a master surface, two or more slidelines should be defined. This procedure is illustrated in Figure 7.5. An exception for the foregoing rule arises if the surfaces are tied. In this case, only one slideline is needed.

- Whenever two surfaces are in contact, the smaller of the two surfaces should be used as the slave surface. For example, in modeling a missile impacting a wall, the contact surface on the missile should be used as the slave surface.

- Care should be used when defining a master surface to prevent the extension from interfering with the solution. In Figures 7.6 and 7.7, slideline extensions are shown.



1		2		3	
Slaves	Masters	Slaves	Masters	Slaves	Masters
s_1	m_1	s_{11}	m_6	s_{24}	m_{14}
s_2	m_2	s_{12}	m_7	s_{23}	m_{13}
.	.	.	m_8	.	.
.	.	.	m_{14}	.	.
.	.	.	.	.	.
		s_{14}			m_9
s_{11}	m_6	s_{24}		s_{15}	m_{15}

Figure 7.5. Proper definition of illustrated slave-master surface requires three slidelines (note that slave surface is to the left of the master surface as one moves along master nodes in order of definition).

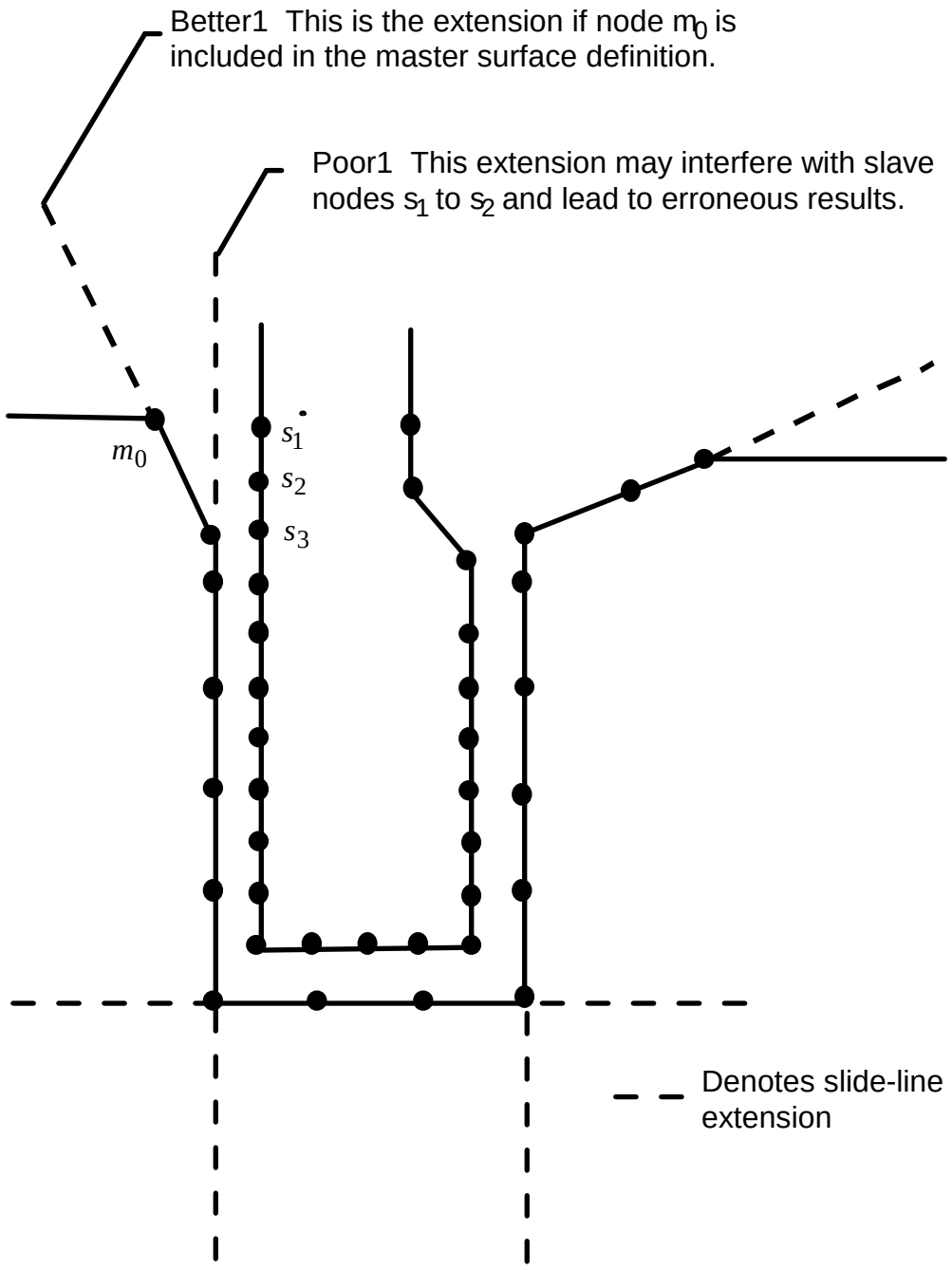


Figure 7.6. Master surface extensions defined automatically by DYNA (extensions are updated every time step to remain tangent to ends of master sides of slidelines unless angle of extension is defined in input).

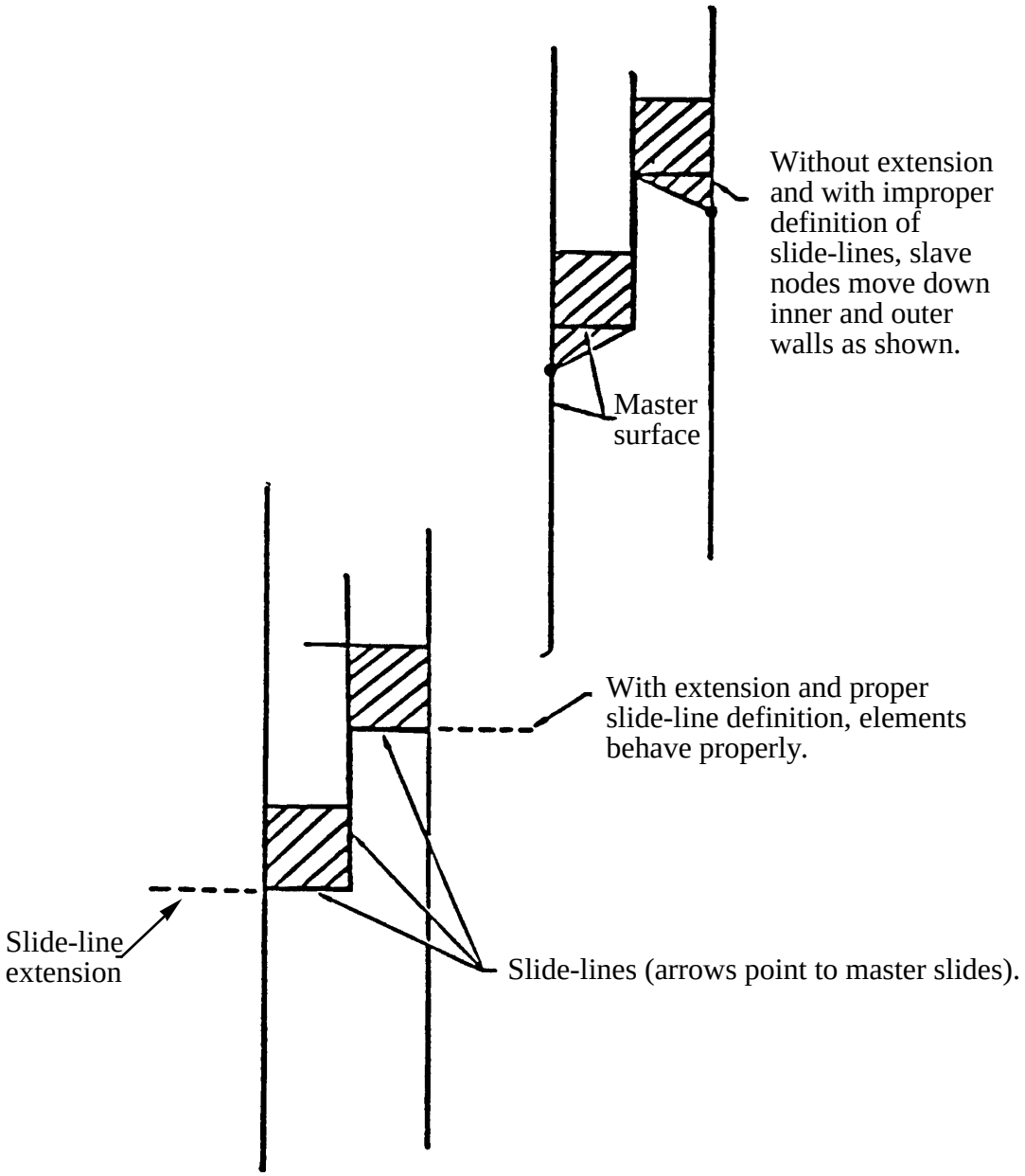


Figure 7.7. Example of slideline extensions helping to provide realistic response.

*CONTROL

The keyword control cards are optional and can be used to change defaults, activate solution options such as mass scaling, adaptive remeshing, and an implicit solution; however, it is advisable to define the *CONTROL_TERMINATION card. **The ordering of the control cards in the input file is arbitrary. To avoid ambiguities, define no more than one control card of each type.** The following control cards are organized in alphabetical order:

***CONTROL_ACCURACY**
***CONTROL_ADAPSTEP**
***CONTROL_ADAPTIVE**
***CONTROL_ALE**
***CONTROL_BULK_VISCOSITY**
***CONTROL_CHECK_**{*OPTION*}
***CONTROL_COARSEN**
***CONTROL_CONTACT**
***CONTROL_COUPLING**
***CONTROL_CPU**
***CONTROL_DYNAMIC_RELAXATION**
***CONTROL_EFG**
***CONTROL_ENERGY**
***CONTROL_EXPLOSIVE_SHADOW**
***CONTROL_FORMING_POSITION**
***CONTROL_FORMING_TEMPLATE**
***CONTROL_FORMING_TRAVEL**
***CONTROL_FORMING_USER**
***CONTROL_HOURGLASS_**{*OPTION*}
***CONTROL_IMPLICIT_AUTO**
***CONTROL_IMPLICIT_BUCKLE**
***CONTROL_IMPLICIT_CONSISTENT_MASS**
***CONTROL_IMPLICIT_DYNAMICS**
***CONTROL_IMPLICIT_EIGENVALUE**
***CONTROL_IMPLICIT_GENERAL**
***CONTROL_IMPLICIT_INERTIA_RELIEF**
***CONTROL_IMPLICIT_JOINTS**
***CONTROL_IMPLICIT_MODES**
***CONTROL_IMPLICIT_SOLUTION**

*CONTROL

***CONTROL_IMPLICIT_SOLVER**
***CONTROL_IMPLICIT_STABILIZATION**
***CONTROL_IMPLICIT_TERMINATION**
***CONTROL_MPP_DECOMPOSITION_AUTOMATIC**
***CONTROL_MPP_DECOMPOSITION_DISTRIBUTE_BAGREF**
***CONTROL_MPP_DECOMPOSITION_CHECK_SPEED**
***CONTROL_MPP_DECOMPOSITION_CONTACT_DISTRIBUTE**
***CONTROL_MPP_DECOMPOSITION_CONTACT_ISOLATE**
***CONTROL_MPP_DECOMPOSITION_DISTRIBUTE_ALE_ELEMENTS**
***CONTROL_MPP_DECOMPOSITION_DISTRIBUTE_SPH_ELEMENTS**
***CONTROL_MPP_DECOMPOSITION_ELCOST**
***CONTROL_MPP_DECOMPOSITION_FILE**
***CONTROL_MPP_DECOMPOSITION_METHOD**
***CONTROL_MPP_DECOMPOSITION_NUMPROC**
***CONTROL_MPP_DECOMPOSITION_OUTDECOMP**
***CONTROL_MPP_DECOMPOSITION_PARTS_DISTRIBUTE**
***CONTROL_MPP_DECOMPOSITION_SCALE_CONTACT_COST**
***CONTROL_MPP_DECOMPOSITION_SCALE_FACTOR_SPH**
***CONTROL_MPP_DECOMPOSITION_SHOW**
***CONTROL_MPP_DECOMPOSITION_TRANSFORMATION**
***CONTROL_MPP_IO_BINOUTONLY**
***CONTROL_MPP_IO_LSTC_REDUCE**
***CONTROL_MPP_IO_NOD3DUMP**
***CONTROL_MPP_IO_NODUMP**
***CONTROL_MPP_IO_NOFULL**
***CONTROL_MPP_IO_SWAPBYTES**
***CONTROL_NONLOCAL**
***CONTROL_OUTPUT**
***CONTROL_PARALLEL**
***CONTROL_PORE_FLUID**
***CONTROL_REMESHING**
***CONTROL_RIGID**
***CONTROL_SHELL**
***CONTROL_SOLID**
***CONTROL_SOLUTION**
***CONTROL_SPH**
***CONTROL_SPOTWELD_BEAM**

***CONTROL_STAGED_CONSTRUCTION**

***CONTROL_STRUCTURED**_{*OPTION*}

***CONTROL_SUBCYCLE**

***CONTROL_TERMINATION**

***CONTROL_THERMAL_NONLINEAR**

***CONTROL_THERMAL_SOLVER**

***CONTROL_THERMAL_TIMESTEP**

***CONTROL_TIMESTEP**

LS-DYNA's implicit mode may be activated in two ways. Using the *CONTROL_IMPLICIT_GENERAL keyword, a simulation may be flagged to run entirely in implicit mode. Alternatively, an explicit simulation may be seamlessly switched into implicit

mode at the termination time using the *INTERFACE_SPRINGBACK_SEAMLESS keyword. The seamless switching feature is intended to simplify metal forming springback calculations, where the forming phase can be run in explicit mode, followed immediately by an implicit static springback simulation. In case of difficulty, restart capability is supported. Eight keywords are available to support implicit analysis. Default values are carefully selected to minimize input necessary for most simulations. These are summarized below:

*CONTROL_IMPLICIT_GENERAL
: Activates implicit mode, selects time step size.

*CONTROL_IMPLICIT_INERTIA_RELIEF
: Allows linear analysis of models with rigid body modes.

*CONTROL_IMPLICIT_SOLVER
: Selects parameters for solving system of linear equations [K]{x}={f}.

*CONTROL_IMPLICIT_SOLUTION
: Selects linear or nonlinear solution method, convergence tolerances.

*CONTROL_IMPLICIT_AUTO
: Activates automatic time step control.

*CONTROL_IMPLICIT_DYNAMICS
: Activates and controls dynamic implicit solution using Newmark method.

*CONTROL_IMPLICIT_EIGENVALUE
: Activates and controls eigenvalue analysis.

*CONTROL_IMPLICIT_MODES
: Activates and controls computation of constraint and attachment modes.

*CONTROL_IMPLICIT_STABILIZATION
: Activates and controls artificial stabilization for multi-step springback.

*CONTROL_ACCURACY

Purpose: Define control parameters that can improve the accuracy of the calculation.

Card 1	1	2	3	4	5	6	7	8
Variable	OSU	INN	PIDOSU					
Type	I	I	I					
Default	0 (off)		optional					

VARIABLE	DESCRIPTION
OSU	Global flag for 2nd order objective stress updates (See Remark 1 below). Generally, for explicit calculations only those parts undergoing large rotations, such as rolling tires, need this option. Objective stress updates can be activated for a subset of part IDs by defining the part set in columns 21-30. EQ.0: Off (default) EQ.1: On
INN	Invariant node numbering for shell and solid elements. (See Remarks 2 and 3 below). EQ.-4: On for both shell and solid elements except triangular shells EQ.-2: On for shell elements except triangular shells EQ. 1: Off (default for explicit) EQ. 2: On for shell elements only (default for implicit) EQ. 3: On for solid elements only EQ. 4: On for both shell and solid elements
PIDOSU	Part set ID for objective stress updates. If this part set ID is given only those part IDs listed will use the objective stress update; therefore, OSU is ignored.

Remarks:

1. Second order objective stress updates are occasionally necessary. Some examples include spinning bodies such as turbine blades in a jet engine, high velocity impacts generating large strains in a few time steps, and large time step sizes due to mass scaling in metal forming. There is a significantly added cost which is due in part to the added cost of the second order terms in the stress update when the Jaumann rate is used and the need to compute the strain-displacement matrix at the mid-point geometry. This option is available for one point brick elements, the selective-reduced integrated brick element which uses eight integration points, the fully integrated plane strain and axisymmetric

volume weighted (type 15) 2D solid elements, the fully integrated thick shell element, and the following shell elements: Belytschko-Tsay, Belytschko-Tsay with warping stiffness, Belytschko-Chiang-Wong, S/R Hughes-Liu, and the type 16 fully integrated shell element.

2. Invariant node numbering for shell elements affects the choice of the local element shell coordinate system. The orientation of the default local coordinate system is based on the shell normal vector and the direction of the 1-2 side of the element. If the element numbering is permuted, the results will change in irregularly shaped elements. With invariant node numbering, permuting the nodes shifts the local system by an exact multiple of 90 degrees. In spite of its higher costs [<5%], the invariant local system is recommended for several reasons. First, element forces are nearly independent of node sequencing; secondly, the hourglass modes will not substantially affect the material directions; and, finally, stable calculations over long time periods are achievable.

3. Invariant node numbering for solid elements is available for anisotropic materials only. This option has no effect on solid elements of isotropic material. This option is recommended when solid elements of anisotropic material undergo significant deformation.

*CONTROL_ADAPSTEP

Purpose: Define control parameters for contact interface force update during each adaptive cycle.

Card 1	1	2	3	4	5	6	7	8
Variable	FACTIN	DFACTR						
Type	F	F						
Default	1.0	0.01						

VARIABLE	DESCRIPTION
FACTIN	Initial relaxation factor for contact force during each adaptive remesh. To turn this option off set FACTIN=1.0. Unless stability problems occur in the contact, FACTIN=1.0 is recommended since this option can create some numerical noise in the resultant tooling forces. A typical value for this parameter is 0.10.
DFACTR	Incremental increase of FACTIN during each time step after the adaptive step. FACTIN is not allowed to exceed unity. A typical value might be 0.01.

Remarks:

1. This command applies to contact with thickness offsets including contact types:

   *CONTACT_FORMING_...,
   *CONTACT_NODES_TO_SURFACE,
   *CONTACT_SURFACE_TO_SURFACE,
   *CONTACT_ONE_WAY_SURFACE_TO_SURFACE.

***CONTROL_ADAPTIVE**

Purpose: Activate adaptive meshing. The parts which are adaptively meshed are defined by *PART. See remarks below.

Purpose: Activate adaptive meshing. The parts which are adaptively meshed are defined by *PART. See remarks below.

Card 1 1 2 3 4 5 6 7 8

Variable	ADPFREQ	ADPTOL	ADPOPT	MAXLVL	TBIRTH	TDEATH	LCADP	IOFLAG
Type	F	F	I	I	F	F	I	I
Default	none	10^{20}	1	3	0.0	10^{20}	0	0

(This card is optional).

Card 2 1 2 3 4 5 6 7 8

Variable	ADPSIZE	ADPASS	IREFLG	ADPENE	ADPTH	MEMORY	ORIENT	MAXEL
Type	F	I	I	F	F	I	I	I
Default		0	0	0.0	inactive	inactive	0	inactive

(This card is optional).

Card 3 1 2 3 4 5 6 7 8

Variable	IADPN90	IADPGH	NCFREQ	IADPCL	ADPCTL	CBIRTH	CDEATH	LCLVL
Type	I	I	I	I	F	F	F	F
Default	0	0	none	1	none	0.0	10^{20}	

(This card is optional)

Card 4	1	2	3	4	5	6	7	8
Variable	CNLA							
Type	F							
Default	0							

VARIABLE	DESCRIPTION
ADPFREQ	Time interval between adaptive refinements, see Figure 8.1.
ADPTOL	Adaptive error tolerance in degrees for ADPOPT set to 1 or 2 below. If ADPOPT is set to 8, ADPTOL is the characteristic element size.
ADPOPT	Adaptive options: EQ.1: angle change in degrees per adaptive refinement relative to the surrounding elements for each element to be refined. EQ.2: total angle change in degrees relative to the surrounding element for each element to be refined. For example, if the adptol=5 degrees, the element will be refined to the second level when the total angle change reaches 5 degrees. When the angle change is 10 degrees the element will be refined to the third level. EQ.4: adapts when the error norm base on: $\sqrt{\sum_{i,j}(\sigma_{ij}-\hat{\sigma}_{ij})^2}$ evaluated at the element integration points exceeds ADPTOL/100 times the mean stress in the mesh. The stress $\hat{\sigma}_{ij}$ is the value of the stress interpolated from the least squares fit of the stress to the nodes. This option works for shell types 2, 4, 16, 18, and 20. EQ.7: 3D r-adaptive remeshing for solid elements. Solid element type 13, a tetrahedron, and 3-D EFG type 41, are used in the adaptive remeshing process. A completely new mesh is generated which is initialized from the old mesh using a least squares approximation. The mesh size is currently based on the minimum and maximum edge lengths defined on the *CONTROL_REMESHING keyword input. This option remains under development, and, we are not sure of its reliability on complex geometries.

VARIABLE	DESCRIPTION
	Q.|8|: 2D r-adaptive remeshing for axisymmetric and plane strain solid elements. A completely new mesh is generated which is initialized from the old mesh using a least squares approximation. The mesh size is currently based on the value, ADPTOL, which gives the characteristic element size. This option is based on earlier work by Dick and Harris [1992]. If ADPOPT is negative, then self-contacting material will not be merged together. The self-merging is often preferred since it eliminates sharp folds in the boundary; however, if the sharp fold is being simulated unexpected results are generated.
MAXLVL	Maximum number of refinement levels. Values of 1, 2, 3, 4, ... allow a maximum of 1, 4, 16, 64, ... elements, respectively, to be created for each original element. The refinement level can be overridden by *DEFINE_BOX_ADAPTIVE, or *DEFINE_SET_ADAPTIVE.
TBIRTH	Birth time at which the adaptive remeshing begins, see Figure 8.1.
TDEATH	Death time at which the adaptive remeshing ends, see Figure 8.1.
LCADP	Adaptive interval is changed as a function of time given by load curve ID, LCADP. If this option is nonzero, the ADPFREQ will be replaced by LCADP. The x-axis is time and the y-axis is the varied adaptive time interval.
IOFLAG	Flag to generate adaptive mesh at exit including *NODE, *ELEMENT, *SHELL, and *BOUNDARY_, *CONTACT_NODE_, and *CONSTRAINED_ADAPTIVITY, to be saved in the file, *adapt.msh*. EQ.1: generate adaptive mesh.
ADPSIZE	Minimum element size to be adapted based on element edge length. If undefined the edge length limit is ignored. LT.0: absolute value defines the minimum characteristic element length to be adapted based on square root of the element area, i.e., instead of comparing the shortest element edge with ADPSIZE, it compares the square root of the element area with |ADPSIZE| whenever ADPSIZE is defined by a negative value.
ADPASS	One or two pass adaptivity flag: EQ.0: two pass adaptivity as shown in Figure 8.1a, EQ.1: one pass adaptivity as shown in Figure 8.1b.

VARIABLE	DESCRIPTION
IREFLG	Uniform refinement level. A value of 1, 2, 3 ... allow 4, 16, 64 ... elements, respectively, to be created uniformly for each original element. If negative, |IREFLG| is taken as a load curve ID. With the curve option, the abscissa values define the refinement time, and the ordinate values define the minimum element size. Only one refinement level is performed per time step. An advantage of the load curve option is that the mesh is adapted to honor the minimum element size, but with the uniform option, IREFLG>0, this is not possible. **Note: If the element size defined with *DEFINE_CURVE is positive, the element size will override the element size defined with *CONTROL_ADAPTIVE and *DEFINE_SET_ADAPTIVE. Also, if the element size defined with *DEFINE_CURVE is negative the element size is used for refinement only.**
ADPENE	Adapt the mesh when the contact surfaces approach or penetrate the tooling surface depending on whether the value of ADPENE is positive (*approach*) or negative (*penetrates*), respectively. The tooling adaptive refinement is based on the curvature of the tooling. If ADPENE is positive the refinement generally occurs before contact takes place; consequently, it is possible that the parameter ADPASS can be set to 1 in invoke the one pass adaptivity.
ADPTH	.EQ.0.0 This parameter is ignored .GT.0.0 Absolute shell thickness level below which adaptive remeshing should began. .LT.0.0 Element thickness ratio. If the ratio of the element thickness to the original element thickness is less than the absolute value of ADPTHK, the element will be refined. *This option works only if ADPTOL is nonzero. If thickness based adaptive remeshing is desired without angle changes, then, set ADPTOL to a large angle.*
MEMORY	This flag can have two meanings depending on whether the memory environmental variable is or is not set. The command "*setenv LSTC_MEMORY auto*" sets the memory environmental variable which causes LS-DYNA to expand memory automatically. Note that automatic memory expansion is not always 100% reliable depending on the machine and operating system level; consequently, it is not yet the default. To see if this is set on a particular machine type the command "*env*". If the environmental variable is not set then when memory usage reaches this percentage, MEMORY, further adaptivity is prevented to avoid exceeding the memory specified at execution time. Caution is necessary since memory usage is checked after each adaptive step, and, if the memory usage increases by more than the residual percentage, 100-PERCENT, the calculation will terminate.

VARIABLE DESCRIPTION

	If the memory environmental variable is set then when the number of words of memory allocated reaches or exceeds this value, MEMORY, further adaptivity is stopped.
ORIENT	This option applies to the FORMING contact option only. If this flag is set to one (1), the user orientation for the contact interface is used. If this flag is set to zero (0), LS-DYNA sets the global orientation of the contact surface the first time a potential contact is observed after the birth time. If slave nodes are found on both sides of the contact surface, the orientation is set based on the principle of "majority rules". Experience has shown that this principle is not always reliable.
MAXEL	Adaptivity is stopped if this number of elements is exceeded.
IADPN90	Maximum number of elements covering 90 degree of radii. See Remark 6.
IADPGH	Fission flag for neighbor splitting. EQ.0: split all neighbor elements EQ.1: do not split neighbor elements
NCFREQ	Frequency of fission to fusion steps. For example, if NCFREQ=4, then fusion will occur on the fourth, eighth, twelfth, etc., fission steps, respectively. If this option is used NCFREQ>1 is recommended.
IADPCL	Fusion will not occur until the fission level reaches IADPCL. Therefore, if IADPCL=2, MAXLVL=5, any element can be split into 256 elements. If the surface flattens out, the number of elements will be reduced if the fusion option is active, i.e., the 256 elements can be fused and reduced to 16.
ADPCTL	Adaptivity error tolerance in degrees for activating fusion. It follows the same rules as ADPOPT above.
CBIRTH	Birth time for adaptive fusion. If ADPENE>0, look-ahead adaptivity is active. In this case, fission, based on local tool curvature, will occur while the blank is still relatively flat. The time value given for CBIRTH should be set to a time later in the simulation after the forming process is well underway.
CDEATH	Death time for adaptive fusion.
LCLVL	Load curve ID of a curve that defines the maximum refinement level as a function of time
CNLA	Limit angle for corner nodes. See Remark 7.

Remarks:

1. D3DUMP and RUNRSF files contain all information necessary to restart an adaptive run. This did not work in version 936 of LS-DYNA.

2. Card 2 input is optional and is not required.

3. In order for this control card to work, the flag ADPOPT=1 must be set in the *PART definition. Otherwise, adaptivity will not function.

4. In order for adaptivity to work optimally, the parameter SNLOG=1, must be set on Optional Control Card B in the *CONTACT Section. On disjoint tooling meshes the contact option *CONTACT_FORMING_..... is recommended.

5. A file *adapt.rid* is left on disk after the adaptive run is completed. This file contains the root ID of all elements that are created during the calculation, and it does not need to be kept if it is not used in post-processing.

6. For springback analysis, IADPN90 is usually chosen between 4 and 6.

7. When using 2D r-adaptive remeshing, the generated new mesh should have a node at each corner so that corners are not smoothed. By default, the mesher will assume a corner wherever the interior angel between adjacent edges is less than 110 degrees. Setting CNLA larger than 110 enables angles larger than 110 to be corners. Care should be taken to avoid an unnecessarily large value of CNLA as this may prevent the mesher from generating smooth meshes.

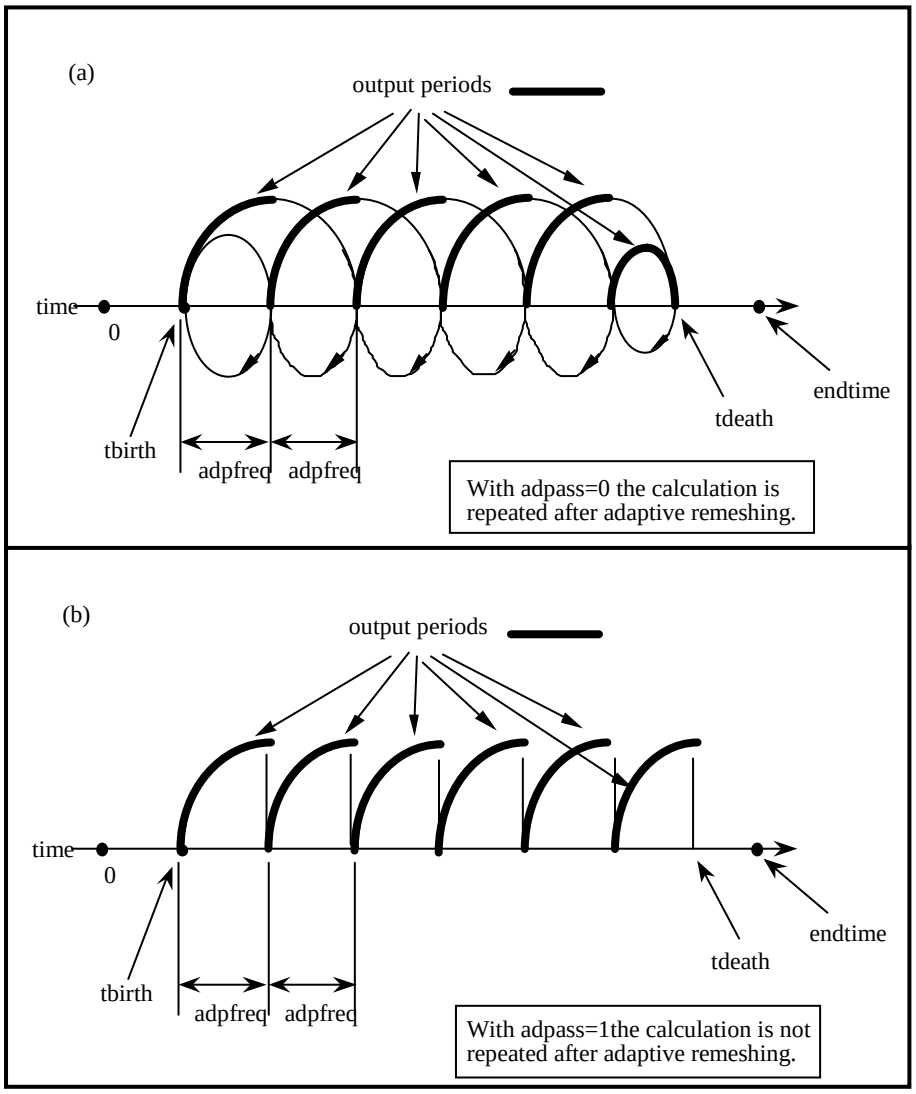


Figure 8.1. At time=*tbirth* the adaptive calculation begins. After computing for a time interval *adpfreq* error norms are computed. If ADPASS=0, then the mesh that existed at time=*tbirth* is refined based on the computed error norms. With the new mesh, the calculation continues to time=*tbirth+2 x adpfreq* where the error norms are again computed. The mesh that existed at time=*tbirth+adpfreq* is refined and the calculation continues to time=*tbirth+3 x adpfreq*, and so on. However, if ADPASS=1, then the mesh that exist at time *=thirth+adpfreq* is refined and the calculation continues. Errors that develop between adaptive remeshing are preserved. Generally, ADPASS=0 is recommended but this option is considerably more expensive.

*CONTROL_ADAPTIVE_CURVE

Purpose: To refine the element mesh along a curve. All curves defined by the keyword *DEFINE_CURVE_TRIM are used in the refinement. This option provides additional refinement to that created by *CONTRO_ADAPTIVE.

Card 1	1	2	3	4	5	6	7	8
Variable	IDSET	ITYPE	N	SMIN	ITRIOPT			
Type	I	I	I	F	I			

VARIABLE	DESCRIPTION
IDSET	Set ID
ITYPE	Set type: EQ.1: IDSET is shell set ID. EQ.2: IDSET is part set ID.
N	Refinement option: EQ.1: Refine until there are no adaptive constraints remaining in the element mesh around the curve.. GT.1: Refine no more than N levels
SMIN	If the element dimension is smaller than this value, do not refine.
ITRIOPT	Option to refine an enclosed area of a trim curve. EQ.0: Refine the elements along the trim curve EQ.1: Refine the elements along the trim curve and enclosed by the trim curve.

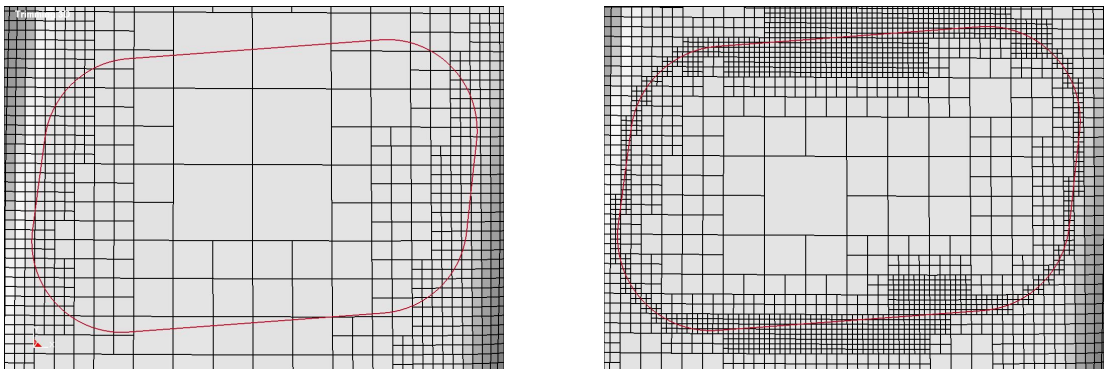


Figure 8.2. Refinement along a curve.

*CONTROL_ADAPTIVE_SOLID_TO_SPH

Purpose: Define an adaptive option to switch solid elements that reach a failure criterion into SPH elements.

Card 1	1	2	3	4	5	6	7	8
Variable	PID	ITYPE	N					
Type	I	I	I					

VARIABLE	DESCRIPTION
PID	Part Set ID or Part ID.
ITYPE	Part type: EQ.1: PID is a part set ID. EQ.2: PID is a part ID.
N	Refinement option: EQ.1: Refine one solid element into one SPH element. EQ.2: Refine one solid element into two SPH elements. EQ.3: Refine one solid element into four SPH elements.

*CONTROL_ALE

Purpose: Set global control parameters for the Arbitrary Lagrange-Eulerian (ALE) and Eulerian calculations. This is required when ELFORM = 5, 6, 7, 11, and 12.

Card 1

Variable	DCT	NADV	METH	AFAC	BFAC	CFAC	DFAC	EFAC
Type	I	I	I	F	F	F	F	F
Default	1	0	1	0	0	0	0	0

Card 2

Variable	START	END	AAFAC	VFACT	PRIT	EBC	PREF	NSIDEBC
Type	F	F	F	F	F	I	F	I
Default	0	1. 0E+20	1	1.0E-06	0.0	0	0.0	none

Optional Card 3

Card 3

Variable	NCPL	NBKT	IMASCL	CHECKR				
Type	I	I	I	F				
Default	1	50	0	0.0				

VARIABLE	DESCRIPTION
DCT	Default continuum treatment: EQ.1: Lagrangian (default), EQ.2: Eulerian, EQ.3: Arbitrary Lagrangian Eulerian, EQ.4: Eulerian Ambient.
NADV	Number of cycles between advections (almost always set to 1).
METH	Advection method: EQ.1: donor cell + HIS (Half-Index-Shift, first order accurate), EQ.2: Van Leer + HIS (Half-Index-Shift, second order). EQ.3: donor cell + HIS, first order accurate, conserving total energy over each advection step instead of conserving internal energy (See Remark 5).
AFAC	ALE smoothing weight factor - Simple average: EQ.-1: turn smoothing off. (See Remark 6).
BFAC	ALE smoothing weight factor – Volume weighting
CFAC	ALE smoothing weight factor – Isoparametric
DFAC	ALE smoothing weight factor – Equipotential
EFAC	ALE smoothing weight factor – Equilibrium
START	Start time for ALE smoothing
END	End time for ALE smoothing
AAFAC	ALE advection factor (donor cell options, default=1.0)
VFACT	Volume fraction limit for stresses in single material and void formulation. All stresses are set to zero for elements with lower volume fraction than VFACT. EQ.0.0: set to default 1.0E-06.
PRIT	A flag to turn on or off the pressure equilibrium iteration option for multi-material elements (See Remark 1). EQ.0: Off (default) EQ.1: On
EBC	Automatic Eulerian boundary condition (See Remark 2). EQ.0: Off EQ.1: On with stick condition EQ.2: On with slip condition

VARIABLE	DESCRIPTION
PREF	Reference pressure applied to the free surfaces of the ALE mesh boundary. (See Remark 3).
NSIDEBC	A node set ID (NSID) which is to be excluded from the EBC constraint.
NCPL	Number of Lagrangian cycles between coupling calculations. This is typically done every cycle; therefore, its default is 1. This is on optional card 3.
NBKT	Number of Lagrangian cycles between global bucket-sort searches to locate the position of the Lagrangian structure (mesh) relative to the ALE fluid (mesh). Default is 50. This is on optional card 3.
IMASCL	A flag for turning ON/OFF mass scaling for ALE parts. The global mass scaling control (parameter DT2MS under *CONTROL_ TIMESTEP card) must be ON. If the run dt is lower than the mass scaling dt, then IMASCL has the following effects: EQ.0: (Default) No mass scaling for ALE parts. Print out maximum 20 warnings. EQ.1: No mass scaling for ALE parts. Stop the run. EQ.2: Do mass scaling for ALE parts (the result may not be correct due to this scaling).
CHECKR	A parameter for reducing or eliminating an ALE pressure locking pattern. It may range from 0.01 to 0.1 (See Remark 4).

Remarks:

1. By default, all materials in a multi-material element are assumed to undergo the same element averaged strain rates. This assumption may not be robust when mixing materials with very different compressibility. In this case, an assumption of pressure equilibrium (PRIT=1) in the element may be more appropriate.

2. This option, used for EULER formulations, automatically defines velocity boundary condition constraints for the user. The constraints, once defined, are applied to all nodes on free surfaces of an Eulerian domain. For problems where the normal velocity of the material at the boundary is zero such as injection molding problems, the automatic boundary condition parameter is set to 2. This will play the same role as the nodal single point constraint. For EBC=1, the material velocity of all free surface nodes of an Eulerian domain is set to zero.

3. The PREF definition is equivalent to using the *LOAD_SEGMENT card to provide pressure loading on the free surfaces of all ALE or Eulerian mesh(es) in a model. This cannot be used to initialize the internal pressure of the material (that must be done via the *EOS_ or *BOUNDARY_AMBIENT_EOS cards).

4. Due to one point integration, ALE elements may experience a spatial instability in the pressure field referred to as checker boarding. CHECKR is a scale for diffusive flux calculation to alleviate this problem.

5. Generally, it is not possible to conserve both momentum and kinetic energy (KE) at the same time. Typically, internal energy (IE) is conserved and KE may not be. This may result in some KE loss (hence, total energy loss). For many analyses this is tolerable, but for airbag application, this may lead to the reduction of the inflating potential of the inflator gas. METH=3 tries to eliminate this loss in KE over the advection step by storing any loss KE under IE, thus conserving total energy of the system.

6. All the smoothing factors (AFAC, BFAC, CFAC, DFAC, EFAC) are generally most applicable to ELFORM=5 (single material ALE formulation).

7. Although this card has many parameters, only a few are required definitions. Typically, one can try, as a first run, setting NADV=1, METH=1, AFAC=-1 and the rest as “0”. Sometimes when needed, PREF may be defined. This is adequate for most cases. METH may be changed to 2 or 3 later depending on the physics of the problem during fine-tuning of the model.

8. Due to the fact that we have internal forces at the nodes, while the pressure is stored at the element center, sometimes there is a "checker-board pattern" in the pressure distribution. It is a kind of locking effect that normally occurs only in problems having very small volumetric strains, i.e., at small pressures. “CHECKR” is designed for alleviating this problem.

*CONTROL_BULK_VISCOSITY

Purpose: Reset the default values of the bulk viscosity coefficients globally. This may be advisable for shock wave propagation and some materials. Bulk viscosity is used to treat shock waves. A viscous term q is added to the pressure to smear the shock discontinuities into rapidly varying but continuous transition regions. With this method the solution is unperturbed away from a shock, the Hugoniot jump conditions remain valid across the shock transition, and shocks are treated automatically.

Card	1	2	3	4	5	6	7	8
Variable	Q1	Q2	TYPE	BTYPE				
Type	F	F	I	I				
Default	1.5	.06	1	0				

VARIABLE	DESCRIPTION
Q1	Default quadratic viscosity coefficient.
Q2	Default linear viscosity coefficient.
TYPE	Default bulk viscosity type, IBQ (Default=1) EQ. -2: standard (also types 2, 4, 10, 16, and 17). With this option the internal energy dissipated by the viscosity in the shell elements is computed and included in the overall energy balance. EQ. -1: standard (also types 2, 4, 10, 16, and 17 shell elements). The internal energy is not computed in the shell elements. EQ.+1: standard. Solid elements only and internal energy is always computed and included in the overall energy balance. EQ.+2: Richards-Wilkins. Two-dimensional plane strain and axisymmetric solid elements only. Internal energy is always computed and included in the overall energy balance.
BTYPE	Beam bulk viscosity type (Default=0) EQ. 0: The bulk viscosity is turned off for beams. EQ. 1: The bulk viscosity is turned on for beam types 1 and 11. The energy contribution is not included in the overall energy balance. EQ. 2: The bulk viscosity is turned on for beam type 1 and 11. The energy contribution is included in the overall energy balance.

Remarks:

The bulk viscosity creates an additional additive pressure term given by:

$$q = \rho l \left(Q_1 l \, \dot{\varepsilon}_{kk}^2 - Q_2 a \dot{\varepsilon}_{kk} \right) \quad \text{if } \dot{\varepsilon}_{kk} < 0$$

$$q = 0 \qquad \text{if } \dot{\varepsilon}_{kk} \geq 0$$

where Q_1 and Q_2 are dimensionless input constants which default to 1.5 and .06, respectively, and l is a characteristic length given as the square root of the area in two dimensions and as the cube root of the volume in three, a is the local sound speed, Q_1 defaults to 1.5 and Q_2 defaults to .06. See Chapter 21 in the LS-DYNA Theory Manual for more details.

The Richards-Wilkins, see [Richards 1965, Wilkins 1976], bulk viscosity considers the directional properties of the shock wave. This has the effect of turning off the bulk viscosity in converging geometries minimizing the effects of "q-heating". The standard option is active whenever the volumetric strain rate is undergoing compression even though no shock waves are present.

***CONTROL_CHECK_**{*OPTION*}

Available options include:

<BLANK>

SHELL

Purpose: Check for various problems in the mesh. Checking is performed during the input phase.

For the SHELL option, shell element integrity checks which have been identified as important in metal forming applications are performed. These checks can improve springback convergence and accuracy. This option will repair bad elements created, for example, during trimming operations.

(*OPTION* = **SHELL**) (include one card for each part to be checked)

Card 1

1	2	3	4	5	6	7	8
Variable	PID	IFAUTO	CONVEX	ADPT	ARATIO	ANGLE	SMIN	
Type	I	I	I	I	F	F	F	
Default	0	0	1	1	0.25	150.0	0.0	

VARIABLE	DESCRIPTION
PID	Part ID to be checked: EQ.0: do not check
IFAUTO	Flag to automatically correct bad elements: EQ.0: write warning message only EQ.1: fix bad element, write message
CONVEX	Check element convexity (internal angles less than 180 degrees) EQ.0: do not check EQ.1: check
ADPT	Check adaptive constraints EQ.0: do not check EQ.1: check
ARATIO	Minimum allowable aspect ratio. Elements which do not meet minimum aspect ratio test will be treated according to IFAUTO above.

VARIABLE	DESCRIPTION
ANGLE	Maximum allowable internal angle. Elements which fail this test will be treated according to IFAUTO above.
SMIN	Minimum element size. Elements which fail this test will be treated according to IFAUTO above.

Remarks:

1. If the convexity test is activated, all failed elements will be fixed regardless of IFAUTO.

2. In addition to illegal constraint definitions (slave which is also a master), checks are performed for mesh connectivities which have been found to cause convergence trouble in implicit springback applications.

*CONTROL_COARSEN

Purpose: Adaptively de-refine (coarsen) a shell mesh by selectively merging four adjacent elements into one. Adaptive constraints are added and removed as necessary.

Card 1

	1	2	3	4	5	6	7	8
Variable	ICOARSE	ANGLE	NSEED	PSID	SMAX			
Type	I	F	I	I	F			
Default	0	none	0	0	0			

Card 2

Variable	N1	N2	N3	N4	N5	N6	N7	N8
Type	I	I	I	I	I	I	I	I
Default	0	0	0	0	0	0	0	0

VARIABLE	DESCRIPTION
ICOARSE	Coarsening flag: EQ.0: do not coarsen (default), EQ.1: coarsen mesh at beginning of simulation for forming model EQ.2: coarsen mesh at beginning of simulation for crash model
ANGLE	Allowable angle change between neighboring elements. Adjacent elements which are flat to within ANGLE degrees are merged. (Suggested starting value = 8.0 degrees)
NSEED	Number of seed nodes (optional). EQ.0: use only automatic searching. EQ.n: also search starting with node IDs given below (maximum = 8 nodes)

VARIABLE	DESCRIPTION
PSID	Part set ID. All the parts defined in this set will be prevented from been coarsened.
SMAX	Maximum element size. For ICOARSE=2, no elements larger than this size will be created.
N1...N8	Optional list of seed node IDs for extra searching. If no seed nodes are specified, leave card 2 blank.

Remarks:

1. Coarsening is performed at the start of a simulation. The first plot state represents the coarsened mesh. By setting the termination time to zero and including the keyword *INTERFACE_SPRINGBACK_LSDYNA a keyword input deck can be generated containing the coarsened mesh.

2. By default, an automatic search is performed to identify elements for coarsening. In some meshes, isolated regions of refinement may be overlooked. Seed nodes can be identified in these regions to assist the automatic search. Seed nodes identify the central node of a four-element group which is coarsened into a single element if the angle criterion is satisfied.

3. The keyword *DEFINE_BOX_COARSEN can be used to indicate regions of the mesh which are protected from coarsening.

*CONTROL_CONTACT

Purpose: Change defaults for computation with contact surfaces.

Card 1

1	2	3	4	5	6	7	8
SLSFAC	RWPNAL	ISLCHK	SHLTHK	PENOPT	THKCHG	ORIEN	ENMASS
F	F	I	I	I	I	I	I
.1	none	1	0	1	0	1	0

Card 2

1	2	3	4	5	6	7	8
USRSTR	USRFRC	NSBCS	INTERM	XPENE	SSTHK	ECDT	TIEDPRJ
I	I	I	I	F	I	I	I
0	0	10-100	0	4.0	0	0	0

Card 3 is optional. The following parameters are the default values used by parts in automatic contacts. These frictional coefficients apply only to contact types: SINGLE_SURFACE, AUTOMATIC_GENERAL, AUTOMATIC_SINGLE_ SURFACE, AUTOMATIC_NODES_TO_..., AUTOMATIC_SURFACE_..., and AUTOMATIC_ ONE_WAY_...., and ERODING_SINGLE_SURFACE. Also see *CONTACT and *PART. *Note that these default values will override the values specified for these contact types in the *CONTACT section.*

Card 3

1	2	3	4	5	6	7	8
SFRIC	DFRIC	EDC	VFC	TH	TH_SF	PEN_SF	
F	F	F	F	F	F	F	
0.0	0.0	0.0	0.0	0.0	0.0	0.0	

Card 4 is optional. If this card is defined, then Card 3 above must be included. A blank card may be inserted for Card 3.

Card 4	1	2	3	4	5	6	7	8
Variable	IGNORE	FRCENG	SKIPRWG	OUTSEG	SPOTSTP	SPOTDEL	SPOTHIN	
Type	I	I	I	I	I	I	F	
Default	0	0	0	0	0	0	inactive	

Card 5 is optional. If this card is defined, then Cards 3 and 4 above must be included. Blank cards may be inserted.

Card 5	1	2	3	4	5	6	7	8
Variable	ISYM	NSEROD	RWGAPS	RWGDTH	RWKSF	ICOV	SWRADF	ITHOFF
Type	I	I	I	F	F	I	F	I
Default	0	0	0	0.	1.0	0	0.	0

Card 6 is optional. If this card is defined, then Cards 3 to 5 above must be included. Blank cards may be inserted.

Card 6	1	2	3	4	5	6	7	8
Variable	SHLEDG	PSTIFF	ITHCNT	TDCNOF	FTALL			
Type	I	I	I	I	I			
Default	0	0	0	0	0			

VARIABLE	DESCRIPTION
SLSFAC	Scale factor for sliding interface penalties, SLSFAC: EQ.0: default = .1.

VARIABLE	DESCRIPTION
RWPNAL	Scale factor for rigid wall penalties, which treat nodal points interacting with rigid walls, RWPNAL. The penalties are set so that an absolute value of unity should be optimal; however, this penalty value may be very problem dependent. If rigid/deformable materials switching is used, this option should be used if the switched materials are interacting with rigid walls. LT.0.0: all nodes are treated by the penalty method. This is required for implicit calculations. Since seven (7) variables are stored for each slave node, only the nodes that may interact with the wall should be included in the node list. EQ.0.0: the constraint method is used and nodal points which belong to rigid bodies are not considered. GT.0.0: rigid bodies nodes are treated by the penalty method and all other nodes are treated by the constraint method.
ISLCHK	Initial penetration check in contact surfaces with indication of initial penetration in output files (see remarks below): EQ.0: the default is set to 1, EQ.1: no checking, EQ.2: full check of initial penetration is performed.
SHLTHK	Shell thickness considered in type surface to surface and node to surface type contact options, where options 1 and 2 below activate the new contact algorithms. The thickness offsets are always included in single surface, constraint method, and automatic surface to surface and node to surface contact types (See remarks below.): EQ.0: thickness is not considered, EQ.1: thickness is considered but rigid bodies are excluded, EQ.2: thickness is considered including rigid bodies.
PENOPT	Penalty stiffness value option. For default calculation of the penalty value please refer to the LS-DYNA Theory Manual. EQ.0: the default is set to 1, EQ.1: minimum of master segment and slave node (default for most contact types), EQ.2: use master segment stiffness (old way), EQ.3: use slave node value, EQ.4: use slave node value, area or mass weighted, EQ.5: same as 4 but inversely proportional to the shell thickness. This may require special scaling and is not generally recommended. Options 4 and 5 can be used for metalforming calculations.
THKCHG	Shell thickness changes considered in single surface contact: EQ.0: no consideration (default), EQ.1: shell thickness changes are included.

VARIABLE	DESCRIPTION
ORIEN	Optional automatic reorientation of contact interface segments during initialization: EQ.0: default is set to 1. EQ.1: active for automated (part) input only. Contact surfaces are given by *PART definitions. EQ.2: active for manual (segment) and automated (part) input. EQ.3: inactive for non-forming contact. EQ.4: inactive for forming contact.
ENMASS	Treatment of the mass of eroded nodes in contact. This option affects all contact types where nodes are removed after surrounding elements fail. Generally, the removal of eroded nodes makes the calculation more stable; however, in problems where erosion is important the reduction of mass will lead to incorrect results. EQ.0: eroding nodes are removed from the calculation. EQ.1: eroding nodes of solid elements are retained and continue to be active in contact. EQ.2: the eroding nodes of solid and shell elements are retained and continue to be active in contact.
USRSTR	Storage per contact interface for user supplied interface control subroutine, see Appendix F. If zero, no input data is read and no interface storage is permitted in the user subroutine. This storage should be large enough to accommodate input parameters and any history data. This input data is available in the user supplied subroutine.
USRFRC	Storage per contact interface for user supplied interface friction subroutine, see Appendix G. If zero, no input data is read and no interface storage is permitted in the user subroutine. This storage should be large enough to accommodate input parameters and any history data. This input data is available in the user supplied subroutine.
NSBCS	Number of cycles between contact searching using three dimensional bucket searches. Defaults recommended.
INTERM	Flag for intermittent searching in old surface-to-surface contact using the interval specified as NSBCS above: EQ.0: off, EQ.1: on.
XPENE	Contact surface maximum penetration check multiplier. If the small penetration checking option, PENCHK, on the contact surface control card is active, then nodes whose penetration then exceeds the product of XPENE and the element thickness are set free, see *CONTACT_*OPTION*_...: EQ.0: default is set to 4.0.

VARIABLE	DESCRIPTION
SSTHK	Flag for using actual shell thickness in single surface contact logic-types 4, 13, 15 and 26. See remarks 1 and 2 below. EQ.0: Actual shell thickness is not used in the contacts. (default), EQ.1: Actual shell thickness is used in the contacts. (sometimes recommended for metal forming calculations).
ECDT	Time step size override for eroding contact: EQ.0: contact time size may control Dt. EQ.1: contact is not considered in Dt determination.
TIEDPRJ	Bypass projection of slave nodes to master surface in types: *CONTACT_TIED_NODES_TO_SURFACE, *CONTACT_TIED_SHELL_EDGE_TO_SURFACE, and *CONTACT_TIED_SURFACE_TO_SURFACE tied interface options: EQ.0: eliminate gaps by projection nodes, EQ.1: bypass projection. Gaps create rotational constraints which can substantially affect results.
SFRIC	Default static coefficient of friction (see *PART_CONTACT)
DFRIC	Default dynamic coefficient of friction (see *PART_CONTACT)
EDC	Default exponential decay coefficient (see *PART_CONTACT)
VFC	Default viscous friction coefficient (see *PART_CONTACT)
TH	Default contact thickness (see *PART_CONTACT)
TH_SF	Default thickness scale factor (see *PART_CONTACT)
PEN_SF	Default local penalty scale factor (see *PART_CONTACT)
IGNORE	Ignore initial penetrations in the *CONTACT_AUTOMATIC options. In the SMP contact this flag is not implement for the AUTOMATIC_ GENERAL option. “Initial” in this context refers to the first timestep that a penetration is encountered. This option can also be specified for each interface on the third optional card under the keyword, *CONTACT. The value defined here will be the default. EQ.0: move nodes to eliminate initial penetrations in the model definition. EQ.1: allow initial penetrations to exist by tracking the initial penetrations. EQ.2: allow initial penetrations to exist by tracking the initial penetrations. However, penetration warning messages are printed with the original coordinates and the recommended coordinates of each slave node given.

VARIABLE	DESCRIPTION
FRCENG	Flag to activate the calculation of frictional sliding energy: EQ.0: do not calculate, EQ.1: calculate frictional energy in contact and store as "Surface Energy Density" in the binary INTFOR file. Convert mechanical frictional energy to heat when doing a coupled thermal-mechanical problem. When PKP_SEN=1 on the keyword card *DATABASE_EXTENT_BINARY, it is possible to identify the energies generated on the upper and lower shell surfaces, which is important in metal forming applications. This data is mapped after each H-adaptive remeshing.
SKIPRWG	Flag not to display stationary rigid wall by default. EQ.0: generate 4 extra nodes and 1 shell element to visualize stationary planar rigid wall. EQ.1: do not generate stationary rigid wall.
OUTSEG	Flag to output each beam spot weld slave node and its master segment for contact type: *CONTACT_SPOTWELD into the D3HSP file. EQ.0: no, do not write out this information. EQ.1: yes, write out this information.
SPOTSTP	If a spot weld node or face, which is related to a *MAT_SPOTWELD beam or solid element, respectively, cannot be found on the master surface, should an error termination occur? EQ.0: no, continue calculation, EQ.1: yes, print error message and terminate, EQ.2: no, delete unconstrained weld and continue calculation.
SPOTDEL	If the nodes of a spot weld beam or solid element are attached to a shell element that fails and are deleted, then the attached spot weld element is deleted if this flag is on. There is a small cost penalty related to this option on non-vector processors. On vector processors, however, this option can significantly slow down the calculation if many weld elements fail since the vector lengths are reduced. EQ.0: no, do not delete the spot weld beam or solid element, EQ.1: yes, delete the weld elements when the attached shells on one side of the element fail.
SPOTHIN	Optional thickness scale factor. If active, define a factor greater than zero, but less than one. Premature failure of spot welds can occur due to contact of the spot welded parts in the vicinity of the spot weld. This contact creates tensile forces in the spot weld. Although this seems physical, the compressive forces generated in the contact are large enough to fail the weld in tension before failure is observed in experimental test. With this option, the thickness of the parts in the vicinity of the weld are automatically scaled, the contact forces do not develop, and the problem is avoided. We recommend setting the IGNORE option to 1 or 2 if SPOTHIN is active. This option applies

VARIABLE	DESCRIPTION
	only to the AUTOMATIC_SINGLE_SURFACE option.
ISYM	Symmetry plane option default for automatic segment generation when contact is defined by part ID's: EQ.0: off, EQ.1: do not include faces with normal boundary constraints (e.g., segments of brick elements on a symmetry plane). This option is important to retain the correct boundary conditions in the model with symmetry.
NSEROD	Flag to use one-way node to surface erosion EQ.0: use two-way algorithm EQ.1: use one-way algorithm
RWGAPS	Flag to add rigid wall gap stiffness, see parameter RWGDTH below. EQ.1: add gap stiffness EQ.2: do not add gap stiffness
RWGDTH	Death time for gap stiffness. After this time the gap stiffness is no longer added.
RWKSF	Rigid wall penalty scale factor for contact with deformable parts during implicit calculations. This value is independent of SLSFAC and RWPNAL. If RWKSF is also specified in *RIGIDWALL_PLANAR, the stiffness is scaled by the product of the two values.
ICOV	Invokes the covariant formulation of Konyukhov and Schweizerhof in the FORMING contact option. This option is available in the third revision of version 971, but is not recommended since it is still being implemented. EQ.0: standard formulation (default) EQ.1: covariant contact formulation.
SWRADF	Spot weld radius scale factor for neighbor segment thinning EQ.0: neighbor segments not thinned (default) GT.0: The radius of beam spot welds are scaled by SWRADF when searching for close neighbor segments to thin.
ITHOFF	Flag for offsetting thermal contact surfaces for thick thermal shells EQ.0: No offset, if thickness is not included in the contact the heat will be transferred between the mid-surfaces of the corresponding contact segments (shells). EQ.1: Offsets are applied so that contact heat transfer is always between the outer surfaces of the contact segments (shells).
SHLEDG	Flag for assuming edge shape for shells when measuring penetration. This is available for segment based contact (see SOFT on *CONTACT) EQ.0: Shell edges are assumed round (default), EQ.1: Shell edges are assumed square and are flush with the nodes

VARIABLE	DESCRIPTION
PSTIFF	Flag to choose the method for calculating the penalty stiffness. This is available for segment based contact (see SOFT on *CONTACT) EQ.0: Based on material density and segment dimensions (default), EQ.1: Based on nodal masses.
ITHCNT	Thermal contact heat transfer methodology LT.0: conduction evevenly distributed (pre R4) EQ.0: default set to 1 EQ.1: conduction weighted by shape functions, reduced intergration EQ.2: conduction weighted by shape functions, full integration
TDCNOF	Tied constraint offset contact update option. EQ.0: Update velocities and displacements from accelerations EQ.1: Update velocities and acclelerations from displacements. This option is recommended only when there are large angle changes where the default does not maintain a constant offset to a small tolerance. This latter option is not as stable as the default and may require additional damping for stability. See *CONTROL_BULK_VISCOSITY and *DAMPING_PART_STIFFNESS.
FTALL	Option to output contact forces to RCFORC for all 2 surface force transducers when the force transducer surfaces overlap. EQ.0: Output to the first force transducer that matches (default) EQ.1: Output to all force transducers that match.

Remarks:

1. The shell thickness change option must be activated in CONTROL_SHELL control input (see ISTUPD) and a nonzero flag specified for SHLTHK above before the shell thickness changes can be included in the surface-to-surface contact types. An additional flag must be set, see THKCHG above, if thickness changes are included in the single surface contact algorithms. The contact algorithms that include the shell thickness are relatively recent and are now fully optimized and parallelized. The searching in these algorithms is considerably more extensive and therefore slightly more expensive.

2. In the single surface contacts types SINGLE_SURFACE, AUTOMATIC_SINGLE_SURFACE, AUTOMATIC_GENERAL, AUTOMATIC_GENERAL_INTERIOR and ERODING_SINGLE_SURFACE, the default contact thickness is taken as the smaller of two values -- the shell thickness or 40% of the minimum edge length. This may create unexpected difficulties if it is the intent to include thickness effects when the in-plane shell element dimensions are less than the thickness. The default is based on years of experience where it has been observed that sometimes rather large nonphysical thicknesses are specified to achieve high stiffness values. Since the global searching algorithm includes the effects of shell thicknesses, it is possible to slow the searches down considerably by using such nonphysical thickness dimensions.

3. The initial penetration check option is always performed in v. 950 irregardless of the value of ISLCHK. If you do not want to remove initial penetrations then set the contact birth time (see *CONTACT_...) so that the contact is not active at time 0.

4. Automatic reorientation requires offsets between the master and slave surface segments. The reorientation is based on segment connectivity and, once all segments are oriented consistently based on connectivity, a check is made to see if the master and slave surfaces face each other based on the right hand rule. If not, all segments in a given surface are reoriented. This procedure works well for non-disjoint surfaces. If the surfaces are disjoint, the AUTOMATIC contact options, which do not require orientation, are recommended. In the FORMING contact options automatic reorientation works for disjoint surfaces.

5. If SPOTHIN is greater than zero and SWRADF is greater than zero, a neighbor segment thinning option is active. The radius of a beam spot weld is scaled by SWRADF, and then a search is made for shell segments that are neighbors of the tied shell segments that are touched by the weld but not tied by it.

6. Segment based contact (see *SOFT on *CONTACT) calculates a penalty stiffness based on the solution time step and the masses of the segments in contact. By default, segment masses are calculated using the material density of the element associated with the segment and the volume of the segment. This method does not take into account added mass introduced by lumped masses or mass scaling and can lead to stiffness that is too low. Therefore, a second method (PSTIFF=1) was added which estimates the segment mass using the nodal masses.

7. Two surface force transducers measure the contact force from any contact interfaces that generate force between the slave and master surfaces of the force transducer. When contact is detected, a search is made to see if the contact force should be added to any 2 surface force transducers. By default, when a force transducer match is found, the force is added and the search terminates. When FTALL=1, the search continues to check for other two surface force transducer matches. This option is useful when the slave and master force transducer surfaces overlap. If there is no overlap, the default is recommended.

*CONTROL_COUPLING

Purpose: Change defaults for MADYMO3D/CAL3D coupling, see Appendix I.

Card 1

1	2	3	4	5	6	7	8
Variable	UNLENG	UNTIME	UNFORC	TIMIDL	FLIPX	FLIPY	FLIPZ	SUBCYL
Type	F	F	F	F	I	I	I	I
Default	1.	1.	1.	0.	0	0	0	1

VARIABLE	DESCRIPTION
UNLENG	Unit conversion factor for length. MADYMO3D/GM-CAL3D lengths are multiplied by UNLENG to obtain LS-DYNA lengths.
UNTIME	Unit conversion factor for time, UNTIME. MADYMO3D/GM-CAL3D time is multiplied by UTIME to obtain LS-DYNA time.
UNFORC	Unit conversion factor for force, UNFORC. MADYMO3D/GM-CAL3D force is multiplied by UNFORC to obtain LS-DYNA force.
TIMIDL	Idle time during which CAL3D or MADYMO is computing and LS-DYNA remains inactive. Important for saving computer time.
FLIPX	Flag for flipping X-coordinate of CAL3D/MADYMO3D relative to the LS-DYNA model: EQ.0: off, EQ.1: on.
FLIPY	Flag for flipping Y-coordinate of CAL3D/MADYMO3D relative to the LS-DYNA model: EQ.0: off, EQ.1: on.
FLIPZ	Flag for flipping Z-coordinate of CAL3D/MADYMO3D relative to the LS-DYNA model: EQ.0: off, EQ.1: on.

VARIABLE	DESCRIPTION
SUBCYL	CAL3D/MADYMO3D subcycling interval (# of cycles): EQ.0: Set to 1, EQ.n: number of LS-DYNA time steps between each CAL3D/MADYMO3D step. Then the position of the contacting rigid bodies is assumed to be constant for n LS-DYNA time steps. This may result in some increase in the spikes in contact, thus this option should be used carefully. As the CAL3D/MADYMO3D programs usually work with a very small number of degrees of freedom, not much gain in efficiency can be achieved.

***CONTROL_CPU**

Purpose: Control cpu time.

Card	1	2	3	4	5	6	7	8
Variable	CPUTIM	IGLST						
Type	F	I						

VARIABLE	DESCRIPTION
CPUTIM	Seconds of cpu time: EQ.0.0: no cpu time limit set
IGLST	Flag for outputting cpu and elapsed times in glstat file EQ.0: no EQ.1: yes

Remarks:

The CPU time limit applies to the current phase of the analysis or restart. The limit is not checked until after the initialization stage of the calculation. Upon reaching the cpu limit, the code will output a restart dump file and terminate. The CPU limit can also be specified on the input control line to LS-DYNA. If a value is specified on both the control line and in the input deck, the minimum value will be used.

*CONTROL_DYNAMIC_RELAXATION

Purpose: Initialize stresses and deformation in a model to simulate a preload. Examples of preload include load due to gravity, load due to a constant angular velocity, and load due to torquing of a bolt. After the preloaded state is achieved by one of three methods described below, the time resets to zero and the normal phase of the solution automatically begins from the preloaded state.

IDRFLG controls the manner in which the preloaded state is computed. If IDRFLG is 1 or -1, a transient 'dynamic relaxation' analysis is invoked in which an explicit analysis, damped by means of scaling nodal velocities by the factor DRFCTR each time step, is performed. When the ratio of current distortional kinetic energy to peak distortional kinetic energy falls below the convergence tolerance (DRTOL) or when the time reaches DRTERM, the dynamic relaxation analysis stops and the current state becomes the initial state of the subsequent normal analysis. Distortional kinetic energy is total kinetic energy less the kinetic energy due to rigid body motion. A history of the distortional kinetic energy computed during the dynamic relaxation phase is automatically written to a file called "relax". This file can be read as an "ASCII" file by LS-PrePost and its data plotted. To create a binary output database having the same format as a d3plot database but which pertains to the dynamic relaxation analysis, use *DATABASE_BINARY_D3DRLF. The output interval is given by this command as an integer representing the number of convergence checks between output states. The frequency of the convergence checks is controlled by the parameter NRCYCK. Dynamic relaxation will be invoked if SIDR is set to 1 or 2 in any of the *DEFINE_CURVE commands, even if there is no *CONTROL_DYNAMIC_RELAXATION command. Curves so tagged are applicable to the preload analysis phase. Curves with SIDR set to 0 or 2 are applicable to the normal phase of the solution. At the conclusion of the dynamic relaxation phase and before the start of the normal solution phase, a binary dump file (d3dump01) and a 'prescribed geometry' file (drdisp.sif) are written by LS-DYNA. Either of these files can be used in a subseuqent analysis to quickly initialize to the preloaded state without having to repeat the dynamic relaxation run. The binary dump file is utilized via a restart analysis (see the *RESTART section of the LS-DYNA Users Manual). The drdisp.sif file is utilized by setting IDRFLG=2 as described below and discussed in Remark 1.

If IDRFLG is set to 2, the preloaded state is quickly reached by linearly ramping nodal displacements, rotations, and temperatures to prescribed values over 100 time steps. See Remark 1.

If IDRFLG is set to 5, an implicit analysis is performed to obtain the preloaded state and in this case, the preload analysis completes when 'time' is equal to DRTERM. The implicit step size is specified with a *CONTROL_IMPLICIT_GENERAL command. The implicit analysis is, by default, static but can be made transient via the *CONTROL_IMPLICIT_DYNAMICS command (see Remark 3).

Card	1	2	3	4	5	6	7	8
Variable	NRCYCK	DRTOL	DRFCTR	DRTERM	TSSFDR	IRELAL	EDTTL	IDRFLG
Type	I	F	F	F	F	I	F	I
Default	250	0.001	0.995	infinity	TSSFAC	0	0.04	0
Remarks				3				1, 2, 3

Define the following card if and only if IDRFLG=3.

	1	2	3	4	5	6	7	8
Variable	DRPSET							
Type	I							
Default	0							
Remarks	4							

VARIABLE	DESCRIPTION
NRCYCK	Number of iterations between convergence checks, for dynamic relaxation option (default = 250).
DRTOL	Convergence tolerance for dynamic relaxation option (default = 0.001).
DRFCTR	Dynamic relaxation factor (default = .995).
DRTERM	Optional termination time for dynamic relaxation. Termination occurs at this time or when convergence is attained (default = infinity).
TSSFDR	Scale factor for computed time step during dynamic relaxation. If zero, the value is set to TSSFAC defined on *CONTROL_TIMESTEP. After converging, the scale factor is reset to TSSFAC.

VARIABLE	DESCRIPTION
IRELAL	Automatic control for dynamic relaxation option based on algorithm of Papadrakakis [1981]: EQ.0: not active, EQ.1: active.
EDTTL	Convergence tolerance on automatic control of dynamic relaxation.
IDRFLG	Dynamic relaxation flag for stress initialization: EQ.-999: dynamic relaxation not activated even if specified on a load curve, see *DEFINE_CURVE, EQ.-1: dynamic relaxation is activated and time history output is produced during dynamic relaxation, see Remark 2. EQ.0: not active, EQ.1: dynamic relaxation is activated, EQ.2: initialization to a prescribed geometry, see Remark 1, EQ.3: dynamic relaxation is activated as with IDRFLG=1, but with a part set ID for convergence checking, EQ.5: initialize implicitly and run explicitly, see Remark 3.

Remarks:

1. When IDRFLG=2, an ASCII file specified by "m=" on the LS-DYNA execution line is read which describes the initialized state. The ASCII file contains each node ID with prescribed values of nodal displacement (x, y, z), nodal rotation (x, y, z) and nodal temperature in (I8, 7E15.0) format.

2. If IDRFLG is set to -1 the dynamic relaxation proceeds as normal but time history data is written to the D3THDT file in addition to the normal data being written to the D3DRLF file. At the end of dynamic relaxation, the problem time is reset to zero. However, information is written to the D3THDT file with an increment to the time value. The time increment used is reported at the end of dynamic relaxation.

3. When IDRFLG=5, LS-DYNA performs an implicit dynamic relaxation by invoking the implicit solver. Parameters for implicit dynamic relaxation can be defined using appropriate CONTROL_IMPLICIT keywords to specify solver type, implicit time step, etc. When using this option, it is required to set appropriate value for DTERM to indicate the termination of the implicit dynamic relaxation. When DTERM is reached during the implicit dynamic relaxation phase, LS-DYNA switches to either implicit or explicit solver depending on IMFLAG in *CONTROL_IMPLICIT_GENERAL. For example, if it is desired to run an implicit dynamic relaxation phase and switch to explicit solver, IMFLAG should be set to 0.

4. When IDRFLG=3, a part set ID is used to check for convergence. For example, if only the tires are being inflated on a vehicle, it may be sufficient in some cases to look at convergence based on the part ID's in the tire and possibly the suspension system.

***CONTROL_EFG**

Purpose: Define controls for the mesh-free computation.

Card 1

Variable	ISPLINE	IDILA	ININT					
Type	I	I	I					
Default	0	0	12					
Remarks			1					

Card 2

Variable	IMLM	ETOL						
Type	I	F						
Default	0	1.eE-4						

VARIABLE	DESCRIPTION
ISPLINE	Optional choice for the mesh-free kernal functions: EQ.0: Cubic spline function (default) EQ.1: Quadratic spline function EQ.2: Cubic spline function with circular disk.
IDILA	Optional choice for the normalized dilation parameter: EQ.0: Maximum distance based on the background element. EQ.1: Maximum distance based on surrounding nodes
ININT	This is the factor needed for the estimation of maximum workspace (MWSPAC) that can be used during the initialization phase.

VARIABLE	DESCRIPTION
IMLM	Optional choice for the matrix operation, linear solving and memory usage: EQ.1: Original BCSLIB-EXT solvers. EQ.2: EFGPACK.
ETOL	Error tolerance in the IMLM. When IMLM=2 is used, ININT in card one becomes redundant. IMLM = 2 is recommended.

Remarks:

1. The mesh-free computation requires calls to use BCSLIB-EXT solvers during the initialization phase. The maximum workspace (MWSPAC) that can be used during the call is calculated according to:

   MWSPAC = ININT**3 * NUMNEFG

   where NUMNEFG is the total number of mesh-free nodes. The ININT default value is 12. This value implicitly tells you how many nodes are within the domain of influence per each node in one direction. Increasing this number when the larger normalized dilation parameters are used.

2. When ISPLINE=2 is used, the input of the normalized dilation parameters (DX, DY, DZ) for the kernel function in *SECTION_SOILD_EFG and SECTIOL_SHELL_EFG only requires DX value.

3. EFGPACK was added to automatically compute the required maximum workspace in the initialization phase and to improve efficiency in the matrix operation, linear solving and memory usage. The original BCSLIB-EXT solver requires an explicit workspace (ININT) for the initialization.

*CONTROL_ENERGY

Purpose: Provide controls for energy dissipation options.

Card	1	2	3	4	5	6	7	8
Variable	HGEN	RWEN	SLNTEN	RYLEN				
Type	I	I	I	I				
Default	1	2	1	1				

VARIABLE	DESCRIPTION
HGEN	Hourglass energy calculation option. This option requires significant additional storage and increases cost by ten percent: EQ.1: hourglass energy is not computed (default), EQ.2: hourglass energy is computed and included in the energy balance. The hourglass energies are reported in the ASCII files GLSTAT and MATSUM, see *DATABASE_*OPTION*.
RWEN	Stonewall energy dissipation option: EQ.1: energy dissipation is not computed, EQ.2: energy dissipation is computed and included in the energy balance (default). The stonewall energy dissipation is reported in the ASCII file GLSTAT, see *DATABASE_*OPTION*.
SLNTEN	Sliding interface energy dissipation option (This parameter is always set to 2 if contact is active. The option SLNTEN=1 is not available.): EQ.1: energy dissipation is not computed, EQ.2: energy dissipation is computed and included in the energy balance. The sliding interface energy is reported in ASCII files GLSTAT and SLEOUT, see *DATABASE_*OPTION*.
RYLEN	Rayleigh energy dissipation option (damping energy dissipation): EQ.1: energy dissipation is not computed (default), EQ.2: energy dissipation is computed and included in the energy balance. The damping energy is reported in ASCII file GLSTAT, see *DATABASE_*OPTION*.

*CONTROL_EXPLOSIVE_SHADOW

Purpose: Compute detonation times in explosive elements for which there is no direct line of sight. If this control card is missing, the lighting time for an explosive element is computed using the distance from the center of the element to the nearest detonation point, L_d; the detonation velocity, D; and the lighting time for the detonator, t_d:

$$t_L = t_d + \frac{L_d}{D}$$

The detonation velocity for this option is taken from the element whose lighting time is computed and does not account for the possibilities that the detonation wave may travel through other explosives with different detonation velocities or that the line of sight may pass outside of the explosive material.

If this control card is present, the lighting time is based on the shortest distance through the explosive material. If inert obstacles exist within the explosive material, the lighting time will account for the extra time required for the detonation wave to travel around the obstacles. The lighting times also automatically accounts for variations in the detonation velocity if different explosives are used. No additional input is required for this control option. This option works for two and three-dimensional solid elements. Also, see *INITIAL_DETONATION and *MAT_HIGH_EXPLOSIVE.

*CONTROL_FORMING_ONESTEP

Purpose: To roughly obtain initial blank size. A three-dimensional part will be unwrap on the flat surface.

Card	1	2	3	4	5	6	7	8
Variable	OPTION							
Type	I							
Default	none							

VARIABLE	DESCRIPTION
OPTION	Method for initial blank guess: EQ.1: Project the 3-D geometry to a 2-D surface. This work for parts without an under cut problem, EQ.2: For parts with under cut.

Remarks:

1. Only *MAT_037, TRANSVERSELY_ANISOTROPIC_ELASTIC_PLASTIC is incorporated into this feature.

*CONTROL_FORMING_POSITION

Purpose: Provides a simple interface for stamping analysis. It is used to position the blank and tooling. When this keyword is used, all the tools must be in the home position, which is the position of the tooling at maximum stroke. From this position each tool will be moved based on this keyword. This keyword is used with the keywords *CONTROL_FORMING_USER and *CONTROL_FORMING_TRAVEL. One *CONTROL_FORMING_POSITION card may be needed for each part.

Card	1	2	3	4	5	6	7	8
Variable	PID	PREMOVE	TARGET					
Type	I	F	I					
Default	none	none	I					

VARIABLE	DESCRIPTION
PID	Part ID
PREMOV	The distance to pre-move the tool in the reverse direction of the movement of the tool.
TARGET	Move part (PID) in the reverse direction of this tool movement, and make sure the minimum distance between PID and TARGET is defined by GAP.

*CONTROL_FORMING_PROJECTION

Purpose: To remove initial penetrations between the blank and the tooling by projecting the penetrated blank (slave) nodes along a normal direction to the surface of the blank with the specified gap between the node and the tooling surface.

Card	1	2	3	4	5	6	7	8
Variable	IDPS	IDPM	GAP	NRSST	NRMST			
Type	I	I	F	I	I			
Default								

VARIABLE	DESCRIPTION
IDPS	Part ID for the blank (slave) side.
IDPM	Part ID for the tool (master) side.
GAP	A distance, which defines the minimum gap required.
NRSST	Normal direction of blank: EQ.0: the normal to the surface of the blank is pointing towards the tool, EQ.1: the normal to the surface of the blank is pointing away from the tool.
NRMST	Normal direction of tool: EQ.0: the normal to the surface of the tool is pointing towards the blank, EQ.1: the normal to the surface of the tool is pointing away from blank.

Remarks:

This option requires consistent normal vectors for both the rigid tooling surface and the blank surface.

*CONTROL_FORMING_TEMPLATE

Purpose: This keyword is used to simplify the required input for sheet metal stamping simulations. With this keyword, five templates are given: three-piece air draw, three-piece toggle draw, four-piece draw, trimming, and springback.

Card 1

Variable	IDTEMP	BLKID	DIEID	PNCH	BNDU	BNDL	TYPE	PREBD
Type	I	I	I	I	I	I	I	F
Default	none	none	none	none	none	none	0	0.0
Remarks	1	2						

Card 2

Variable	LCSS	AL/FE	R00	R45	R90	E	DENSITY	PR
Type	I	C	F	F	F	F	F	F
Default	none	Fe	1.0	R00	R00	none	none	none
Remarks								

Card 3

Variable	K	N	MTYP	UNIT	THICK	GAP	FS	
Type	F	F	I	I	F	F	F	
Default	none	none	37	1	none	1.1t	0.1	
Remarks								

Card 4

Variable	PATERN	VMAX	VX	VY	VZ	VID	AMAX	
Type	I	F	F	F	F	I	F	
Default	1	1000	0	0	-1	none	1.0e+6	
Remarks								

Card 5

Variable	LVLADA	SIZEADA	TIMSADA	D3PLT				
Type	I	F	I	I				
Default	1	none	20	10				
Remarks								

VARIABLE	DESCRIPTION
IDTEMP	Type of forming process (See Remarks below.) EQ.1: 3-piece air-draw EQ.2: 3-piece Toggle-draw EQ.3: 4-piece draw EQ.4: Springback EQ.5: Trimming
BLKID	Part or part set ID (see TYPE) that defines the blank.
DIEID	Part or part set ID that defines the die. See Figures 8.3a, 8.3b and 8.3c for more information
PNCHID	Part or part set ID that defines the punch.
BNDUID	Part or part set ID that defines the upper binder.
BNDLID	Part or part set ID that defines the lower binder.
TYPE	Flag for part or part set ID used in the definition of BLKID, DIEID, PNCHID, BNDUID, and BNDLID. EQ.0: Part ID, EQ.1: Part set ID.
PREBD	Distance between the lower binder and punch in the 4 piece draw. See Figure 8.3c for more information.
LCSS	If the material for the blank is not user defined, this curve ID will define the stress-strain relationship; otherwise, this curve is ignored.
AL/FE	This parameter is used to define the Young's Modulus and density of the blank. If this parameter is defined, E and DENSITY will defined in the units given by UNIT below. EQ.A: the blank is aluminum EQ.F: the blank is steel (default)
R00, R45, R90	Material anisotropic parameters. For transverse anisotropy the R value is set to the average value of R00, R45, and R90.
E	Young's Modulus. If AL/FE is user defined, E is unnecessary
DENSITY	Material density of blank. If AL/FE is user defined, this parameter is unnecessary
PR	Poisson's ratio.
K	Strength coefficient for exponential hardening. If LCSS is defined, or if a blank material is user defined, this parameter is ignored. ($\bar{\sigma} = k\bar{\varepsilon}^n$)

VARIABLE	DESCRIPTION
N	Exponent for exponential hardening. If LCSS is defined, or if a blank material user defined, this parameter is ignored.
MTYP	Material model types 36 and 37 are supported: *MAT_3-PARAMETER_BARAT and *MAT_TRANSVERSELY_ANISOTROPIC_ELASTIC_PLASTIC, respectively.
UNIT	Units adopted in this simulation. Define a number between 1 and 10. See Table 8.1 to determine the value for UNIT. This unit is used to obtain proper punch velocity, acceleration, time step, and material properties.
THICK	Blank thickness. If the blank thickness is already defined, this parameter is ignored.
GAP	The home gap between rigid tools for automatic positioning and tooling motion. If *BOUNDARY_PRESCRIBED_RIGID_BODY is user defined, this parameter is ignored. The default is 1.1 x blank thickness.
FS	Friction coefficient (default=0.10). If the contact is user defined, this parameter is ignored.
PATERN	Velocity profile of moving tool. If the velocity is user defined by *BOUNDARY_PRESCRIBED_RIGID_BODY, PATERN is ignored. EQ.1: Ramped velocity profile EQ.2: Smooth velocity curve
VX, VY, VZ	Vector components defining the direction of the movement of the punch. The default direction is defined by VID
VID	VID is the vector ID defining the direction of the movement of the punch. The vector, VID, overrides the vector (VX, VY, VZ). If VID and (VX, VY, VZ) are undefined, the punch is assumed to move in the negative z-direction.
AMAX	The maximum allowable acceleration.
LVLADA	Maximum adaptive level.
SIZEADA	Minimum element size permitted in the adaptive mesh.
TIMSADA	Total number of adaptive steps during the forming simulation.
D3PLT	The total number of output states in the D3PLOT database.

UNIT	1	2	3	4	5	6	7	8	9	10
Mass	Ton	Gm	Gm	Gm	Gm	Kg	Kg	Kg	Kg	Kg
Length	Mm	Mm	Mm	Cm	Cm	Mm	Cm	Cm	Cm	m
Time	S	Ms	S	Us	S	Ms	Us	Ms	S	S
Force	N	N	1.e-6N	1e7N	Dyne	KN	1e10N	1e4N	1e-2N	N

Table 8.1 Available units for metal stamping simulations.

IDTEMP = 1: 3-Piece Air-Draw

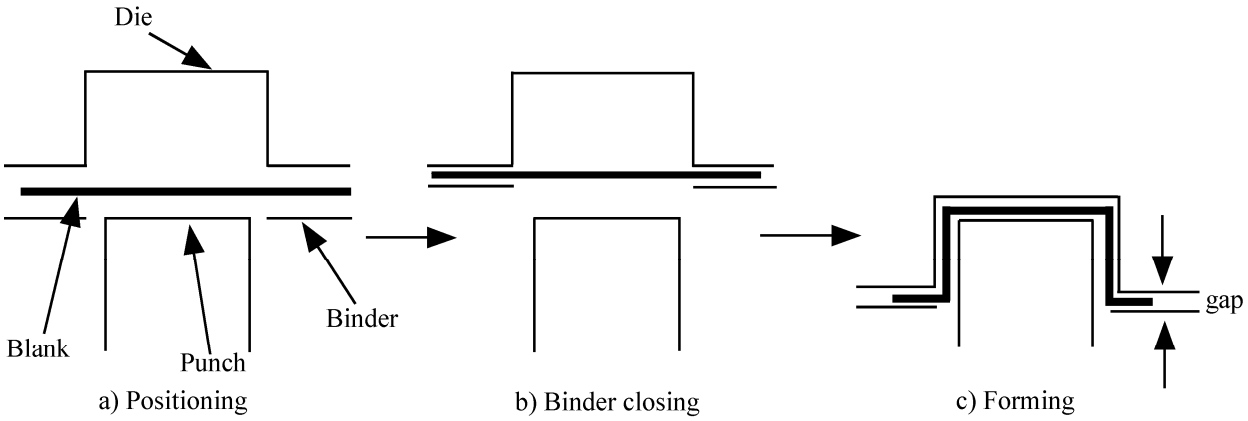


Figure 8.3a

IDTEMP = 2: 3-Piece Toggle Draw

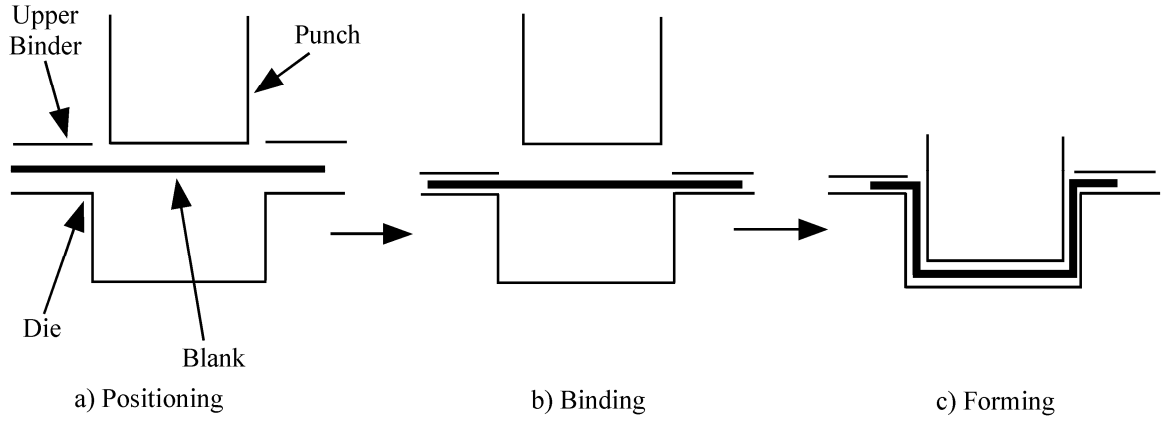


Figure 8.3b

Notes: 1. In step a), LS-DYNA will automatically position the tools and minimize the punch travel.
2. In steps b) and c), LS-DYNA will calculate the binder and punch travel based on the blank thickness and the home gap.
3. Termination time will be automatically set based on b) and c).
4. The rigid body motion of the tooling is automatically defined.
5. All the contacts between the blank and rigid tools are defined.
6. All necessary control parameters are defined.
7. If user defines any of the parameter, it will override the automatic setting
8. User does not need to use keywords, such as *PART, *CONTROL, *SECTION, *MAT_…, *CONTACT_…(Drawbead definition is an exception), *BOUNDARY_PRESCRIPTION_RIGID, etc.

IDTEMP=3: Four-Piece Draw

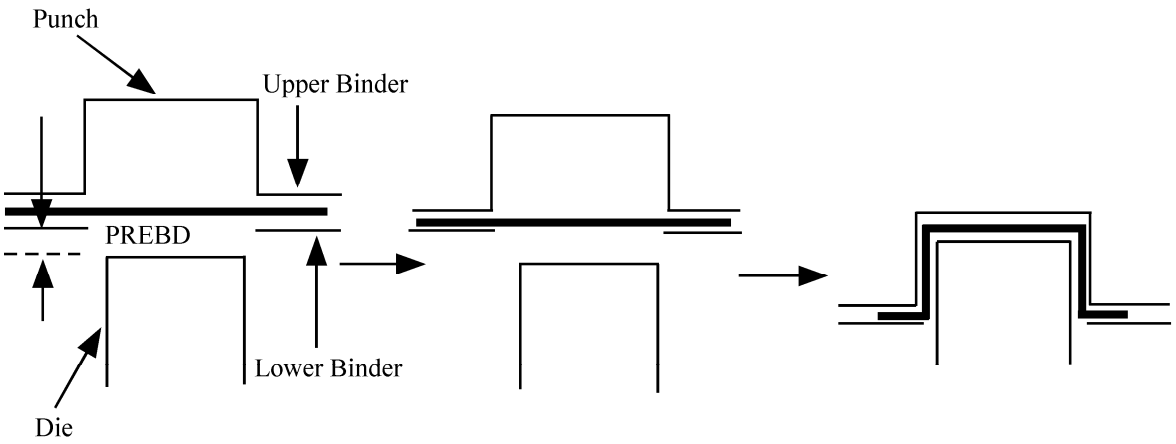


Figure 8.3c

Note: a) PREBD is used for the preposition of the lower
b) First, upper binder goes down to meet lower binder
c) Second, lower binder and upper binder move together and move to the home position
d) Third, punch move down to its home position

IDTEMP=4 Springback Simulation

Note: The necessary keywords, excluding this one, are *BOUNDARY to specify the constraints, and keywords include in dynain file. A new dynain file will be automatically output.

IDTEMP=5 Trimming operation

Note: The necessary keywords, excluding this one, are *DEFINE_CURVE_TRIM, and keywords included in dynain file.

*CONTROL_FORMING_TRAVEL

Purpose: Provide a simple interface for arbitrary stamping analysis. This keyword provides information on how to move the tools in each phase. This keyword should be used with *CONTROL_FORMING_POSITION and *CONTROL_FORMING_USER.

Card 1	1	2	3	4	5	6	7	8
Variable	PID	VID	TRAVEL	TARGET	GAP	PHASE	FOLLOW	
Type	I	I	F	I	F	I	I	
Default	none	none	none	none	none	none	none	

VARIABLE	DESCRIPTION
PID	Part ID of tool.
VID	Vector ID defining the direction of travel for the tool defined by PID.
TRAVEL	Move tool this distance in the direction specified by the VID. If TRAVEL is defined, it is not necessary to define TARGET.
TARGET	Move tool (PID) to meet the TARGET, where the TARGET is the part ID of the matching tool.
GAP	The minimum distance between tool and TARGET in the home position. The GAP is by default the blank thickness.
PHASE	Phase number. Start sequentially from 1. For example, phase 1 is the binder closing, and phase 2 is the stamping operation.
FOLLOW	The tool (PID) can also move by following the part ID, FOLLOW. During this phase, the distance between the tool (PID) and part ID, FOLLOW, will be constant.

***CONTROL_FORMING_USER**

Purpose: To provide a simple interface for arbitrary stamping analysis. It provides blank material information. This keyword should be used with *CONTROL_FORMING_POSITION and *CONTROL_FORMING_TRAVEL.

Card 1

Variable	BLANK	TYPE	THICK	R00	R45	R90	AL/FE	UNIT
Type	I	I	F	F	F	F	A	I
Default	none	0	none	1.0	R00	R00	F	1
Remarks								

Card 2

Variable	LCSS	K	N	E	DENSITY	PR	FS	MTYPE
Type	I	F	F	F	F	F	F	I
Default	none	none	none	none	none	none	0.1	37
Remarks				1	1	1		

Card 3	1	2	3	4	5	6	7	8
Variable	PATERN	VMAX	AMAX	LVLADA	SIZEADA	ADATIMS	D3PLT	GAP
Type	I	F	F	I	F	I	I	F
Default	1	1000.0	500000.	0	0	0	10	1.1t
Remarks								

VARIABLE	DESCRIPTION
BLANK	Blank ID
TYPE	Flag for part or part set ID. EQ.0: Part ID, EQ.1: Part set ID.
THICK	Blank thickness. If the blank thickness is already defined, this parameter is ignored.
R00, R45, R90	Material anisotropic parameters. For transverse anisotropy the R value is set to the average value of R00, R45, and R90.
AL/FE	This parameter is used to define blank Young's Modulus and density. If this parameter is defined, E and Density will be found by using the proper unit, which is specified below. EQ.A: the blank is aluminum EQ.F: the blank is steel (default)
UNIT	Units adopted in this simulation. Define a number between 1 and 10. See Table 8.1 to determine the value for UNIT. This unit is used to obtain proper punch velocity, acceleration, time step, and material properties.
LCSS	If the material for the blank has not been defined, this curve will be used to define the stress-strain relation. Otherwise, this curve is ignored.
PREBD	Distance between the lower binder and punch in the 4 piece draw. See Figure 8.3c for more information.
K	Strength coefficient for exponential hardening. If LCSS is defined, or if a blank material is user defined, this parameter is ignored. ($\bar{\sigma} = k\bar{\varepsilon}^n$)

VARIABLE	DESCRIPTION
N	Exponent for exponential hardening. If LCSS is defined, or if a blank material user defined, this parameter is ignored.
E	Young's Modulus. If AL/FE is user defined, E is unnecessary.
DENSITY	Material density of blank. If AL/FE is user defined, this parameter is unnecessary.
PR	Poisson's ratio. If AL/FE is user defined, E is unnecessary.
FS	Friction coefficient. If contact is defined, this parameter is ignored.
MTYP	Material model types 36 and 37 are supported: *MAT_3-PARAMETER _BARAT and *MAT_TRANSVERSELY_ANISOTROPIC_ELASTIC _PLASTIC, respectively.
PATERN	Velocity profile of moving tool. If the velocity is user defined by *BOUNDARY_PRESCRIBED_RIGID_BODY, PATERN is ignored. EQ.1: Ramped velocity profile EQ.2: Smooth velocity curve
VMAX	The maximum allowable tool velocity
AMAX	The maximum allowable acceleration.
LVLADA	Maximum adaptive level.
SIZEADA	Minimum element size permitted in the adaptive mesh.
ADATIMS	Total number of adaptive steps during the forming simulation.
D3PLT	The total number of output states in the D3PLOT database.
GAP	Minimum gap between tools.

*CONTROL_FREQUENCY_RESPONSE_FUNCTION

Purpose: Set the FRF (frequency response function) controls.

Card1

1	2	3	4	5	6	7	8
N1	N1TYP	DOF1	VAD1	VID	FNMAX	MDMIN	MDMAX
I	I	I	I	I	F	I	I
none	0	none	3	0	0.0	0	0
			3		4	5	

Card 2

1	2	3	4	5	6	7	8
DAMPF	LCDAM	LCTYP	DMPMAS	DMPSTF			
F	I	I	F	F			
0.0	0	0	0.0	0.0			
6							

Card3

Variable	N2	N2TYP	DOF2	VAD2				
Type	I	I	I	I				
Default	none	0	none	2				
Remarks								

Card4

Variable	FMIN	FMAX	NFREQ	RESTRT				
Type	F	F	I	I				
Default	none	none	2	0				
Remarks				8				

VARIABLE **DESCRIPTION**

N1 Node / Node set/Segment set ID for excitation input.

N1TYP Type of N1:
EQ.0: node ID,
EQ.1: node set ID,
EQ.2: segment set ID.

DOF1 Applicable degrees-of-freedom for excitation input:
EQ. ±1: x-translational degree-of-freedom (positive or negative),
EQ. ±2: y-translational degree-of-freedom (positive or negative),
EQ. ±3: z-translational degree-of-freedom (positive or negative),
EQ. ±4: translational movement in direction given by vector VID (positive or negative).

VARIABLE	DESCRIPTION
VAD1	Excitation input type: EQ.0: velocity, EQ.1: acceleration, EQ.2: displacement, EQ.3: nodal force. EQ.4: pressure.
VID	Vector ID for DOF1=4 for excitation input, see *DEFINE_VECTOR.
FNMAX	Optional maximum natural frequency employed in frequency response function computation.
MDMIN	The first mode employed in frequency response function computation. This mode id is optional.
MDMAX	The last mode employed in frequency response function computation. This mode id is optional.
DAMPF	Modal damping coefficient, ζ.
LCDAM	Load Curve ID defining frequency dependent modal damping coefficient, ζ.
LCTYP	Type of load curve defining modal damping coefficient: EQ.0: Abscissa value defines frequency, EQ.1: Abscissa value defines mode number.
DMPMAS	Mass proportional damping constant α, in Rayleigh damping.
DMPSTF	Stiffness proportional damping constant β, in Rayleigh damping.
N2	Node / Node set/Segment set ID for response output.
N2TYP	Type of N2: EQ.0: node ID, EQ.1: node set ID, EQ.2: segment set ID.
DOF2	Applicable degrees-of-freedom for response output: EQ.1: x-translational degree-of-freedom, EQ.2: y-translational degree-of-freedom, EQ.3: z-translational degree-of-freedom.

VARIABLE	DESCRIPTION
VAD2	Response output type: EQ.0: velocity, EQ.1: acceleration, EQ.2: displacement, EQ.3: force.
FMIN	Minimum frequency for frequency response function output.
FMAX	Maximum frequency for frequency response function output.
NFREQ	Number of frequencies for frequency response function output.
RESTRT	Restart option: EQ.0: Initial run, EQ.1: Restart using modeshp.dat and d3eigv created in last run.

Remarks:

1. This command computes the frequency response functions due to nodal excitations.
2. Natural frequencies and mode shapes are needed for computing the frequency response functions. Thus, keyword *CONTROL_IMPLICIT_EIGENVALUE has to be included in input.
3. The FRF (frequency response functions) can be given as Displacement / Force (called Admittance, Compliance, or Receptance), Velocity / Force (called Mobility), Acceleration / Force (called Accelerance, Inertance), Force / Displacement (called Dynamic Stiffness), Force / Velocity (called Mechanical Impedance), Force / Acceleration (called Apparent Mass, Dynamic Mass).
4. FNMAX decides how many natural vibration modes are adopted in the frequency response function computation. LS-DYNA uses only modes with lower or equal frequency than FNMAX in frequency response function computation. If FNMAX is not given, the number of modes in frequency response function computation is same as the number of modes, NEIG, from the *CONTROL_IMPLICIT_EIGENVALUE keyword card, unless MDMIN and MDMAX are prescribed (see remark 5).
5. MDMIN and MDMAX decides which mode(s) are adopted in the frequency response function computation. This option is useful for calculating the contribution from a single mode (MDMIN = MDMAX) or several modes (MDMIN < MDMAX). If only MDMIN is given, LS-DYNA use the single mode (MDMIN) to compute the frequency response function.
6. Damping can be prescribed in several ways:

   To use a constant modal damping coefficient ζ for all the modes, define DAMPF only. LCDMP, LCTYP, DMPMAS and DMPSTF are ignored.

   To use frequency dependent modal damping, define a load curve (*DEFINE_CURVE) and specify that if the abscissa value defines the frequency or mode number by LCTYP. DMPMAS and DMPSTF are ignored.

To use Rayleigh damping, define DMPMAS (α) and DMPSTF (β) and keep DAMPF as 0.0, and keep LCDMP, LCTYP as 0. The damping matrix in Rayleigh damping is defined as $C = \alpha M + \beta K$, where, C, M and K are the damping, mass and stiffness matrices, respectively.

7. To keep rigid body modes from participating in frequency response function computation, set LFLAG=1 and set LFTEND as the tolerance for rigid body eigenvalues in *CONTROL_IMPLICIT_EIGENVALUE.

8. To save time in subsequent runs, user can use the restart option by setting RESTRT=1. LSDYNA will skip the mode analysis and use modeshp.dat and d3eigv generated in the first run, to compute Frequency Response Functions.

***CONTROL_HOURGLASS_**{*OPTION*}

Available options include:

<BLANK>

936

which switches the hourglass formulation so that it is identical to that used in LS-DYNA version 936. The modification in the hourglass control from version 936 was to ensure that all components of the hourglass force vector are orthogonal to rigid body rotations. However, problems that run under version 936 sometimes lead to different results in versions 940 and later. This difference in results is primarily due to the modifications in the hourglass force vector. Versions released after 936 should be more accurate.

Purpose: Set the default values of the hourglass control to override the default values.

Card	1	2	3	4	5	6	7	8
Variable	IHQ	QH						
Type	I	F						
Default		0.1						
Remarks	1							

VARIABLE	DESCRIPTION
IHQ	Default hourglass viscosity type: EQ.1: standard LS-DYNA, EQ.2: Flanagan-Belytschko integration, EQ.3: Flanagan-Belytschko with exact volume integration, EQ.4: stiffness form of type 2 (Flanagan-Belytschko), EQ.5: stiffness form of type 3 (Flanagan-Belytschko), EQ.6: Belytschko-Bindeman [1993] assumed strain co-rotational stiffness form for 2D and 3D solid elements only. This form is available for explicit and IMPLICIT solution methods. In fact, type 6 is mandatory for the implicit options. EQ.8: Applicable to the type 16 fully integrated shell element. IHQ=8 activate warping stiffness for accurate solutions. A speed penalty of 25% is common for this option.

VARIABLE	DESCRIPTION
	EQ.9: Puso [2000] enhanced assumed strain stiffness form for 3D hexahedral elements. This form is available for explicit and implicit solution methods, hence it is an alternative to the Belytschko-Bindeman hourglass type 6 for implicit simulations. In the shell elements, IHQ < 4 is the viscous form based on Belytschko-Tsay. If IHQ = 4, 5 or 6, the stiffness form is obtained. The stiffness forms, however, can stiffen the response, especially if the deformations are large, and therefore should be used with care. For high velocities the viscous forms are recommended and for low velocities the stiffness forms are recommended. For large deformations and non-regular solids, option 3 or 5 is recommended.
QH	Default hourglass coefficient, QH. Values of QH that exceed .15 may cause instabilities. The recommended default applies to all options except for IHQ=6 (See remark 2). For hourglass type 9, see remark 3.

Remarks:

1. Hourglass coefficients and type can be set by part ID using *HOURGLASS and HGID in *PART. These will override values set with *CONTROL_HOURGLASS.

   Starting with release 3 of version 971, the default hourglass type for underintegrated elements is as follows:

   For shells: viscous type (1=2=3) for explicit; stiffness type (4=5) for implicit
   For solids: type 2 for explicit; type 6 for implicit (only type 6 is implemented)

2. Type 6 hourglass control (IHQ=6) is for 2D and 3D solid elements only. If IHG is set to 6, any underintegrated shell element parts that do not have hourglass type defined by ***HOURGLASS** data will be automatically switched to type 4 hourglass control. If this behavior is not desired, it may be better to use ***HOURGLASS** to change individual solid parts to type 6 hourglass control. For a more detailed discussion of type 6 hourglass control, please see Remark 4 in the *HOURGLASS section.

3. Hourglass type 9 is available for hexahedral elements and is based on physical stabilization using an enhanced assumed strain method. In performance it is similar to the Belytschko-Bindeman hourglass formulation (type 6) but gives more accurate results for distorted meshes, e.g., for skewed elements. If QH=1.0, it produces accurate coarse bending results for elastic materials. The hourglass stiffness is by default based on elastic properties, hence the QH parameter should be reduced to about 0.1 for plastic materials in order not to stiffen the structure during plastic deformation. For materials 3, 18 and 24 there is the option to use a negative value of QH. With this option, the hourglass stiffness is based on the current material properties, i.e., the plastic tangent modulus, and scaled by $|\mathrm{QH}|$.

*CONTROL_IMPLICIT_AUTO

Purpose: Define parameters for automatic time step control during implicit analysis (see also *CONTROL_IMPLICIT_GENERAL).

Card	1	2	3	4	5	6	7	8
Variable	IAUTO	ITEOPT	ITEWIN	DTMIN	DTMAX	DTEXP	KFAIL	KCYCLE
Type	I	I	I	F	F	F		
Default	0	11	5	DT/1000.	DT*10.	none		

VARIABLE	DESCRIPTION
IAUTO	Automatic time step control flag EQ.0: constant time step size EQ.1: automatically adjusted time step size
ITEOPT	Optimum equilibrium iteration count per time step. See Figure 8.4.
ITEWIN	Defines range of allowable iteration window. If iteration count is within ITEWIN iterations of ITEOPT, step size will not be adjusted for the next step. In other words, the time step will be reduced if the iteration count to convergence is greater than ITEOPT+ITEWIN and the time step will be increased if the iteration count to convergence is less than ITEOPT-ITEWIN. See Figure 8.4.
DTMIN	Minimum allowable time step size. Simulation stops with error termination if time step falls below DTMIN.
DTMAX	Maximum allowable time step size. LT.0: curve ID = (-DTMAX) gives max step size as a function of time. Also, the step size is adjusted automatically so that the time value of each point in the curve is reached exactly (see Figures 8.5 and 8.6).
DTEXP	Time interval to run in explicit mode before returning to implicit mode. Applies only when automatic implicit-explicit switching is active (IMFLAG= 4 or 5 on *CONTROL_IMPLICIT_GENERAL). Also, see KCYCLE. EQ.0: defaults to the current implicit time step size. LT.0: curve ID = (-DTEXP) gives the time interval as a function of time.

VARIABLE	DESCRIPTION
KFAIL	Number of failed attempts to converge implicitly for the current time step before automatically switching to explicit time integration. Applies only when automatic implicit-explicit switching is active. The default is one attempt. If IAUTO=0, any input value is reset to unity.
KCYCLE	Number of explicit cycles to run in explicit mode before returning to the implicit mode. The actual time interval that is used will be the maximum between DTEXP and KCYCLE*(latest estimate of the explicit time step size).

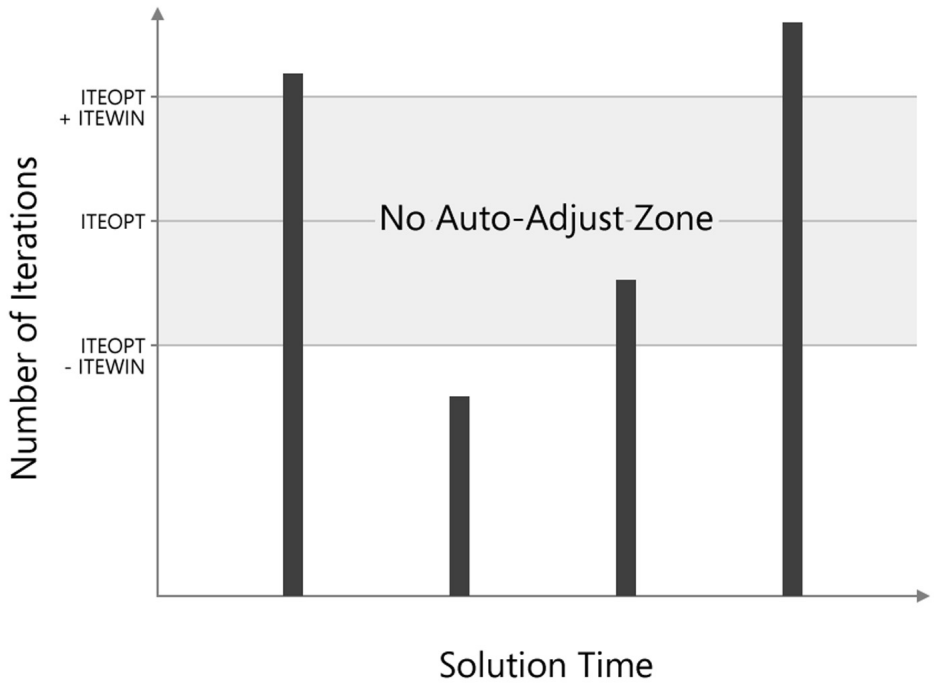


Figure 8.4. Iteration Window as defined by ITEOPT and ITEWIN.

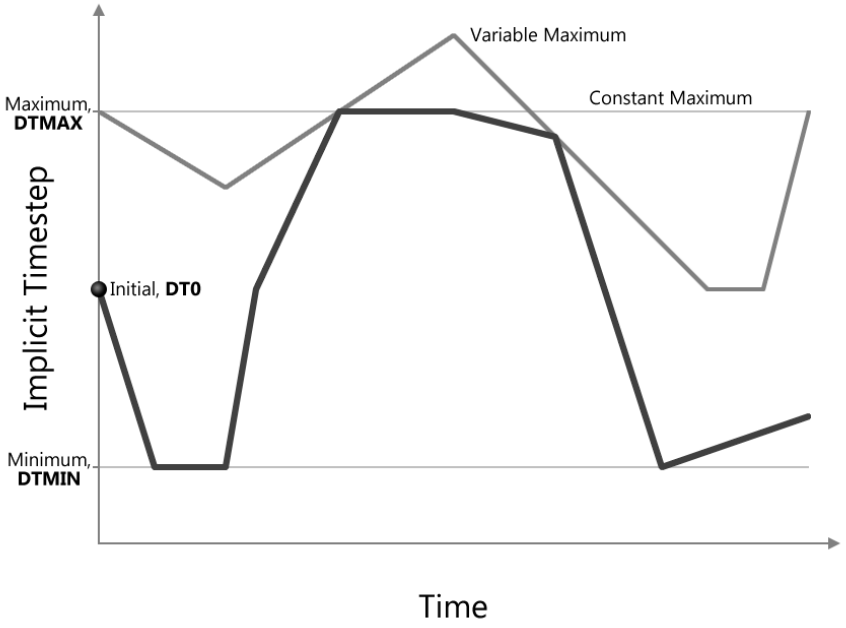


Figure 8.5. The implicit time step size changes continuously as a function of convergence within the bounds set by DTMIN and DTMAX.

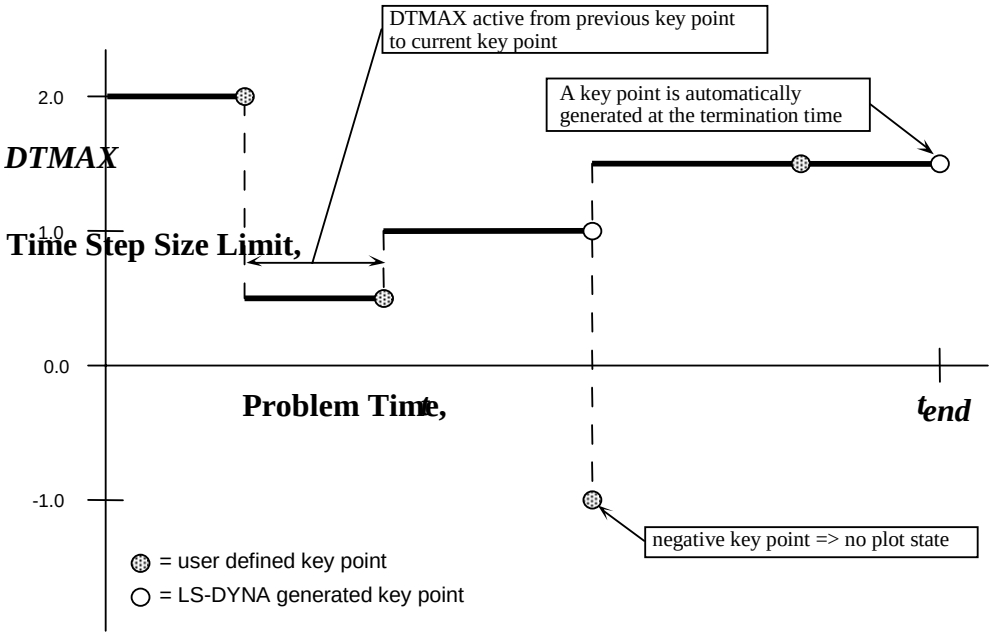


Figure 8.6. A *key point curve* can be identified using a negative value for DTMAX. Function values of each curve point give DTMAX. Time values are reached exactly by the automatic step controller, and a plot state is output unless DTMAX is negative.

Remarks:

IAUTO The default for IAUTO depends on the analysis type. For "springback" analysis, automatic time step control and artificial stabilization are activated by default.

ITEOPT With IAUTO=1, the time step size is adjusted if convergence is reached in a number of iterations that falls outside the specified 'iteration window', increasing after "easy" steps, and decreasing after "difficult" but successful steps. ITEOPT defines the midpoint of the iteration window. A value of ITEOPT=30 or more can be more efficient for highly nonlinear simulations by allowing more iterations in each step, hence fewer total steps.

ITEWIN The step size is not adjusted if the iteration count falls within ITEWIN of ITEOPT. Large values of ITEWIN make the controller more tolerant of variations in iteration count.

DTMAX To strike a particular simulation time exactly, create a key point curve (Figure 8.6) and enter DTMAX = -(curve ID). This is useful to guarantee that important simulation times, such as when peak load values occur, are reached exactly.

DTEXP When the automatic implicit-explicit switching option is activated (IMFLAG = 4 or 5 on *CONTROL_IMPLICIT_GENERAL), the solution method will begin as implicit, and if convergence of the equilibrium iterations fails, automatically switch to explicit for a time interval of DTEXP. A small value of DTEXP should be chosen so that significant dynamic effects do not develop during the explicit phase, since these can make recovery of static equilibrium difficult during the next (static) implicit time step. A reasonable starting value of DTEXP may equal several hundred explicit time steps.

*CONTROL_IMPLICIT_BUCKLE

Purpose: Activate implicit buckling analysis when termination time is reached (see also *CONTROL_IMPLICIT_GENERAL).

Card	1	2	3	4	5	6	7	8
Variable	NMODE							
Type	I							
Default	0							

VARIABLE	DESCRIPTION
NMODE	Number of buckling modes to compute EQ.0: none (DEFAULT) EQ.n: compute n lowest buckling modes

Remarks:

Buckling analysis is performed at the end of a static implicit simulation. The simulation may be linear or nonlinear. After loads have been applied to the model, the buckling eigenproblem is solved:

$$[\mathbf{K}_M + \lambda \mathbf{K}_G]\{u\} = 0$$

where $\mathbf{K}_M$ is the material tangent stiffness matrix, and the geometric or initial stress stiffness matrix $\mathbf{K}_G$ is a function of internal stress in the model. The lowest n eigenvalues and eigenvectors are computed. The eigenvalues, written to text file "eigout", represent multipliers to the applied loads which give buckling loads. The eigenvectors, written to binary database "d3eigv", represent buckling mode shapes. View and animate these modes using LS-PrePost.

The geometric stiffness terms needed for buckling analysis will be automatically computed when the termination time is reached, regardless of the value of the geometric stiffness flag IGS on *CONTROL_IMPLICIT_GENERAL.

A double precision executable should be used for best accuracy in buckling analysis.

*CONTROL_IMPLICIT_CONSISTENT_MASS

Purpose: Use the consistent mass matrix in implicit dynamics and eigenvalue solutions.

Card	1	2	3	4	5	6	7	8
Variable	IFLAG							
Type	I							
Default	0							

VARIABLE	DESCRIPTION
IFLAG	Consistent mass matrix flag EQ.0: Use the standard lumped mass formulation (DEFAULT) EQ.1: Use the consistent mass matrix.

Remarks:

The consistent mass matrix formulation is currently available only for shell elements and the 10 node tetrahedron solid. All other element types continue to use a lumped mass matrix.

*CONTROL_IMPLICIT_DYNAMICS

Purpose: Activate implicit dynamic analysis and define time integration constants (see also *CONTROL_IMPLICIT_GENERAL).

Card	1	2	3	4	5	6	7	8
Variable	IMASS	GAMMA	BETA	TDYBIR	TDYDTH	TDYBUR	IRATE	
Type	I	F	F	F	F	F	I	
Default	0	.50	.25	0.0	1.E+28	1.E+28	0	

VARIABLE	DESCRIPTION
IMASS	Implicit analysis type EQ.0: static analysis EQ.1: dynamic analysis using Newmark time integration. EQ.2: dynamic analysis by modal superposition following the solution of the eigenvalue problem EQ.3: dynamic analysis by modal superposition using the eigenvalue solution in the d3eigv files that are in the runtime directory.
GAMMA	Newmark time integration constant (see remarks below.)
BETA	Newmark time integration constant.
TDYBIR	Birth time for application of dynamic terms. See Figure 8.7. LT.0: curve ID = (-SCALE) used to control amount of implicit dynamic effects applied to the analysis
TDYDTH	Death time for application of dynamic terms.
TDYBUR	Burial time for application of dynamic terms.
IRATE	Rate effects switch: EQ.0: rate effects are on in constitutive models EQ.1: rate effects are off in constitutive models

Remarks:

For the dynamic problem, the linearized equilibrium equations may be written in the form

$$M\ddot{u}^{n+1} + D\dot{u}^{n+1} + K_t\left(x^n\right)\Delta u = P\left(x^n\right)^{n+1} - F\left(x^n\right)$$

where

M = lumped mass matrix

D = damping matrix

$u^{n+1} = x^{n+1} - x^0$ = nodal displacement vector

$\dot{u}^{n+1}$ = nodal point velocities at time n+1

$\ddot{u}^{n+1}$ = nodal point accelerations at time n+1.

Between the birth and death times 100% of the dynamic terms, that is the terms involving M and D, are applied. Between the death and burial time the dynamic terms are decreased linearly with respect to time until 0% of the dynamic terms are applied after the burial time. This feature is useful for problems that are initially singular because the parts are not in contact initially such as in metal stamping. For these problems dynamics is required for stable convergence. When contact is established the problem becomes well conditioned and the dynamic terms are no longer required for stable convergence. It is recommend that for such problems the user set the death time to be after contact is established and the burial time for 2 or 3 time steps after the death time.

For problems with more extensive loading and unloading patterns the user can control the amount of dynamic effects added to the model by using a load curve, see TDYBTH. This curve should have ordinate values between 0.0 and 1.0. The user should use caution in ramping the load curve and the associated dynamic effects from 1.0 to 0.0. Such a ramping down should take place over 2 or 3 implicit time steps.

The time integration is by the unconditionally stable, one-step, Newmark-β time integration scheme

$$\ddot{u}^{n+1} = \frac{\Delta u}{\beta\Delta t^2} - \frac{\dot{u}^n}{\beta\Delta t} - \frac{1}{\beta}\left(\frac{1}{2} - \beta\right)\ddot{u}^n$$

$$\dot{u}^{n+1} = \dot{u}^n + \Delta t\left(1-\gamma\right)\ddot{u}^n + \gamma\Delta t\ddot{u}^{n+1}$$

$$x^{n+1} = x^n + \Delta u$$

Here, Δt is the time step size, and β and γ are the free parameters of integration. For $\gamma = \frac{1}{2}$ and $\beta = \frac{1}{4}$ the method reduces to the trapezoidal rule and is energy conserving. If

$$\gamma > \frac{1}{2}$$

$$\beta > \frac{1}{4}\left(\frac{1}{2} + \gamma\right)^2$$

numerical damping is induced into the solution leading to a loss of energy and momentum.

When modal superposition is invoked, NEIGV on *CONTROL_IMPLICIT_EIGENVALUE indicates the number of modes to be used. With modal superposition, stresses are computed only for linear shell formulation 18.

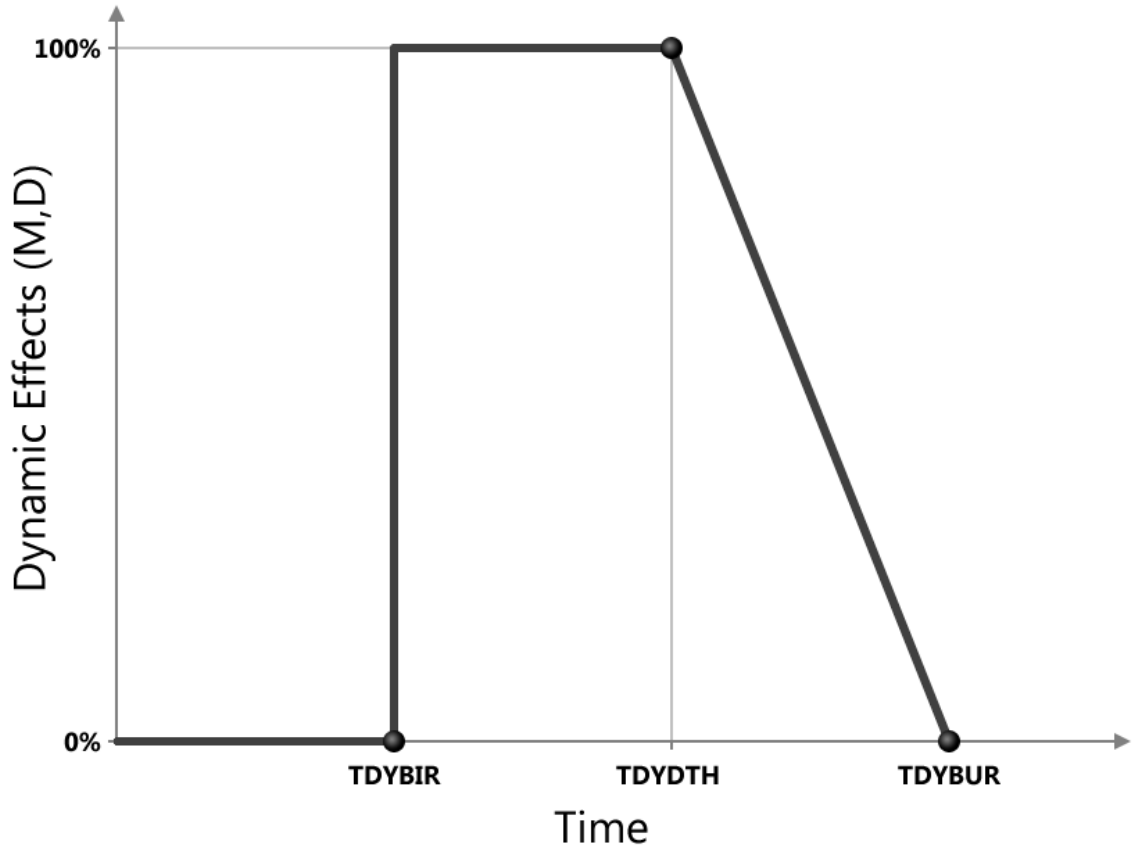


Figure 8.7. Birth, death, and burial time for implicit dynamics.

*CONTROL_IMPLICIT_EIGENVALUE

Purpose: Activate implicit eigenvalue analysis and define associated input parameters (see also *CONTROL_IMPLICIT_GENERAL).

Card	1	2	3	4	5	6	7	8
Variable	NEIG	CENTER	LFLAG	LFTEND	RFLAG	RHTEND	EIGMTH	SHFSCL
Type	I	F	I	F	I	F	I	F
Default	0	0.0	0	-infinity	0	+infinity	2	0.0

Optional Card 2

Card	1	2	3	4	5	6	7	8
Variable	ISOLID	IBEAM	ISHELL	ITSHELL				
Type	I	I	I	I				
Default	0	0	0	0				

VARIABLE	DESCRIPTION
NEIG	Number of eigenvalues to extract. This must be specified. The other parameters below are optional. LT.0: curve ID = (-NEIG) used for intermittent eigenvalue analysis
CENTER	Center frequency. This option finds the nearest NEIG eigenvalues located about this value.
LFLAG	Left end point finite flag. EQ.0: left end point is -infinity EQ.1: left end point is LFTEND.
LFTEND	Left end point of interval. Only used when LFLAG = 1.
RFLAG	Right end point finite flag: EQ.0: right end point is +infinity EQ.1: right end point is RHTEND.

VARIABLE	DESCRIPTION
RHTEND	Right end point of interval. Only used when RFLAG = 1.
EIGMTH	Eigenvalue extraction method: EQ.2: Block Shift and Invert Lanczos (default). EQ.3: Lanczos with [M] = [I] (for debug only). EQ.5: Same as 3 but include Dynamic Terms
SHFSCL	Shift scale. Generally not used, but see explanation below.
ISOLID	If nonzero, reset all solid element formulations to ISOLID for the implicit computations. Can be used for all implicit computations not just eigenvalue computations.
IBEAM	If nonzero, reset all beam element formulations to IBEAM for the implicit computations. Can be used for all implicit computations not just eigenvalue computations.
ISHELL	If nonzero, reset all shell element formulations to ISHELL for the implicit computations. Can be used for all implicit computations not just eigenvalue computations.
ITSHELL	If nonzero, reset all thick shell element formulations to ITSHELL for the implicit computations. Can be used for all implicit computations not just eigenvalue computations.

Remarks:

To perform an eigenvalue analysis, activate the implicit method by selecting IMFLAG=1 on *CONTROL_IMPLICIT_GENERAL, and indicate a nonzero value for NEIG above. By default, the lowest NEIG eigenvalues will be found. If a nonzero center frequency is specified, the NEIG eigenvalues nearest to CENTER will be found.

When NEIG > 0, eigenvalues will be computed at time=0 and LS-DYNA will terminate.

When NEIG < 0, an intermittent eigenvalue analysis will be performed. This is a transient simulation during which loads are applied, with eigenvalues computed periodically during the simulation. Changes in geometry, stress, material, and contact conditions will affect the eigenvalues. The transient simulation can be either implicit or explicit according to IMFLAG=1 or IMFLAG=6, respectively, on *CONTROL_IMPLICIT_GENERAL. The curve ID = -NEIG indicates when to extract eigenvalues, and how many to extract. Define one curve point at each desired extraction time, with a function value equal to the number of eigenvalues desired at that time. A d3plot database will be produced for the transient solution results. Consecutively numbered d3eigv and eigout databases will be produced for each intermittent extraction. The extraction time is indicated in each database's analysis title.

The Block Shift and Invert Lanczos code is from BCSLIB-EXT, Boeing's Extreme Mathematical Library.

When using Block Shift and Invert Lanczos, the user can specify a semifinite or finite interval region in which to compute eigenvalues. Setting LFLAG = 1 changes the left end point from -infinity to the value specified by LFTEND. Setting RFLAG = 1 changes the right end point from +infinity to the values given by RHTEND. If the interval includes CENTER (default value of 0.0) then the problem is to compute the NEIG eigenvalues nearest to CENTER. If the interval does not include CENTER, the problem is to compute the smallest in magnitude NEIG eigenvalues.

If all of the eigenvalues are desired in an interval where both end points are finite just input a large number for NEIG. The software will automatically compute the number of eigenvalues in the interval and lower NEIG to that value. The most general problem specification is to compute NEIG eigenvalues nearest CENTER in the interval [LFTEND,RHTEND]. Computing the lowest NEIG eigenvalues is equivalent to computing the NEIG eigenvalues nearest 0.0.

For some problems it is useful to override the internal heuristic for picking a starting point for Lanczos shift strategy, that is the initial shift. In these rare cases, the user may specify the initial shift via the parameter SHFSCL. SHFSCL should be in the range of first few nonzero frequencies.

Eigenvectors are written to an auxiliary binary plot database named "d3eigv", which is automatically created. These can be viewed using a postprocessor in the same way as a standard "d3plot" database. The time value associated with each eigenvector plot is the corresponding circular frequency. A summary table of eigenvalue results is printed to the "eigout" file. In addition to the eigenvalue results, modal participation factors and modal effective mass tables are written to the “eigout” file.

The print control parameter, LPRINT, and ordering method parameter, ORDER, from the *CONTROL_IMPLICIT_SOLVER keyword card also apply to the Block Shift and Invert Eigensolver.

*CONTROL_IMPLICIT_FORMING

Purpose: This keyword can be used to help convergence in implicit analysis, especially for forming processes, such as gravity loading and binder wrapping. A systematic study has also been conducted to identify the key factors affecting convergence, and the preferred values are automatically set with this keyword. This keyword is mostly suitable for gravity loading and binder wrapping simulations, though it can also be applied to other applications.

Card	1	2	3	4	5	6	7	8
Variable	IOPTION	NSMIN	NSMAX	BIRTH	DEATH	PENCHK		
Type	I	I	I	F	F	F		
Default	1	none	2	0.0	1.e+20	0.0		

VARIABLE	DESCRIPTION
IMFLAG	Solution type: EQ.1: Gravity loading simulation. See remarks 1 and 2 below. EQ.2: Binder wrapping simulation.
NSMIN	Minmum number of implicit steps for IOPTION=2.
NSMAX	Maximum number of implicit steps for IOPTION=2.
BIRTH	Birth time to activate this feature.
DEATH	Death time.
PENCHK	Relative allowed penetration with respect to the part thickness in contact for IOPTION=2.

Remarks:

1. With this keyword, an implicit analysis can be performed without using other implicit control cards in most applications.

2. When gravity loading simulation is selected, it is assumed that the simulation will be done within one implicit step. The other control parameters under this keyword will be ignored.

*CONTROL_IMPLICIT_GENERAL

Purpose: Activate implicit analysis and define associated control parameters. This keyword is required for all implicit analyses.

Card	1	2	3	4	5	6	7	8
Variable	IMFLAG	DT0	IMFORM	NSBS	IGS	CNSTN	FORM	ZERO_V
Type	I	F	I	I	I	I	I	I
Default	0	none	2	1	2	0	0	0

VARIABLE **DESCRIPTION**

IMFLAG — Implicit/Explicit analysis type flag
- EQ.0: explicit analysis
- EQ.1: implicit analysis
- EQ.2: explicit followed by implicit (activates "seamless" *springback*)
- EQ.4: implicit with automatic implicit-explicit switching
- EQ.5: implicit with automatic switching and mandatory implicit finish
- EQ.6: explicit with intermittent eigenvalue extraction
- EQ.-n: curve ID=n gives IMFLAG as a function of time.

DT0 — Initial time step size for implicit analysis

IMFORM — Element formulation flag for "seamless" springback (IMFLAG=2 or *INTERFACE_SPRINGBACK_SEAMLESS)
- EQ.1: switch to fully integrated shell formulation for springback
- EQ.2: retain original element formulation (default)

NSBS — Number of implicit steps in "seamless" springback (IMFLAG=2 or *INTERFACE_SPRINGBACK_SEAMLESS)

IGS — Geometric (initial stress) stiffness flag
- EQ.1: include
- EQ.2: ignore

CNSTN — Indicator for consistent tangent stiffness (solid materials 3 & 115 only):
- EQ.0: do not use (default)
- EQ.1: use.

VARIABLE	DESCRIPTION
FORM	Fully integrated element formulation (IMFLAG=2 and IMFORM=1 only) EQ.0: type 16 EQ.1: type 6.
ZERO_V	Zero out the velocity before switching from explicit to implicit. EQ.0: The velocities are not zeroed out. EQ.1: The velocities are set to zero.

Remarks:

IMFLAG The default value 0 indicates a standard explicit analysis will be performed. Using value 1 causes an entirely implicit analysis to be performed. Value 2 is automatically activated when the keyword *INTERFACE_SPRINGBACK_ SEAMLESS is present, causing the analysis type to switch from explicit to implicit when the termination time is reached. Other nonzero values for IMFLAG can also be used with *INTERFACE_SPRINGBACK_SEAMLESS. After this switch, the termination time is extended by NSBS*DT0, or reset to twice its original value if DT0=0.0. The implicit simulation then proceeds until the new termination time is reached. Contact interfaces are automatically disabled during the implicit phase of "seamless" springback analysis.

When the automatic implicit-explicit switching option is activated (IMFLAG=4 or 5), the solution method will begin as implicit. If convergence of the equilibrium iterations fails, the solution will automatically switch to explicit for a time interval of DTEXP (see *CONTROL_IMPLICIT_AUTO). After this time interval, the solution method will switch back to implicit and attempt to proceed. The implicit simulation may be either static or dynamic. When this feature is used in a static implicit job, simulation time is no longer arbitrary, and must be chosen along with DTEXP in a realistic way to allow efficient execution of any explicit phases. Mass scaling may also be activated (see *CONTROL_TIMESTEP), and will apply only during the explicit phases of the calculation. In cases where much switching occurs, users must exercise caution to ensure that negligible dynamic effects are introduced by the explicit phases.

When IMFLAG=5, the final step of the simulation must be implicit. The termination time will be extended automatically as necessary, until a successfully converged implicit step can be obtained. This is useful for example in difficult metal forming springback simulations.

When IMFLAG=6, an explicit simulation will be performed. Eigenvalues will be extracted intermittently according to a curve indicated by NEIG=(-curve ID) on *CONTROL_IMPLICIT_EIGENVALUE. Beware that dynamic stress oscillations which may occur in the explicit simulation will influence the geometric (initial stress) stiffness terms used in the eigen solution, potentially producing misleading results and/or spurious modes. As an alternative, eigenvalues can also be extracted intermittently during an implicit analysis, using IMFLAG=1 and NEIG=(-curve ID).

When IMFLAG < 0, a curve ID is indicated which gives the solution method as a function of time. Define a curve value of zero during explicit phases, and a value of one during implicit phases. Use steeply sloping sections between phases. An arbitrary number of formulation switches may be activated with this method. See Figure 8.8.

DT0 This parameter selects the initial time step size for the implicit phase of a simulation. The step size may change during a multiple step simulation if the automatic time step size control feature is active (see *CONTROL_IMPLICIT_AUTO.)

IMFORM Adaptive mesh must be activated when using element formulation switching. For best springback accuracy, use of shell type 16 is recommended during the entire stamping and springback analysis, in spite of the increased cost of using this element during the explicit stamping phase.

NSBS The NSBS option allows a "seamless" springback analysis to use multiple unloading steps (*CONTROL_IMPLICIT_STABILIZATION is also required in this case).

IGS The geometric stiffness adds the effect of initial stress to the global stiffness matrix. This effect is seen in a piano string whose natural frequency changes with tension. Geometric stiffness does not always improve nonlinear convergence, especially when compressive stresses are present, so its inclusion is optional.

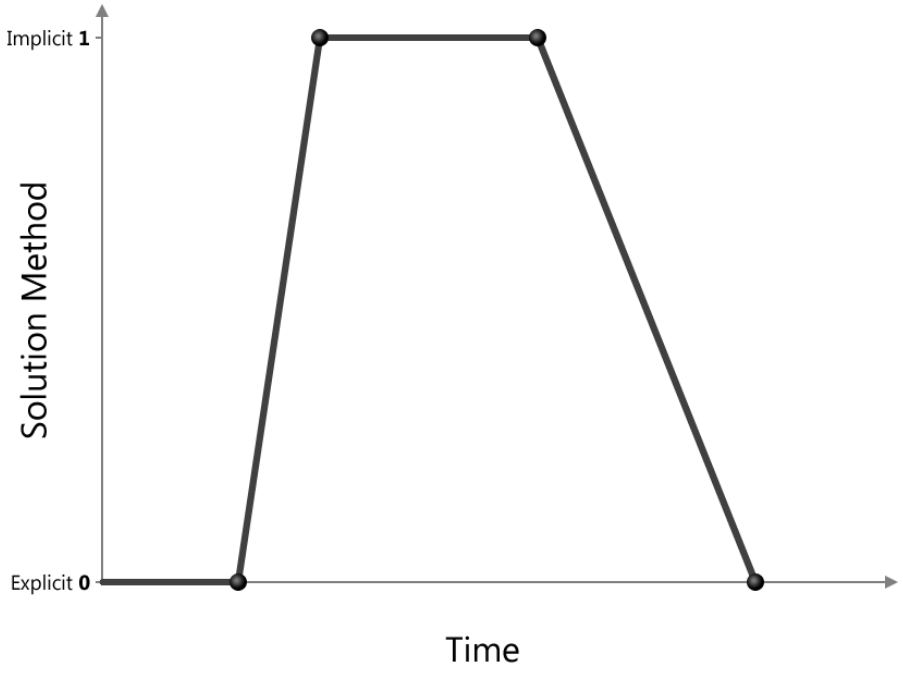


Figure 8.8. Solution method, implicit or explicit, controlled by a load curve.

*CONTROL_IMPLICIT_INERTIA_RELIEF

Purpose: Allows analysis of linear static problems that have rigid body modes.

Card	1	2	3	4	5	6	7	8
Variable	IRFLAG	THRESH						
Type	I	F						
Default	0	0.001						

VARIABLE	DESCRIPTION
IRFLAG	Inertia relief flag EQ.0: do not perform inertia relief EQ.1: do perform inertia relief
THRESH	Threshold for what is a rigid body mode. The default is set to 0.001 Hertz where it is assumed that the units are in seconds.

*CONTROL_IMPLICIT_JOINTS

Purpose: Specify explicit or implicit treatment of joints for implicit analysis.

Card	1	2	3	4	5	6	7	8
Variable	ISPHER	IREVOL	ICYLIN					
Type	I	I	I					
Default	1	1	1					

VARIABLE	DESCRIPTION
ISPHER	Treatment of spherical joints EQ.1: use constraint method for all spherical joints (default) EQ.2: use penalty method for all spherical joints
IREVOL	Treatment of revolute joints EQ.1: use constraint method for all revolute joints (default) EQ.2: use penalty method for all revolute joints
ICYLIN	Treatment of cylindrical joints EQ.1: use constraint method for all cylindrical joints (default) EQ.2: use penalty method for all cylindrical joints

Remarks:

For most implicit applications one should use the constraint (default) method for the treatment of joints. When explicit-implicit switching is used the joint treatment should be consistent. This keyword allows the user to choose the appropriate treatment for their application.

***CONTROL_IMPLICIT_MODES_**{*OPTION*}

Available options include:

<BLANK>

BINARY

Purpose: Request calculation of constraint, attachment, and/or eigen modes for later use in modal analysis using *PART_MODES (see also *CONTROL_IMPLICIT_GENERAL) or *ELEMENT_DIRECT_MATRIX_INPUT.

Card	1	2	3	4	5	6	7	8
Variable	NSIDC	NSIDA	NEIG	IBASE	SE_MASS	SE_DAMP	SE_STIFF	SE_INERT
Type	I	I	I	I	C	C	C	C
Default	0	0						

Card Format (A80)

Card 2

Variable	SE_FILENAME
Type	C

VARIABLE	DESCRIPTION
NSIDC	Node set ID for constraint modes EQ.0: no constraint modes will be generated
NSIDA	Node set ID for attachment modes EQ.0: no attachment modes will be generated
NEIG	Number of eigenmodes EQ.0: no attachment modes will be generated
IBASE	Offset for numbering of the generalized internal degrees of freedom for the superelement

SE_MASS	Name of the superelement mass matrix. If left blank it is not generated.
SE_DAMP	Name of the superelement damping matrix. If left blank it is not generated.
SE_STIFF	Name of the superelement stiffness matrix. If left blank it is not generated.
SE_INERT	Name of the superelement inertia matrix, required for gravity loading applications of the superelement. If left blank it is not generated.
SE_FILENAME	If any of SE_MASS, SE_DAMP, SE_STIFF, or SE_INERT is blank then the second line is required and contains the file name for the superelement.

Remarks:

To use this feature, an implicit analysis must be requested using IMFLAG=1 on *CONTROL_IMPLICIT_GENERAL, and a non-zero termination time must be specified on *CONTROL_TERMINATION. A double precision version of LS-DYNA should be used for best accuracy. Care must be taken to apply a sufficient number of constraints to the model to eliminate static rigid body motion. Computed modes are written to binary output file d3mode, which can be viewed using LS-PREPOST. Eigenmodes are also written to binary output file d3eigv.

Constraint and attachment modes are generated by applying unit displacements and unit forces, respectively, to each specified degree of freedom. By default, modes are computed for all degrees of freedom for each node in sets NSIDC and NSIDA. The first and second node set attribute parameters can be optionally used to restrict the translational and rotational degrees of freedom for which modes are requested, respectively, according to the following syntax:

Node set attribute parameters DA1 and A1: translational degree of freedom codes

Node set attribute parameters DA2 and A2: rotational degree of freedom codes

code	modes computed
0	(See note below.)
1	X degree of freedom only
2	Y degree of freedom only
3	Z degree of freedom only
4	X, Y degrees of freedom only
5	Y, Z degrees of freedom only
6	X, Z degrees of freedom only
7	X, Y, Z degrees of freedom

Setting both node set attributes to zero is equivalent to setting both node set attributes to 7 (X, Y, and Z for translational and rotational degrees of freedom).

If one node set attribute is nonzero (codes 1 to 7) and the other node set attribute is zero, then the zero attribute means NO degrees of freedom are considered. For example, if DA1=2 and DA2=0, then only the Y-translational degree of freedom modes are calculated.

Eigenmodes are generated for the model with single point constraints applied on the constraint modes. The number of eigenmodes is specified here. If the user wants to compute eigenmodes other than the lowest ones, the controls on *CONTROL_IMPLICIT_EIGENVALUE can be used.

When the superelement is created an internal numbering must be applied to the attachment and eigen modes. This numbering starts at IBASE+1.

The user can create the superelement representation of the reduced model by specifying the SE_MASS, SE_DAMP, SE_STIFF, SE_INERT and SE_FILENAME fields. The file, by default is written in the Nastran DMIG file format and can be used as input to *ELEMENT_DIRECT_MATRIX_INPUT. The keyword option _BINARY can be used to create a binary representation for the superelement which can be used with *ELEMENT_DIRECT_MATRIX_INPUT_BINARY to reduce the file size.

The combination of constraint modes and eigenmodes form the Hurty-Craig-Bampton linearization for a model. Using only constraint modes is the same as static condensation.

*CONTROL_IMPLICIT_SOLUTION

Purpose: These optional cards apply to implicit calculations. Use these cards to specify whether a linear or nonlinear solution is desired. Parameters are also available to control the implicit nonlinear and arc length solution methods (see also *CONTROL_IMPLICIT_GENERAL).

Card 1 Format

Card	1	2	3	4	5	6	7	8
Variable	NSOLVR	ILIMIT	MAXREF	DCTOL	ECTOL	RCTOL	LSTOL	ABSTOL
Type	I	I	I	F	F	F	F	F
Default	2	11	15	0.001	0.01	1.0e+10	0.90	1.e-10

Optional Card 2

Card	1	2	3	4	5	6	7	8
Variable	DNORM	DIVERG	ISTIF	NLPRINT	NLNORM	D3ITCTL	CPCHK	
Type	I	I	I	I	I	I	I	
Default	2	1	1	0	2	0	0	

Optional Card 3 (if card 3 is used, then card 2 above must also be used)

Card	1	2	3	4	5	6	7	8
Variable	ARCCTL	ARCDIR	ARCLEN	ARCMTH	ARCDMP	ARCPSI	ARCALF	ARCTIM
Type	I	I	F	I	I	F	F	F
Default	0	none	0	1	2	0.	0.	0.

Optional Card 4 (if card 4 is used, then cards 2 and 3 above must also be used)

Card	1	2	3	4	5	6	7	8
Variable	LSMTD	LSDIR	IRAD	SRAD	AWGT	SRED		
Type	I	I	F	F	F	F		
Default	1	2	0.0	0.0	0.0	0.0		

(See remarks below)

VARIABLE	DESCRIPTION
NSOLVR	Solution method for implicit analysis: EQ.1: Linear EQ.2: Nonlinear with BFGS updates (default) EQ.3: Nonlinear with Broyden updates EQ.4: Nonlinear with DFP updates EQ.5: Nonlinear with Davidon updates EQ.6: Nonlinear with BFGS updates + arclength EQ.7: Nonlinear with Broyden updates + arclength EQ.8: Nonlinear with DFP updates + arclength EQ.9: Nonlinear with Davidon updates + arclength EQ.12:Experimental nonlinear with BFGS updates + optional arclength
ILIMIT	Iteration limit between automatic stiffness reformations
MAXREF	Stiffness reformation limit per time step
DCTOL	Displacement relative convergence tolerance
ECTOL	Energy relative convergence tolerance
RCTOL	Residual (force) relative convergence tolerance (DEFAULT=inactive)
LSTOL	Line search convergence tolerance
ABSTOL	Absolute convergence tolerance.
DNORM	Displacement norm for convergence test EQ.1: Increment vs. displacement over current step EQ.2: Increment vs. total displacement (default)
DIVERG	Divergence flag (force imbalance increase during equilibrium iterations) EQ.1: reform stiffness if divergence detected (default)

VARIABLE	DESCRIPTION
	EQ.2: ignore divergence
ISTIF	Initial stiffness formation flag EQ.1: reform stiffness at start of each step (default) EQ.n: reform stiffness at start of every "n"th step
NLPRINT	Nonlinear solver print flag EQ.0: no nonlinear iteration information printed (new v970 default) EQ.1: print iteration information to screen, message, d3hsp files EQ.2: print extra norm information (NLNORM=1) NOTE: during execution, interactive commands can be used: interactive command — response <ctrl-c> **nlpr**int — toggle NLPRINT between 0 and 1 <ctrl-c> **diag**nostic — toggle NLPRINT between 0 and 2 <ctrl-c> **info**rmation — set NLPRINT=2 for one iteration
NLNORM	Nonlinear convergence norm type EQ.1: consider translational and rotational degrees of freedom EQ.2: consider translational degrees of freedom only (default)
D3ITCTL	Control D3ITER database. If nonzero, the search directions for the nonlinear implicit solution are written to the D3ITER database. To reduce the size of the D3ITER database the database is reset every n time steps where n=D3ITCTL.
CPCHK	Contact penetration check flag EQ.0: no contact penetration is performed (default) EQ.1: check for contact penetration during the nonlinear solution procedure. If such penetration is found modify the line search to prevent unnecessary penetration.
	The following 8 parameters are for use with arc length methods only (6 ≤ NSOLVR ≤ 9 or NSOLVR=12):
ARCCTL	Arc length controlling node ID EQ.0: generalized arc length method
ARCDIR	Arc length controlling node direction (ignored if ARCCTL=0 above) EQ.1: global X-translation EQ.2: global Y-translation EQ.3: global Z-translation
ARCLEN	Relative arc length size. See remarks below.

VARIABLE	DESCRIPTION
	LE.0.0: use automatic size, GT.0.0: use ARCLEN*automatic step size.
ARCMTH	Arc length method EQ.1: Crisfield (default) EQ.2: Ramm EQ.3: Modified Crisfield (used with NSOLVR=12 only)
ARCDMP	Arc length damping option EQ.2: off (default) EQ.1: on, oscillations in static solution are suppressed
ARCPSI	Relative influence of load/time parameter in spherical arclength constraint, default value is 0 which corresponds to a cylindrical arclength constraint. Applies to ARCMTH=3.
ARCALF	Relative influence of predictor step direction for positioning of the arc center, default is 0 which means that the center is at the origin. Applies to ARCMTH=3.
ARCTIM	Optional time when arclength method is initiated. Applies to ARCMTH=3.

VARIABLE	DESCRIPTION
LSMTD	Line search convergence method: EQ.1: Energy method using only translational variables (default) EQ.2: Residual method EQ.3: Energy method using both translational and rotational variables
LSDIR	Line search direction method: EQ.1: Search on all variables (traditional approach used in versions prior to 971) EQ.2: Search only on the independent (unconstrained) variables EQ.3: Use adaptive line search (see AWGT, SRED) EQ.4: Use curved line search (see IRAD, SRAD)
IRAD	Normalized curvature factor for curved line search, where 0 indicates a straight line search and 1 indicates full curved line search.
SRAD	Radius of influence for determining curve in curved line search. For each independent node, all nodes within this radius are used for determining the curve. If 0, then all nodes connected to the same element as the independent node are used.
AWGT	Adaptive line search weight factor between 0 and 1. A high value tends to restrict the motion of oscillating nodes during the implicit process.
SRED	Initial step reduction between 0 and 1 for adaptive line search, use large number for conservative start in implicit procedure.

Remarks:

NSOLVR If a linear analysis is selected, equilibrium checking and iterations are not performed.

The Full Newton nonlinear solution method can be invoked by using the default BFGS solver, and selecting ILIMIT=1 to form a new stiffness matrix every iteration.

In the neighborhood of limit points the Newton based iteration schemes often fail. The arc length method of Riks and Wempner (combined here with the BFGS method) adds a constraint equation to limit the load step to a constant "arc length" in load-displacement space. This method is frequently used to solve snap through buckling problems. When applying the arc-length method, the curves that define the loading should contain only two points, and the first point should be at the origin (0,0). LS-DYNA will extrapolate, if necessary, to determine the load. In this way, time and load magnitude are related by a constant. It is possible that time can become negative in case of load reversal. The arc length method cannot be used in a dynamic analysis.

ILIMIT In the default BFGS method, the global stiffness matrix is only reformed every ILIMIT iterations. Otherwise, an inexpensive stiffness update is applied. By setting ILIMIT=1, a stiffness reformation is performed every iteration. This is equivalent to the Full Newton method (with line search). A higher value of ILIMIT (20-25) can reduce the number of stiffness matrix reformations and factorizations which may lead to a significant reduction in cost. Note that the storage requirements for implicit include storing 2 vectors per iteration. Large values of ILIMIT will cause substantial increase in storage requirements.

MAXREF The nonlinear equilibrium search will continue until the stiffness matrix has been reformed MAXREF times, with ILIMIT iterations between each reformation. If equilibrium has not been found, control will be passed to the automatic time step controller if it is activated. Otherwise, error termination will result. When the auto time step controller is active, it is often efficient to choose MAXREF=5 and try another stepsize quickly, rather than wasting too many iterations on a difficult step.

DCTOL When the displacement norm ratio is reduced below DCTOL, this condition is satisfied. Smaller numbers lead to more accurate determination of equilibrium and, on the negative side, result in more iterations and higher costs. Use NLPRINT to display norm data each iteration.

ECTOL When the energy norm ratio is reduced below ECTOL, this condition is satisfied. Smaller numbers lead to more strict determination of equilibrium and, on the negative side, result in more iterations and higher costs. Use NLPRINT to display norm data each iteration.

RCTOL When the residual norm ratio is reduced below RCTOL, this condition is satisfied. Smaller numbers lead to more strict determination of equilibrium and, on the negative side, result in more iterations and higher costs. By default this convergence criterion is effectively disabled using RCTOL=1.e10. Use NLPRINT to display norm data each iteration.

LSTOL A line search is performed on stiffening systems to guard against divergence of Newton-based nonlinear solvers. With the Full Newton method, it is sometimes helpful to define a large value (LSTOL=9999.0) to effectively disable line search.

DNORM When computing the displacement ratio, the norm of the incremental displacement vector is divided by the norm of “total” displacement. This “total” displacement may be either the total over the current step, or the total over the entire simulation. The latter tends to be more lax, and can be poor at the end of simulations where large motions develop. For these problems, an effective combination is DNORM=1, and DCTOL=0.01 or larger.

DIVERG By default, a new stiffness matrix is formed whenever divergence (growing out-of-balance force) is detected. This flag can be used to suppress this stiffness reformation.

ISTIF By default, a new stiffness matrix is formed at the start of every time step. Suppressing this stiffness reformation can decrease the cost of simulations which have many tiny steps that are mostly linear, such as transient dynamics.

NLPRINT This flag controls printing of displacement and energy convergence measures during the nonlinear equilibrium search. If convergence difficulty occurs, this information is helpful in determining the problem.

NLNORM By default, only translational degrees of freedom are used in evaluating convergence norms. Use this flag to include rotational degrees of freedom, or to make additional data available for diagnosing convergence problems. This additional data includes the worst offending node and degree of freedom contributing to each norm.

ARCCTL The arc length method can be controlled based on the displacement of a single node in the model. For example, in dome reversal problems the node at the center of the dome can be used. By default, the generalized arc length method is used, where the norm of the global displacement vector controls the solution. This includes all nodes.

ARCLEN In many cases the arc length method has difficulty tracking the load displacement curve through critical regions. Using 0<ARCLEN<1 will reduce the step size to assist tracking the load-displacement curve with more accuracy. Use of ARCLEN<1 will cause more steps to be taken. Suggested values are 1.0 (the default), 0.5, 0.25, and 0.10.

ARCDMP Some static problems exhibit oscillatory response near instability points. This option numerically suppresses these oscillations, and may improve the convergence behavior of the post-buckling solution.

LSMTD The default method for determining convergence of the nonlinear line search is to find the minimum of the energy. This parameter allows choosing the energy on only the translational variables, energy of both the translational and rotational variables, or for minimizing the residual (forces). The effect of using a residual based line search is not always positive, sometimes it is too restrictive and stops convergence. However, it is a more conservative approach than using the energy based method since it explicitly controls the norm of the residual. It should not be seen as a better strategy than the energy method but as an alternative to try in cases when the default method seems to be working poorly.

LSDIR In Version 971 of LS-DYNA new line search options were added. The traditional approach (LSDIR=1) computes the line search direction using all variables. The new (default) approach of LSIDR=2 computes the line search direction only on the unconstrained variables. It has proven to be both robust and more efficient. We have also included two new approaches to try for problems where the default and traditional approach fail and the user is using Full Newton (ILIMIT=1). See the next two remarks for more information on those methods.

IRAD, SRAD The parameters IRAD and SRAD are for the curved line search (LSDIR=4). The first parameter is a switch (0 or 1) to invoke this line search, an intermediate value is interpreted as weighted combination of a straight and curved line search (the curvature radius is decreased with increasing IRAD). A value of unit is recommended in situations with rather smooth responses, e.g. springback and similar problems. Also, IRAD=1 seems to work best with full Newton iterations. The SRAD parameter should be equal to 0 for most cases, this means that the search curve for a node is determined from the search direction of nodes connected to the same elements as that node. SRAD>0 is interpreted as a radius of influence, meaning that the search curve for a node is determined from the search direction of nodes within a distance SRAD of this node. This option was introduced as an experiment to see if this had a smoothing and stabilizing effect. A value of 0.0 is currently recommended.

AWGT, SRED The parameters AWGT and SRED are for the adaptive line search. The intention is to improve robustness for problems that have tendencies to oscillate or diverge, indicated by the dnorm and enorm parameter outputs in the iterations (stdout). A value of 0.5 is recommended for AWGT as a starting point. With a nonzero value the motions of individual nodes are tracked. For nodes that are oscillating (going back and forth in space), the maximum step size for the next iteration is reduced in proportion to the parameter AWGT, and for nodes that are not oscillating but going nicely along a straight path, the maximum step size for the next iteration is increased in proportion to 1-AWGT. In test problems, the introduction of the adaptive line search has stabilized the implicit procedure in the sense that the dnorm and enorm values are more monotonically decreasing until convergence with virtually no oscillations. If a problem is still oscillating or diverging, the user should try to increase the AWGT parameter since this is a more restrictive approach but probably gives a slower convergence rate. An option for nasty problems is also to use SRED>0 which is the initial step reduction factor (less than 1). This means that the initial step size is reduced by this value but the maximum step size will increase by an amount that is determined by the success in the iterative procedure, eventually it will reach unity. It can never decrease. Also here, it is intended to be used with full Newton method.

*CONTROL_IMPLICIT_SOLVER

Purpose: These optional cards apply to implicit calculations. The linear equation solver performs the CPU-intensive stiffness matrix inversion (see also *CONTROL_IMPLICIT_ GENERAL).

Card	1	2	3	4	5	6	7	8
Variable	LSOLVR	LPRINT	NEGEV	ORDER	DRCM	DRCPRM	AUTOSPC	AUTOTOL
Type	I	I	I	I	I	F	I	F
Default	4	0	2	0	1	see below	1	see below

Optional Card 2

Card	1	2	3	4	5	6	7	8
Variable	LCPACK	MTXDMP						
Type	I	I						
Default	2	0						

(See remarks below)

VARIABLE	DESCRIPTION
LSOLVR	Linear equation solver method EQ.4: SMP parallel multi-frontal sparse solver (default). EQ.5: SMP parallel multi-frontal sparse solver, double precision EQ.6: BCSLIB-EXT, direct, sparse, double precision EQ.10: iterative, best of currently available iterative methods EQ.11: iterative, Conjugate Gradient method EQ.12: iterative, CG with Jacobi preconditioner EQ.13: iterative, CG with Incomplete Choleski preconditioner EQ.14: iterative, Lanczos method EQ.15: iterative, Lanczos with Jacobi preconditioner EQ.16: iterative, Lanczos with Incomplete Choleski preconditioner

VARIABLE	DESCRIPTION
LPRINT	Linear solver print flag controls screen and message file output EQ.0: no printing EQ.1: output summary statistics on memory, cpu requirements EQ.2: more statistics EQ.3: even more statistics and debug checking (NOTE: during execution, use the interactive command "<ctrl-c> lprint" to toggle this print flag between 0 and 1.)
NEGEV	Negative eigenvalue flag. Selects procedure when negative eigenvalues are detected during stiffness matrix inversion. EQ.1: stop, or retry step if auto step control is active EQ.2: print warning message, try to continue (default)
ORDER	Ordering option EQ.0: method set automatically by LS-DYNA EQ.1: MMD, Multiple Minimum Degree. EQ.2: Metis
DRCM	Drilling rotation constraint method: EQ.1: add stiffness (old default in Version 970) EQ.2: add no stiffness EQ.3: add no stiffness EQ.4: add stiffness (new default in Version 971 Release 2)
DRCPRM	Drilling rotation constraint parameter, DRCPRM. If adding stiffness, DRCM=1, then, for linear problems, DRCPRM=1.0; for nonlinear problems, DRCPRM=100.0; and for eigenvalue problems either 1.E-12 or 1.E-8 is used depending on the shell element type. In the latter case, the input value for DRCPRM is ignored.
AUTOSPC	Automatic Constraint Scan flag EQ.1: scan the assembled stiffness matrix looking for unconstrained, unattached degrees of freedom. Generate additional constraints as necessary to avoid negative eigenvalues. EQ.2: do not add constraints.
AUTOTOL	AUTOSPC tolerance. The test for singularity is the ratio of the smallest singular value and the largest singular value. If this ratio is less than AUTOTOL, then the triple of columns are declared singular and a constraint is generated. Default value in single precision is 1.E-4 and in double precision, 1.E-8.
LCPACK	Matrix assembly package. EQ.2: Use v970's LCPACK (default, only available option in 971)

VARIABLE	DESCRIPTION
MTXDMP	Matrix and right-hand-side dumping. To assist in evaluating other linear equation solution package, LS-DYNA has the option of dumping the globally assembled stiffness matrix and right-hand-side vectors to ASCII files in Harwell-Boeing sparse matrix format. EQ.0: No dumping EQ.1: Dump all matrices and right-hand-side vectors every MTXDMP time steps. Filenames of the form K.matrix.xxxx, M.matrix.xxxx, and RHS.vector.xxxx are used.

Remarks:

LSOLVR The linear solver is used to compute the inverse of the global stiffness matrix, which is a costly procedure both in memory and cpu time. Direct solvers apply Gaussian elimination, while iterative solvers successively improve “guesses” at the correct solution. Iterative solvers require far less memory than direct solvers, but may suffer from convergence problems. Generally, iterative solvers are poor for automotive applications, but can be superior for large brick element soil models in civil engineering.

Solvers 5 and 6 promote the global matrix to double precision before factoring to reduce numerical truncation error. Solvers 4 and 5 are equivalent if a double precision executable is used.

Solver 6 is the direct linear equation solver from BCSLIB-EXT, Boeing's Extreme Mathematical Library. This option should be used whenever the factorization is too large to fit into memory. It has extensive capabilities for out-of-core solution and can solve larger problems than any of the other direct factorization methods. Solver 6 also includes a sophisticated pivoting strategy which can be superior for nearly singular matrices.

LPRINT Select printing of the timing and storage information (LPRINT = 1) if you are comparing performance of linear equation solvers, or if you are running out of memory for large models. Minimum memory requirements for in-core and out-of-core solution are printed. This flag can also be toggled using sense switch "<ctrl-c> lprint". *For best performance, increase available memory using “memory=“ on the command line until an IN-CORE solution is indicated.*

When using solver option 6, LPRINT = 2 and 3 will cause increased printed output of statistics and performance information.

NEGEV Negative eigenvalues result from underconstrained models (rigid body modes), severely deformed elements, or non-physical material properties. This flag allows control to be passed directly to the automatic time step controller when negative eigenvalues are detected. Otherwise, significant numerical roundoff error is likely to occur during factorization, and equilibrium iterations may fail (see *CONTROL_IMPLICIT_AUTO).

ORDER The system of linear equations is reordered to optimize the sparsity of the factorization when using direct methods. Metis is a ordering method from University of Minnesota which is very effective for larger problems and for 3D solid problems, but also very expensive. MMD is inexpensive, but may not produce an optimum reordering, leading to higher cost during numeric factorization. MMD is usually best for smaller problems (less than 100,000 degrees of freedom).

Reordering cost is included in the symbolic factorization phase of the linear solver (LPRINT.ge.1). For large models, if this cost exceeds 20% of the numeric factorization cost, it may be more efficient to select the MMD method.

Note that the values of LPRINT and ORDER also affect the eigensolution software. That is LPRINT and ORDER from this keyword card is applicable to eigensolution.

*CONTROL_IMPLICIT_STABILIZATION

Purpose: This optional card applies to implicit calculations. Artificial stabilization is required for multi-step unloading in implicit springback analysis (see also *CONTROL_IMPLICIT_ GENERAL).

Card	1	2	3	4	5	6	7	8
Variable	IAS	SCALE	TSTART	TEND				
Type	I	F	F	F				
Default	2	1.0	(see below)	(see below)				

VARIABLE	DESCRIPTION
IAS	Artificial Stabilization flag EQ.1: active EQ.2: inactive (default)
SCALE	Scale factor for artificial stabilization. For flexible parts with large springback, like outer body panels, a value of 0.001 may be required. EQ.-n: curve ID = n gives SCALE as a function of time
TSTART	Start time. (Default: immediately upon entering implicit mode)
TEND	End time. (Default: termination time)

Remarks:

Artificial stabilization allows springback to occur over several steps. This is often necessary to obtain convergence during equilibrium iterations on problems with large springback deformation. Stabilization is introduced at the start time TSTART, and slowly removed as the end time TEND is approached. Intermediate results are not accurate representations of the fully unloaded state. The end time TEND must be reached exactly for total springback to be predicted accurately.

IAS The default for IAS depends on the analysis type in *CONTROL_IMPLICIT_ GENERAL. For “seamless” springback analysis, automatic time step control and artificial stabilization are activated by default. Otherwise, IAS is inactive by default.

SCALE	This is a penalty scale factor similar to that used in contact interfaces. If modified, it should be changed in order-of-magnitude increments at first. Large values suppress springback deformation until very near the termination time, making convergence during the first few steps easy. Small values may not stabilize the solution enough to allow equilibrium iterations to converge.

***CONTROL_IMPLICIT_STATIC_CONDENSATION_**{*OPTION*}

Available options include:

<BLANK>

BINARY

Purpose: Request static condensation of a part to build a reduced linearized model for later computation with *ELEMENT_DIRECT_MATRIX_INPUT. Optionally the analysis can continue using the linearization for the current analysis.

Card	1	2	3	4	5	6	7	8
Variable	SC_FLAG	SC_NSID	SC_PSID	SE_MASS	SE_STIFF	SE_INERT		
Type	I	I	I	C	C	C		
Default	0	0	0					

Card Format (A80)

Card 2

Variable	SE_FILENAME
Type	C

VARIABLE	DESCRIPTION
SC_FLAG	Static Condensation Control Flag EQ.0: no static condensation will be performed EQ.1: create superelement representation based on static condensation. EQ.2: use static condensation to build a linearized representation for a part and use that linearized representation in the following analysis.
SC_NSID	Node set ID for nodes to be preserved in the static condensation procedure. Required when SC_FLAG = 1.

SC_PSID	Part set ID for parts to be included in the static condensation procedure. When SC_FLAG = 1 SC_PSID can be used to specify a subset of the model with the default being the entire model. When SC_FLAG = 2 SC_PSID is required. SC_PSID = 0 implies that the entire model is condensed.
SE_MASS	Name of the superelement mass matrix. If left blank it is not generated.
SE_STIFF	Name of the superelement stiffness matrix. If left blank it is not generated.
SE_INERT	Name of the superelement inertia matrix, required for gravity loading applications of the superelement. If left blank it is not generated.
SE_FILENAME	If any of SE_MASS, SE_DAMP, SE_STIFF, or SE_INERT is blank then the second line is required and contains the file name for the superelement.

Remarks:

To use this feature, an implicit analysis must be requested using IMFLAG=1 on *CONTROL_IMPLICIT_GENERAL, and a non-zero termination time must be specified on *CONTROL_TERMINATION. A double precision version of LS-DYNA should be used for best accuracy. The superelement model is written to file SE_FILENAME.

Static condenstation is the reduction of the global stiffness and mass matrices to a specified sets of rows and columns associated with the nodes in the node set SC_NSID. The first and second node set attribute parameters can be optionally used to restrict the translational and rotational degrees of freedom for which modes are requested, respectively, according to the following syntax:

Node set attribute parameters DA1 and A1: translational degree of freedom codes

Node set attribute parameters DA2 and A2: rotational degree of freedom codes

code	modes computed
0	(See note below.)
1	X degree of freedom only
2	Y degree of freedom only
3	Z degree of freedom only
4	X, Y degrees of freedom only
5	Y, Z degrees of freedom only
6	X, Z degrees of freedom only
7	X, Y, Z degrees of freedom

Setting both node set attributes to zero is equivalent to setting both node set attributes to 7 (X, Y, and Z for translational and rotational degrees of freedom).

If one node set attribute is nonzero (codes 1 to 7) and the other node set attribute is zero, then the zero attribute means NO degrees of freedom are considered. For example, if DA1=2 and DA2=0, then only the Y-translational degree of freedom modes are calculated.

The user can create the superelement representation of the reduced model by specifying the SE_MASS, SE_STIFF, SE_INERT and SE_FILENAME fields. This implementation does not include SE_DAMP. The file, by default is written in the Nastran DMIG file format and can be used as input to *ELEMENT_DIRECT_MATRIX_INPUT. The keyword option _BINARY can be used to create a binary representation for the superelement which can be used with *ELEMENT_DIRECT_MATRIX_INPUT_BINARY to reduce the file size.

Static Condensation is equivalent to using only constraint modes with *CONTROL_IMPLICIT_MODES.

Static Condensation does have the ability to continue the analysis using the linear representation for a part set.

*CONTROL_IMPLICIT_TERMINATION

Purpose: Specify termination criteria for implicit transient simulations.

Card	1	2	3	4	5	6	7	8
Variable	DELTAU	DELTA1	KETOL	IETOL	TETOL	NSTEP		
Type	F	F	F	F	F	I		
Default	0.0	0.0	0.0	0.0	0.0	3		

VARIABLE	DESCRIPTION
DELTAU	Terminate based on relative total displacement in the Euclidean norm. GT.0.0: terminate when displacement in the Euclidean norm for last time step relative to the total displacement in the Euclidean norm is less than DELTAU.
DELTA1	Terminate based on relative total displacement in the max norm. GT.0.0: terminate when displacement in the max norm for last time step relative to the total displacement in the max norm is less than DELTAU.
KETOL	Terminate based on kinetic energy GT.0.0: terminate when kinetic energy drops below KETOL for NSTEP consecutive implicit time steps.
IETOL	Terminate based on internal energy GT.0.0: terminate when internal energy drops below IETOL for NSTEP consecutive implicit time steps.
TETOL	Terminate based on total energy GT.0.0: terminate when total energy drops below TETOL for NSTEP consecutive implicit time steps.
NSTEP	Number of steps used in the early termination tests for kinetic, internal, and total energy.

Remarks:

For some implicit applications it is useful to terminate when there is no change in displacement or low energy. This keyword provides the ability to specify such a stopping criterias to terminate the simulation prior to ENDTIM.

*CONTROL_MPP_DECOMPOSITION_AUTOMATIC

Purpose: Instructs the program to apply a simple heuristic to try to determine the proper decomposition for the simulation.

There are no input parameters. The existence of this keyword triggers the automated decomposition. This option should not be used if there is more than one occurrence of any of the following options in the model:

*INITIAL_VELOCITY
*CHANGE_VELOCITY
*BOUNDARY_PRESCRIBED_MOTION

And the following control card must not be used:

*CONTROL_MPP_DECOMPOSITION_TRANSFORMATION

For the general case, it is recommended that you specify the proper decomposition using the command *CONTROL_MPP_DECOMPOSITION_TRANSFORMATION instead.

*CONTROL_MPP_DECOMPOSITION_BAGREF

Purpose: Using airbag reference geometry for decomposition instead the folded geometry

There are no input parameters. Using initial folded geometry for decomposition may cause load unbalance while the bag fully deployed. This option will allow to decompose the model in reference geometry to get better MPP load distribution while the bag is fully opened.

Remarks:

Command in partition file(pfile): BAGREF

*CONTROL_MPP_DECOMPOSITION_CHECK_SPEED

Purpose: Modifies the decomposition depending on the relative speed of the processors involved.

There are no input parameters. The existence of this keyword causes a short floating point timing routine to be executed on each processor. The information gathered is used during the decomposition, with faster processors being given a relatively larger portion of the problem. This option is not recommended on homogeneous systems.

*CONTROL_MPP_DECOMPOSITION_CONTACT_DISTRIBUTE

Purpose: Ensures that the indicated contact interfaces are distributed across all processors, which can lead to better load balance for large contact interfaces.

Card	1	2	3	4	5	6	7	8
Variable	ID1	ID2	ID3	ID4	ID5			
Type	I	I	I	I	I			
Default	None	None	None	None	None			

VARIABLE	DESCRIPTION
ID1	First contact interface ID to distribute. If no contact ID's are specified, the number given here corresponds to the order of the interfaces as they appear in the input, with the first being 1.
ID2, ID3, ID4, ID5	Remaining interfaces ID's to distribute.

Remarks:

Up to 5 contact interface ID's can be specified. The decomposition is modified as follows: First, all the elements involved in the first contact interface are decomposed across all the processors. Then all the elements involved in the second contact interface (excluding any already assigned to processors) are distributed, and so on. After all the contact interfaces given are processed, the rest of the input is decomposed in the normal manner. This will result in each processor having possibly several disjoint portions of the input assigned to it, which will increase communications somewhat. However, this can be offset by improved load balance in the contact. It is generally recommended that at most one or two interfaces be specified, and then only if they are of substantial size relative to the whole problem.

*CONTROL_MPP_DECOMPOSITION_CONTACT_ISOLATE

Purpose: Ensures that the indicated contact interfaces are isolated on a single processor, which can lead to decreased communication.

Card	1	2	3	4	5	6	7	8
Variable	ID1	ID2	ID3	ID4	ID5			
Type	I	I	I	I	I			
Default	None	None	None	None	None			

VARIABLE	DESCRIPTION
ID1	First contact interface ID to isolate. If no contact ID's are specified, the number given here corresponds to the order of the interfaces as they appear in the input, with the first being 1.
ID2, ID3, ID4, ID5	Remaining interfaces ID's to isolate.

Remarks:

Up to 5 contact interfaces can be specified. The decomposition is modified as follows: First, all the elements involved in the first contact interface ID are assigned to the first processor. Then all the elements involved in the second contact interface ID (excluding any already assigned to processors) are assigned to the next processor, and so on. After all the contact interfaces given are processed, the rest of the input is decomposed in the normal manner. This will result in each of the interfaces being processed on a single processor. For small contact interfaces this can result in better parallelism and decreased communication.

*CONTROL_MPP_DECOMPOSITION_DISTRIBUTE_ALE_ELEMENTS

Purpose: Ensures ALE elements are evenly distributed to all processors

There are no input parameters. ALE elements usually have higher computational cost than other type of elements and it is better to distribute them to all CPU for better load balance. The existence of this keyword causes DYNA/MPP to extract ALE parts from input and then evenly distributed to all processors.

Remarks:

Command in partition file(pfile): ALEDIST.

*CONTROL_MPP_DECOMPOSITION_DISTRIBUTE_SPH_ELEMENTS

Purpose: Ensures SPH elements are evenly distributed to all processors

There are no input parameters. SPH elements usually have higher computational cost than other type of elements and it is better to distribute them to all CPU for better load balance. The existence of this keyword causes DYNA/MPP to extract SPH parts from input and then evenly distributed to all processors.

Remarks:

Command in partition file (pfile): SPHDIST

*CONTROL_MPP_DECOMPOSITION_ELCOST

Purpose: Instructs the program to use a hardware specific element cost weighting for the decomposition

Card	1	2	3	4	5	6	7	8
Variable	ITYPE							
Type	I							
Default	None							

VARIABLE	DESCRIPTION
ITYPE	Hardware specific cost profile. EQ.1: Fujitsu PrimePower EQ.2: Intel IA 64, AMD Opteron EQ.3: Intel Xeon 64 EQ.4: General profile

Remarks:

Command in partition file(pfile): elcost itype.

*CONTROL_MPP_DECOMPOSITION_FILE

Purpose: Allow for pre-decomposition and a subsequent run or runs without having to do the decomposition.

Card	1	2	3	4	5	6	7	8
Variable	Name							
Type	A80							
Default	None							

VARIABLE	DESCRIPTION
NAME	Name of a file containing (or to contain) a decomposition record.

Remarks:

If the indicated file does not exist, it is created with a copy of the decomposition information from this run. If the file exists, it is read and the decomposition steps can be skipped. The original run that created the file must be for a number of processors that is a multiple of the number of processors currently being used. Thus, a problem can be decomposed once for, say, 48 processors. Subsequent runs are then possible on any number that divides 48: 1, 2, 3, 4, 6, etc. Since the decomposition phase generally requires more memory than execution, this allows large models to be decomposed on one system and run on another (provided the systems have compatible binary formats). The file extension ".pre" is added automatically.

*CONTROL_MPP_DECOMPOSITION_METHOD

Purpose: Specify the decomposition method to use.

Card	1	2	3	4	5	6	7	8
Variable	Name							
Type	A80							
Default	RCB							

VARIABLE	DESCRIPTION
NAME	Name of the decomposition method to use. There are currently two options: RCB = recursive coordinate bisection GREEDY = a simple heuristic method In almost all cases the RCB method is superior and should be used.

*CONTROL_MPP_DECOMPOSITION_NUMPROC

Purpose: Specify the number of processors for decomposition.

Card	1	2	3	4	5	6	7	8
Variable	N							
Type	I							
Default	None							

VARIABLE	DESCRIPTION
N	Number of processors for decomposition.

Remarks:

This is used in conjunction with the CONTROL_MPP_DECOMPOSITION_FILE command to allow for later runs on different numbers of processors. By default, the decomposition is performed for the number of processors currently being used. However, a different value can be specified here. If N > 1 and only one processor is currently being used, the decomposition is done and then the program terminates. Similarly, if N is NOT a multiple of the current number of processors, the execution terminates after decomposition. Otherwise, the decomposition is performed for N processors, and the execution continues.

*CONTROL_MPP_DECOMPOSITION_OUTDECOMP

Purpose: Instructs the program to output element's ownership data to file for post-processor to show state data from different processors

Card	1	2	3	4	5	6	7	8
Variable	TYPE							
Type	I							
Default	None							

VARIABLE	DESCRIPTION
ITYPE	1: database in ls-prepost format to file decomp_parts.lsprepost.
	2: database in animator format to file decomp_parts.ses

Remarks:

Command in partition file(pfile): OUTDECOMP ITYPE.

*CONTROL_MPP_DECOMPOSITION_PARTS_DISTRIBUTE

Purpose: Distribute the parts given in this option to all processors before perform the decomposition for the full model. Only the first occurrence of this option will be activated and the rest of them will be ignored. The option in pfile has higher priority than in keyword.

Card	1	2	3	4	5	6	7	8
Variable	ID1	ID2	ID3	ID4	ID5	ID6	ID7	ID8
Type	I	I	I	I	I	I	I	I
Default	None	None	None	None	None	None	None	None

VARIABLE	DESCRIPTION
ID1, ID2, ID3, ...	Part ID to be distributed. If ID1,ID2,.. < 0, abs(ID1) will be used as part set ID. All parts defined in this card will be treated as a single region to be decomposed.

Remarks:

Up to 16 parts/part sets can be specified. The decomposition is modified as follows: First, all the elements involved in above parts will be treated as separate domain from the model and are distributed to all processors evenly based on their computational cost. Then all the rest of elements will be distributed as second domain.

Command in partition file(pfile): PARTLIST ID1,ID2,ID3,....

Part set is not supported under pfile option

*CONTROL_MPP_DECOMPOSITION_SCALE_CONTACT_COST

Purpose: Instructs the program to apply a scale factor to the list of contacts to change the partition weight for the decomposition.

Card	1	2	3	4	5	6	7	8
Variable	SF	ID1	ID2	ID3	ID4	ID5	ID6	ID7
Type	F	I	I	I	I	I	I	I
Default	None	None	None	None	None	None	None	None

VARIABLE	DESCRIPTION
SF	Scale factor for the contact segments listed in the interface ID.
ID1, ID2,...	interfaces ID's to be considered for scaling. Include second card if necessary.

Remarks:

Up to 15 contact interfaces ID can be specified. The decomposition is modified by applying this scale factor to the default computational cost of elements for the given contact interface ID.

Command in partition file(pfile): CTCOST ID1,ID2,...,SF

*CONTROL_MPP_DECOMPOSITION_SCALE_FACTOR_SPH

Purpose: Instructs the program to apply a scale factor to SPH elements to change the partition weight for the decomposition.

Card	1	2	3	4	5	6	7	8
Variable	SF							
Type	F							
Default	None							

VARIABLE	DESCRIPTION
ITYPE	1: database in ls-prepost format to file decomp_parts.lsprepost.
	2: database in animator format to file decomp_parts.ses

Remarks:

Command in partition file(pfile): SPHSF SF.

*CONTROL_MPP_DECOMPOSITION_SHOW

Purpose: Allows display of the final decomposition. There are no input parameters. The existence of this keyword causes the d3plot file to be modified so that all elements belonging to the first processor have material type 1, those on the second processor type 2, and so on. Execution terminates immediately after the decomposition phase, and no simulation is performed. This can be used in conjunction with the CONTROL_MPP_DECOMPOSITION_NUMPROC command to run on 1 processor and produce a d3plot file to visualize the resulting decomposition.

*CONTROL_MPP_DECOMPOSITION_TRANSFORMATION

Purpose: Specifies transformations to apply to modify the decomposition.

There are 10 different transformations that can be applied. The input is described here. For a detailed description of each decomposition transformation, see the description in the Appendix for the "pfile".

Any number of transformations can appear with no need for further *CONTROL cards – all non-comment cards up the next control card are expected to be decomposition transformations.

The first 6 transformations each take one parameter:

Card	1	2	3	4	5	6	7	8
Variable	TYPE	VAL						
Type	A10	F						
Default	None	0.0						

VARIABLE	DESCRIPTION
TYPE	Which transformation to apply. The possible values are: RX, RY, RZ, SX, SY, SZ
VAL	The amount of scaling/rotation to apply.

The remaining 4 transformations each take 9 parameters:

Card 1	1	2	3	4	5	6	7	8
Variable	TYPE	V1	V2	V3	V4	V5	V6	
Type	A10	F	F	F	F	F	F	
Default	None	0.0	0.0	0.0	0.0	0.0	0.0	

Card 2

Variable	V7	V8	V9					
Type	F	F	F					
Default	0.0	0.0	0.0					

VARIABLE	DESCRIPTION
TYPE	Which transformation to apply. The possible values are: VEC3, C2R, S2R, MAT
V1-V9	Parameters to the transformation.

***CONTROL_MPP_IO_BINOUTONLY**

Purpose: Use binout as the default format for ASCII files. There are no input parameters. The existence of this keyword will ignore the ASCII output format in the input file and use binout as default.

Remarks:

Command in partition file(pfile): BINOUTONLY

*CONTROL_MPP_IO_LSTC_REDUCE

Purpose: Use LSTC' own reduce routine to get consistent summation of floating point data among processors. The option is only working with single precision (I4R4) releases. There are no input parameters.

Remarks:

Command in partition file (pfile): lstc_reduce

***CONTROL_MPP_IO_NOD3DUMP**

Purpose: Suppresses the output of all dump files.

There are no input parameters. The existence of this keyword causes the d3dump and runrsf file output routines to be skipped.

*CONTROL_MPP_IO_NODUMP

Purpose: Suppresses the output of all dump files and full deck restart files.

There are no input parameters. The existence of this keyword causes the d3dump and runrsf file output routines to be skipped. It also suppresses output of the full deck restart file d3full.

***CONTROL_MPP_IO_NOFULL**

Purpose: Suppresses the output of the full deck restart files.

There are no input parameters. The existence of this keyword suppresses the output of the full deck restart file “d3full”.

*CONTROL_MPP_IO_SWAPBYTES

Purpose: Swap bytes on some of the output files.

There are no input parameters. The existence of this keyword causes the d3plot file and the “interface component analysis” file to be output with bytes swapped. This is to allow further processing of data on a different machine that has big endian vs. little endian incompatibilities compared to the system on which the analysis is running.

*CONTROL_NONLOCAL

Purpose: Allocate additional memory for *MAT_NONLOCAL option.

Card	1	2	3	4	5	6	7	8
Variable	MEM							
Type	I							
Default	none							

VARIABLE	DESCRIPTION
MEM	Percentage increase of memory allocated for MAT_NONLOCAL option over that required initially. This is for additional storage that may be required due to geometry changes as the calculation proceeds. Generally, a value of 10 should be sufficient.

*CONTROL_OUTPUT

Purpose: Set miscellaneous output parameters. This keyword does not control the information, such as the stress and strain tensors, which is written into the binary databases. For the latter, see the keyword *DATABASE_EXTENT_BINARY.

Card	1	2	3	4	5	6	7	8
Variable	NPOPT	NEECHO	NREFUP	IACCOP	OPIFS	IPNINT	IKEDIT	IFLUSH
Type	I	I	I	I	F	I	I	I
Default	0	0	0	0	0.	0	100	5000

Optional

Card	1	2	3	4	5	6	7	8
Variable	IPRTF	IERODE	TET10	MSGMAX	IPCURV	GMDT		
Type	I	I	I	I	I	F		
Default	0	0	2	50	0	0.		

VARIABLE	DESCRIPTION
NPOPT	Print suppression during input phase flag for the printed output file: EQ.0: no suppression, EQ.1: nodal coordinates, element connectivities, rigid wall definitions and initial velocities are not printed.
NEECHO	Print suppression during input phase flag for echo file: EQ.0: all data printed, EQ.1: nodal printing is suppressed, EQ.2: element printing is suppressed, EQ.3: both node and element printing is suppressed.

VARIABLE	DESCRIPTION
NREFUP	Flag to update reference node coordinates for beam elements. **This option requires that each reference node is unique to the beam:** EQ.0: no update, EQ.1: update.
IACCOP	Flag to average or filter nodal accelerations output to file "nodout" and the time history database "d3thdt": EQ.0: no average (default), EQ.1: averaged between output intervals, EQ.2: accelerations for each time step are stored internally and then filtered over each output interval using a filter from General Motors [Sala, Neal, and Wang, 2004] based on a low-pass Butterworth frequency filter. See also [Neal, Lin, and Wang, 2004]. DT2MS in *CONTROL_TIMESTEP must be set to a negative value when IACCOP=2 so that the maximum possible number of time steps for an output interval is known and adequate memory can be allocated. See Figure 8.9.
OPIFS	Output interval for interface file (Δt), see INTRODUCTION, Execution syntax.
IPNINT	Print initial time step sizes for all elements on the first cycle: EQ.0: 100 elements with the smallest time step sizes are printed. EQ.1: the governing time step sizes for each element are printed.
IKEDIT	Problem status report interval steps to the D3HSP (printed output) file. This flag is ignored if the GLSTAT file is written, see *DATABASE_GLSTAT.
IFLUSH	Number of time steps interval for flushing I/O buffers. The default value is 5000. If the I/O buffers are not emptied and an abnormal termination occurs, the output files can be incomplete. The I/O buffers for restart files are emptied automatically whenever a restart file is written so these files are not affected by this option.
IPRTF	Default print flag for RBDOUT and MATSUM files. This flag defines the default value for the print flag which can be defined in the part definition section, see *PART. This option is meant to reduce the file sizes by eliminating data which is not of interest. EQ.0: write part data into both MATSUM and RBDOUT EQ.1: write data into RBDOUT file only EQ.2: write data into MATSUM file only EQ.3: do not write data into RBDOUT and MATSUM

VARIABLE	DESCRIPTION
IERODE	Output eroded internal and kinetic energy into the MATSUM file. Also, output to the MATSUM file under the heading of part ID 0 is the kinetic energy from nonstructural mass, lumped mass elements and lumped inertia elements. EQ.0: do not output extra data EQ.1: output the eroded internal and kinetic energy
TET10	Output ten connectivity nodes into D3PLOT database. The current default is set to 2 since this change in the database may make the data unreadable for many popular post-processors and older versions of Ls-prepost. The default will change to 1 later. EQ.1: write the full ten node connectivity into the D3PLOT database EQ.2: write the four corner nodes of the ten node connectivity into the D3PLOT database
MSGMAX	Maximum number of each error/warning message
IPCURV	Flag to output digitized curve data to d3msg and d3hsp files. EQ.0: off EQ.1: on
GMDT	Output interval for recorded motions from *INTERFACE_SSI_AUX

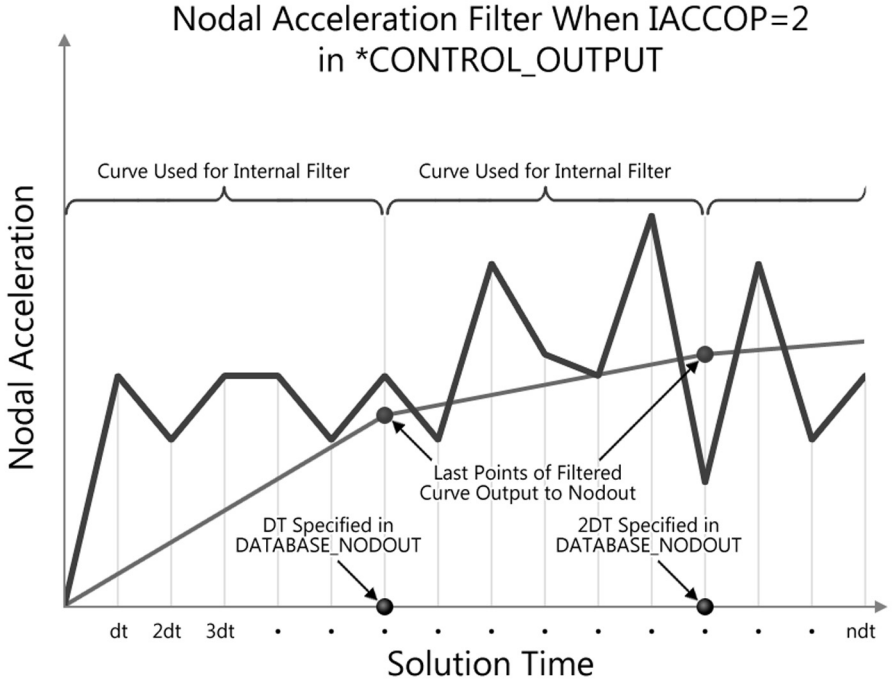


Figure 8.9. Nodal Acceleration Filter for IACOOP=2.

*CONTROL_PARALLEL

Purpose: Control parallel processing usage for shared memory computers by defining the number of processors and invoking the optional consistency of the global vector assembly.

Card	1	2	3	4	5	6	7	8
Variable	NCPU	NUMRHS	CONST	PARA				
Type	I	I	I	I				
Default	1	0	2	0				
Remarks		1	2	3				

VARIABLE	DESCRIPTION
NCPU	Number of cpus used. (If runs are submitted through a batch queue or a job scheduler see the Execution Syntax section of Getting Started and the *KEYWORD command for alternative ways to set the number of CPUS.)
NUMRHS	Number of right-hand sides allocated in memory: EQ.0: same as NCPU, always recommended, EQ.1: allocate only one.
CONST	Consistency flag for parallel solution (NCPU >1). Option 2 is recommended for metal forming applications. EQ.1: on EQ.2: off, for a faster solution (default).
PARA	Flag for parallel force assembly if CONST=1. EQ.0: off EQ.1: on

Remarks:

1. It is recommended to always set NUMRHS=NCPU since great improvements in the parallel performance are obtained since the force assembly is then done in parallel. Setting NUMRHS to one reduces storage by one right hand side vector for each additional processor after the first. If the consistency flag is active, i.e., CONTST=1, NUMRHS defaults to unity.

2. For any given problem with the consistency option off, i.e., CONST=2, slight differences in results are seen when running the same job multiple times with the same number of processors and also when varying the number of processors. Comparisons of nodal accelerations often show wide discrepancies; however, it is worth noting that the results of accelerometers often show insignificant variations due to the smoothing effect of the accelerometers which are generally attached to nodal rigid bodies. The accuracy issues are not new and are inherent in numerical simulations of automotive crash and impact problems where structural bifurcations under compressive loads are common. This problem can be easily demonstrated by using a perfectly square thin-walled tubular beam of uniform cross section under a compressive load. Typically, every run on one processor that includes a minor input change (i.e., element or hourglass formulation) will produces dramatically different results in terms of the final shape, and, likewise, if the same problem is again run on a different brand of computer. If the same problem is run on multiple processors the results can vary dramatically from run to run WITH NO INPUT CHANGE. The problem here is due to the randomness of numerical round-off which acts as a trigger in a "perfect" beam. Since summations with (CONST=2) occur in a different order from run to run, the round-off is also random. The consistency flag, CONST=1, provides for identical results (or nearly so) whether one, two, or more processors are used while running in the shared memory parallel (SMP) mode. This is done by requiring that all contributions to global vectors be summed in a precise order independently of the number of processors used. When checking for consistent results, nodal displacements or element stresses should be compared. The NODOUT and ELOUT files should be digit to digit identical. However, the GLSTAT, SECFORC, and many of the other ASCII files will not be identical since the quantities in these files are summed in parallel for efficiency reasons and the ordering of summation operations are not enforced. The biggest drawback of this option is the CPU cost penalty which is at least 15 percent if PARA=0 and is much less if PARA=1 and 2 or more processors are used. Unless the PARA flag is on (for non-vector processors), parallel scaling is adversely affected. The consistency flag does not apply to MPP parallel.

3. The PARA flag will cause the force assembly for the consistency option to be performed in parallel for the shared memory parallel option. Better scaling will be obtained with the consistency option, but with more memory usage. However, the single processing speed is slightly diminished. The logic for parallelization cannot be efficiently vectorized and is not recommended for vector computers since is will degrade CPU performance. This option does not apply to MPP parallel. If PARA=CONST=0 and NUMRHS=NCPU the force assembly by default is done in parallel.

***CONTROL_PORE_FLUID**

Purpose: Set parameters for pore water pressure calculations.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	ATYPE	(blank)	WTABLE	PF_RHO	GRAV	PF_BULK	OUTPUT	TMF
Type	I	F	F	F	F	F	I	F
Default	0	0.0	0.0	(none)	(none)	(none)	0	1.0

Card 2	1	2	3	4	5	6	7	8
Variable	TARG	FMIN	FMAX	(blank)	CONV	CONMAX	ETERM	
Type	F	F	F	F	F	F	F	
Default	0.0	0.0	0.0	1.0	1.E-4	1.E20	0.0	

VARIABLE	DESCRIPTION
ATYPE	Analysis type for pore water pressure calculations: EQ.0: No pore water pressure calculation. EQ.1: Undrained analysis, EQ.2: Drained analysis, EQ.3: Time dependent consolidation (coupled) EQ.4: Consolidate to steady state (uncoupled) EQ.5: Drained in dynamic relaxation, undrained in transient. EQ.6: As 4 but do not check convergence, continue to end time
WTABLE	Default z-coordinate of water table (where pore pressure is zero).
PF_RHO	Default density for pore water.
GRAV	Gravitational acceleration used to calculate hydrostatic pore water pressure.

PF_BULK	Default bulk modulus of pore fluid (stress units).
OUTPUT	Output flag controlling stresses to D3PLOT and D3THDT binary files: EQ.0: total stresses are output EQ.1: effective stresses are output, see notes
TMF	Initial Time Magnification factor on seepage (ATYPE=3,4 only). GT.0: Factor (can be used with automatic control, see TARG, FMIN, FMAX). LT.0: Load Curve ID (see *DEFINE_CURVE) giving Time Magnification Factor versus analysis time.
TARG	Target for maximum change of excess pore pressure at any node, per timestep. If the actual change falls below the target, the time factor on the seepage calculation will be increased (see notes). If zero, the constant value of TMF is used. If non-zero, TMF is taken as the initial factor.
FMIN	Minimum time factor on seepage calculation
FMAX	Maximum time factor on seepage calculation
CONV	Convergence tolerance for ATYPE=4 – maximum head change per timestep at any node (length units)
CONMAX	Maximum factor on permeability with ATYPE=-4
ETERM	Event time termination (ATYPE=3)

Notes:

General

LS-DYNA uses Terzaghi's concept of Effective Stress to simulate materials with pore pressure. The pore fluid and soil skeleton are assumed to occupy the same volume and to carry loads in parallel. Thus the total stress in an element is the sum of the "effective stress" in the soil skeleton, plus the hydrostatic stress in the pore fluid. The "effective stress" is determined by the LS-DYNA material model in the normal way – the pore fluid treatment is independent of material model. The pore pressure is calculated at nodes, and interpolated onto the elements. The pore fluid hydrostatic stress is then the negative of the element pore pressure.

The description here relates to analysis of soils, which is the main intended application, but other materials containing pore fluid could be treated by the same methods.

The pore pressure capabilities are available in SMP and MPP versions of LS-DYNA (explicit solution method) but are not available for implicit solutions.

Pressure head

Pressure head (length units) = pressure/ρg where ρ=pore fluid density (PF_RHO) and g is the acceleration due to gravity (GRAV).

Analysis Types

"Undrained": the pore fluid is trapped within the material. Volume changes result in pore pressure changes. This approximation is used to simulate the effect of rapidly-applied loads on relatively impermeable soil.

"Drained": the pore fluid is free to move within the material such that the user-defined pressure-versus-z-coordinate relationship is always maintained. This approximation is used to model high-permeability soils.

"Time-dependent consolidation": pressure gradients cause pore fluid to flow through the material according to Darcy's law:

$\mathbf{v} = \kappa.\text{grad}(p+z)$

where **v** = fluid velocity vector, κ=permeability, p=pressure head, z=z-coordinate.

Nett inflow or outflow at a node leads to a theoretical volume gain or loss. The analysis is coupled, i.e. any difference between actual and theoretical volume leads to pore pressure change, which in turn affects the fluid flow. The result is a prediction of response-versus-time.

"Steady-state consolidation": iterative calculation to achieve steady-state pore pressure. The analysis is uncoupled, i.e. only the final state is meaningful, not the response-versus-time.

Time factoring

Consolidation occurs over time intervals of days, weeks or months. To simulate this process using explicit time integration, a speed-up factor is used. The permability of the soil is increased by this "time factor" so that consolidation occurs more quickly. The output times in the D3PLOT and D3THDT files are modified to reflect the magnification used. The modified time (Event time) will be different to the analysis time. Loading and other load curve data use the analysis time to calculate loading and other values. It is recommended to use automatic factoring (e.g. TMF = 1.0, TARG = 1m head, FMIN = 1.0, FMAX = 1.0e6)

See also:

*BOUNDARY_PORE_FLUID (essential - without this card, no parts will have pore fluid)

*BOUNDARY_PWP_option

*DATABASE_PWP_OUTPUT

*DATABASE_PWP_FLOW

*LOAD_ADDED_PWP

*MAT_ADD_PERMEABILITY

Output: Extra variables for solid elements are automatically written to the d3plot and d3thdt files when the model contains *CONTROL_PORE_FLUID. At present, 15 additional extra variables are written. This number will be reduced in future versions of LS-DYNA. Of these, only the Second ("Excess pore pressure head") and seventh (Pore pressure in stress units) are generally used. These 15 follow any extra variables requested by the user, e.g. if the user requested 3 extra variables, then the pore pressure outputs will be extra variables 4 through 18 and the pore pressure will be the 10th.

Further optional output to d3plot and d3thdt files is available – see *DATABASE_PWP_OUTPUT.

For time-dependent and steady-state consolidation, information on the progress of the analysis is written to d3hsp file.

*CONTROL_REMESHING

Purpose: Provide control over the remeshing of solids which are meshed with the solid tetrahedron element type 13. The element size for three-dimensional adaptivity can be set on the surface mesh of the solid part, and adaptivity can be activated based on the criterions of volume loss, mass increase, or minimum time step size.

Card	1	2	3	4	5	6	7	8
Variable	RMIN	RMAX	VF_LOSS	MFRAC	DT_MIN			
Type	F	F	F	F	F			
Default	none	none	1.0	0.0	0.			

VARIABLE	DESCRIPTION
RMIN	Minimum edge length for the surface mesh surrounding the parts which should be remeshed.
RMAX	Maximum edge length for the surface mesh surrounding the parts which should be remeshed.
VF_LOSS	Volume fraction loss required in a type 13 solid elements to trigger a remesh. In the type 13 solid elements, the pressures are computed at the nodal points; therefore, it is possible for volume to be conserved but for individual tetrahedrons to experience a significant volume loss or gain. The volume loss can lead to numerical problems. Recommended values for VF_LOSS in the range of 0.10 to 0.30 may be reasonable.
MFRAC	Mass ratio gain during mass scaling required for triggering a remesh. For a one percent increase in mass, set MFAC=0.010.
DT_MIN	Time step size required for triggering a remesh. This option is checked before mass scaling is applied and the time step size reset.

Remarks:

1. The value of RMIN and RMAX should be of the same order. The value of RMAX can be set to 2-5 times greater than RMIN.

*CONTROL_RIGID

Purpose: Special control options related to rigid bodies and the rigid-flexible bodies, see *PART_MODES.

Card	1	2	3	4	5	6	7	8
Variable	LMF	JNTF	ORTHMD	PARTM	SPARSE	METALF		
Type	I	I	I	I	I	I		
Default	0	0	0	0	0	0		

VARIABLE **DESCRIPTION**

LMF — Switch the explicit rigid body joint treatment to an implicit formulation which uses Lagrange multipliers to impose prescribed kinematic boundary conditions and joint constraints. This is a new option which is under development in version 970. There is a slight cost overhead due to the assembly of sparse matrix equations which are solved using standard procedures for nonlinear problems in rigid multi-body dynamics. Lagrange multiplier flag:
EQ.0: explicit penalty formulation,
EQ.1: implicit formulation with Lagrange multipliers.

JNTF — Generalized joint stiffness formulation; see remark 1 below:
EQ.0: incremental update,
EQ.1: total formulation (exact).

ORTHMD — Orthogonalize modes with respect to each other:
EQ.0: true.
EQ.1: false, the modes are already orthogonalized.

PARTM — Use global mass matrix to determine part mass distribution. This mass matrix may contain mass from other parts that share nodes. See remark 2 below.
EQ.0: true,
EQ.1: false.

SPARSE — Use sparse matrix multiply subroutines for the modal stiffness and damping matrices. See Remark 3.
EQ.0: false, do full matrix multiplies (frequently faster),
EQ.1: true.

VARIABLE	DESCRIPTION
MATELF	Metalforming option, which should not be used for crash and other applications involving rigid bodies. Use fast update of rigid body nodes. If this option is active the rotational motion of all rigid bodies should be suppressed. EQ.0: full treatment is used EQ.1: fast update for metalforming applications

Remarks:

1. As the default, the calculation of the relative angles between two coordinate systems is done incrementally. This is an approximation, in contrast to the total formulation where the angular offsets are computed exactly. The disadvantage of the latter approach is that a singularity exists when an offset angle equals 180 degrees. For most applications, the stop angles prevent this occurrence and JNTF=1 should not cause a problem.

2. If the determination of the normal modes included the mass from both connected bodies and discrete masses, or if there are no connected bodies, then the default is preferred. When the mass of a given part ID is computed, the resulting mass vector includes the mass of all rigid bodies that are merged to the given part ID, but does not included discrete masses. See the keyword: *CONSTRAINED_RIGID_BODIES. A lumped mass matrix is always assumed.

3. Sparse matrix multipliers save a substantial number of operations if the matrix is truly sparse. However, the overhead will slow the multipliers for densely populated matrices.

*CONTROL_SHELL

Purpose: Provide controls for computing shell response.

Card	1	2	3	4	5	6	7	8
Variable	WRPANG	ESORT	IRNXX	ISTUPD	THEORY	BWC	MITER	PROJ
Type	F	I	I	I	I	I	I	I
Default	20.	0	-1	0	2	2	1	0

First optional Card

Card	1	2	3	4	5	6	7	8
Variable	ROTASCL	INTGRD	LAMSHT	CSTYP6	TSHELL	NFAIL1	NFAIL4	PSNFAIL
Type	F	I	I	I	I	I	I	I
Default	1..	0	0	1	0	inactive	inactive	0

Second optional Card (The first card must also be defined)

Card	1	2	3	4	5	6	7	8
Variable	PSSTUPD	IRQUAD	CNTCO					
Type	I	I	I					
Default	1..	0	0					

VARIABLE	DESCRIPTION
WRPANG	Shell element warpage angle in degrees. If a warpage greater than this angle is found, a warning message is printed. Default is 20 degrees.

VARIABLE	DESCRIPTION
ESORT	Automatic sorting of triangular shell elements to treat degenerate quadrilateral shell elements as C0 or DKT triangular shells, see option THEORY below: EQ.0: no sorting required (default), EQ.1: full sorting (C0 triangular shells), EQ.2: full sorting (DKT triangular shells) 971 version R3 and later. The DKT triangular element will be unstable if used to model a thick shell.
IRNXX	Shell normal update option. This option affects the Hughes-Liu, Belytschko-Wong-Chiang, and the Belytschko-Tsay shell formulations. The latter is affected if and only if the warping stiffness option is active, i.e., BWC=1. EQ.-2: unique nodal fibers which are incrementally updated based on the nodal rotation at the location of the fiber, EQ.-1: recomputed fiber directions each cycle, EQ.0: default set to -1, EQ.1: compute on restarts, EQ.n: compute every n cycles (Hughes-Liu shells only).
ISTUPD	Shell thickness change option for deformable shells. The parameter, PSSTUPD, on the second optional card allows this option to be applied by part ID. For crash analysis, neglecting the elastic component of the strains, ISTUPD=4, may improve energy conservation and stability. EQ.0: no thickness change. EQ.1: membrane straining causes thickness change in 3 and 4 node shell elements. This option is very important in sheet metal forming or whenever membrane stretching is important. EQ.2: membrane straining causes thickness change in 8 node thick shell elements, types 1 and 2. This option is not recommended for implicit or explicit solutions which use the fully integrated type 2 element. The type 3 thick shell is a continuum based shell and thickness changes are always considered. EQ.3: options 1 and 2 apply. EQ.4: option 1 applies, but the elastic strains are neglected for the thickness update. This option only applies to the most common elastic-plastic materials for which the elastic response is isotropic.
THEORY	Default shell theory: EQ.1: Hughes-Liu, EQ.2: Belytschko-Tsay (default), EQ.3: BCIZ triangular shell (not recommended), EQ.4: C^0 triangular shell, EQ.5: Belytschko-Tsay membrane, EQ.6: S/R Hughes Liu, EQ.7: S/R co-rotational Hughes Liu, EQ.8: Belytschko-Leviathan shell,

VARIABLE	DESCRIPTION
	EQ.9: fully integrated Belytschko-Tsay membrane, EQ.10: Belytschko-Wong-Chiang, EQ.11: Fast (co-rotational) Hughes-Liu. EQ.12: Plane stress (x-y plane), EQ.13: Plane strain (x-y plane), EQ.14: Axisymmetric solid (y-axis of symmetry) - area weighted, EQ.15: Axisymmetric solid (y-axis of symmetry) – volume weighted EQ.16: Fully integrated shell element (very fast) EQ.17: Discrete Kirchhoff triangular shell (DKT) EQ.18: Discrete Kirchhoff linear shell either quadrilateral or Triangular with 6DOF per node, EQ.20: C^0 linear shell element with 6 DOF per node EQ.21: C^0 linear shell element with 5 DOF per node with the Pian-Sumihara membrane hybrid quadrilateral membrane. EQ.25: Belytschko-Tsay shell with thickness stretch. EQ.26: Fully integrated shell with thickness stretch. EQ.27: C^0 triangular shell with thickness stretch. For the 2D axisymmetric solid elements, high explosive applications work best with the area weighted approach and structural applications work best with the volume weighted approach. The volume weighted approach can lead to problems along the axis of symmetry under very large deformations. Often the symmetry condition is not obeyed, and the elements will kink along the axis. The volume weighted approach must be used if 2D shell elements are used in the mesh. Type 14 and 15 elements cannot be mixed in the same calculation.
BWC	Warping stiffness for Belytschko-Tsay shells: EQ.1: Belytschko-Wong-Chiang warping stiffness added. EQ.2: Belytschko-Tsay (default).
MITER	Plane stress plasticity option (applies to materials 3, 18, 19, and 24): EQ.1: iterative plasticity with 3 secant iterations (default), EQ.2: full iterative plasticity, EQ.3: radial return noniterative plasticity. May lead to false results and has to be used with great care.
PROJ	Projection method for the warping stiffness in the Belytschko-Tsay shell (the BWC option above) and the Belytschko-Wong-Chiang elements (see remarks below). This parameter applies to explicit calculations since the full projection method is always used if the solution is implicit and this input parameter is ignored. EQ.0: drill projection, EQ.1: full projection.

VARIABLE	DESCRIPTION
ROTASCL	Define a scale factor for the rotary shell mass. This option is not for general use. The rotary inertia for shells is automatically scaled to permit a larger time step size. A scale factor other than the default, i.e., unity, is not recommended.
INTGRD	Default shell through thickness numerical integration rule: EQ.0: Gauss integration. If 1-10 integration points are specified, the default rule is Gauss integration. EQ.1: Lobatto integration. If 3-10 integration points are specified, the default rule is Lobatto. For 2 point integration, the Lobatto rule is very inaccurate, so Gauss integration is used instead. Lobatto integration has an advantage in that the inner and outer integration points are on the shell surfaces.
LAMSHT	Laminated shell theory is available for all thin shell and thick shell materials except those that use the Green-Lagrange strain by setting LAMSHT=3, 4, or 5 and by using *PART_COMPOSITE or *INTEGRATION_SHELL to define the integration rule. Lamination theory is applied to correct for the assumption of a uniform constant shear strain through the thickness of the shell. Unless this correction is applied, the stiffness of the shell can be grossly incorrect if there are drastic differences in the elastic constants from ply to ply, especially for sandwich type shells. Generally, without this correction the results are too stiff. For the discrete Kirchhoff shell elements, which do not consider transverse shear, this option is ignored. For thin shells of material types, *MAT_COMPOSITE_DAMAGE, *MAT_ENHANCED_COMPOSITE_DAMAGE, and *MAT_GENERAL_VISCOELASTIC, laminated shell theory may also be done by stiffness correction by setting LAMSHT=1. EQ.0: do not update shear corrections, EQ.1: activate laminated shell theory, EQ.3: activate laminated thin shells, EQ.4: activate laminated shell theory for thick shells, EQ.5: activate laminated shell theory for thin and thick shells.
CSTYP6	Coordinate system for the type 6 shell element. The default system computes a unique local system at each in plane point. The uniform local system computes just one system used throughout the shell element. This involves fewer calculations and is therefore more efficient. The change of systems has a slight effect on results; therefore, the older, less efficient method is the default. EQ.1: variable local coordinate system (default), EQ.2: uniform local system.
TSHELL	Thermal shell option. Four node shells are treated internally as twelve node brick elements to allow heat conduction through the thickness of the shell.

NFAIL1	Flag to check for highly distorted under-integrated shell elements, print a message, and delete the element or terminate. Generally, this flag is not needed for one point elements that do not use the warping stiffness. A distorted element is one where a negative Jacobian exist within the domain of the shell, not just at integration points. The checks are made away from the CPU requirements for one point elements. If nonzero, NFAIL1 can be changed in a restart.

VARIABLE	DESCRIPTION
	EQ.1: print message and delete element. EQ.2: print message, write D3DUMP file, and terminate GT.2: print message and delete element. When NFAIL1 elements are deleted then write D3DUMP file and terminate. These NFAIL1 failed elements also include all shell elements that failed for other reasons than distortion. Before the D3DUMP file is written, NFAIL1 is doubled, so the run can immediately be continued if desired.
NFAIL4	Flag to check for highly distorted fully-integrated shell elements, print a message and delete the element or terminate. Generally, this flag is recommended. A distorted element is one where a negative Jacobian exist within the domain of the shell, not just at integration points. The checks are made away from the integration points to enable the bad elements to be deleted before an instability leading to an error termination occurs. If nonzero, NFAIL1 can be changed in a restart. EQ.1: print message and delete element. EQ.2: print message, write D3DUMP file, and terminate GT.2: print message and delete element. When NFAIL4 elements are deleted then write D3DUMP file and terminate. These NFAIL4 failed elements also include all shell elements that failed for other reasons than distortion. Before the D3DUMP file is written, NFAIL4 is doubled, so the run can immediately be continued if desired.
PSNFAIL	Optional shell part set ID specifying which part ID's are checked by the NFAIL1 and NFAIL4 options. If zero, all shell part ID's are included.
PSSTUPD	|PSSTUPD| is the optional shell part set ID specifying which part ID's have or do not have their thickness updated. The shell thickness update by default applies to all shell elements in the mesh. Generally, this part set ID is not needed. LT.0: these shell parts are excluded from the shell thickness update EQ.0: all deformable shells have their thickness updated GT.0: these shell parts are included in the shell thickness update
IRQUAD	In plane integration rule for the 8 node shell element: EQ.2: 2 x 2 Gauss quadrature, EQ.3: 3 x 3 Gauss quadrature.
CNTCO	Flag to account for shell reference surface offsets in the contact treatment EQ.0: offsets are ignored EQ.1: offsets are treated using shell thickness EQ.2: offsets are treated using the user defined contact thickness which may be different than the shell thickness used in the element formulations

Remarks:

1. The drill projection is used in the addition of warping stiffness to the Belytschko-Tsay and the Belytschko-Wong-Chiang shell elements. This projection generally works well and is very efficient, but to quote Belytschko and Leviathan:

> "The shortcoming of the drill projection is that even elements that are invariant to rigid body rotation will strain under rigid body rotation if the drill projection is applied. On one hand, the excessive flexibility rendered by the 1-point quadrature shell element is corrected by the drill projection, but on the other hand the element becomes too stiff due to loss of the rigid body rotation invariance under the same drill projection".

They later went on to add in the conclusions:

> "The projection of only the drill rotations is very efficient and hardly increases the computation time, so it is recommended for most cases. However, it should be noted that the drill projection can result in a loss of invariance to rigid body motion when the elements are highly warped. For moderately warped configurations the drill projection appears quite accurate".

In crashworthiness and impact analysis, elements that have little or no warpage in the reference configuration can become highly warped in the deformed configuration and may affect rigid body rotations if the drill projection is used, i.e., DO NOT USE THE DRILL PROJECTION. Of course it is difficult to define what is meant by "moderately warped". The full projection circumvents these problems but at a significant cost. The cost increase of the drill projection versus no projection as reported by Belytschko and Leviathan is 12 percent and by timings in LS-DYNA, 7 percent, but for the full projection they report a 110 percent increase and in LS-DYNA an increase closer to 50 percent is observed.

In Version 940.xx of LS-DYNA the drill projection was used exclusively, but in one problem the lack of invariance was observed and reported; consequently, the drill projection was replaced in the Belytschko-Leviathan shell with the full projection and the full projection is now optional for the warping stiffness in the Belytschko-Tsay and Belytschko-Wong-Chiang elements. Until this problem occurred, the drill projection seemed okay. In version 950.xx and later versions of LS-DYNA the Belytschko-Leviathan shell is somewhat slower than previously. In general in light of these problems, the drill projection cannot be recommended. For implicit problems, the full projection method is used in the development of the stiffness matrix.

***CONTROL_SOLID**

Purpose: Provide controls for solid element response.

Card	1	2	3	4	5	6	7	8
Variable	ESORT	FMATRX	NIPTETS	SWLOCL	PSFAIL			
Type	I	I	I	I	I			
Default	0	0	4	2	0			

This card is optional. Card Format (10I8)

Card	1	2	3	4	5	6	7	8	9	10
Variable	PM1	PM2	PM3	PM4	PM5	PM6	PM7	PM8	PM9	PM10
Type	I	I	I	I	I	I	I	I	I	I
Default	none	none	none	none	none	none	none	none	none	none

VARIABLE	DESCRIPTION
ESORT	Automatic sorting of tetrahedron and pentahedron elements to treat degenerate tetrahedron and pentahedron elements as tetrahedron (formulation 10) and pentahedron (formulation 15) solids, respectively. See *SECTION_SOLID. EQ.0: no sorting required (default). EQ.1: full sorting,
FMATRX	Default method used in the calculation of the deformation gradient matrix. EQ.1: Update incrementally in time. This is the default for explicit. EQ.2: Directly compute F. This is the default for implicit and implicit/explicit switching.
NIPTETS	Number of integration points used in the quadratic tetrahedron elements. Either 4 or 5 can be specified. This option applies to the types 4, 16, and 17 tetrahedron elements.

VARIABLE	DESCRIPTION
SWLOCL	Output option for stresses in solid elements used as spot welds with material *MAT_SPOTWELD. EQ.1: Global (default), EQ.2: Local
PSFAIL	Optional solid part set ID specifying which part ID's are checked for negative volumes prior to element processing. If zero, and if ERODE on *CONTROL_TIMESTEP is set to 1, all solid elements are checked each step. The ERODE flag is ignored whenever PSFAIL is defined.
PM1-PM10	Components of a permutation vector for nodes that define the 10-node tetrahedron. The nodal numbering of 10-node tetrahedron elements is somewhat arbitrary. The permutation vector allows other numbering schemes to be used. Unless defined, this permutation vector is not used. PM1-PM10 are unique numbers between 1 to 10 inclusive that reorders the input node ID's for a 10-node tetrahedron into the order used by LS-DYNA.

*CONTROL_SOLUTION

Purpose: To specify the analysis solution procedure if thermal only or coupled thermal analysis is performed. Other solutions parameters including the vector length and NaN (not a number) checking can be set.

Card	1	2	3	4	5	6	7	8
Variable	SOLN	NLQ	ISNAN	LCINT				
Type	I	I	I	I				
Default	0	0	0	100				

VARIABLE	DESCRIPTION
SOLN	Analysis solution procedure: EQ.0: Structural analysis only, EQ.1: Thermal analysis only, EQ.2: Coupled structural thermal analysis.
NLQ	Define the vector length used in solution. This value must not exceed the vector length of the system which varies based on the machine manufacturer. The default vector length is printed at termination in the MESSAG file.
ISNAN	Flag to check for a NaN in the force and moment arrays after the assembly of these arrays is completed. This option can be useful for debugging purposes. A cost overhead of approximately 2% is incurred when this option is active. EQ.0: No checking, EQ.1: Checking is active.
LCINT	Number of equally spaced intervals used in the load curve discretization.

*CONTROL_SPH

Purpose: Provide controls for computing SPH particles

Card	1	2	3	4	5	6	7	8
Variable	NCBS	BOXID	DT	IDIM	MEMORY	FORM	START	MAXV
Type	I	I	F	I	I	I	F	F
Default	1	0	1.e20	none	150	0	0.0	1.e15

Optional Card

Card	1	2	3	4	5	6	7	8
Variable	CONT	DERIV	INI					
Type	I	I	I					
Default	0	0	0					

VARIABLE	DESCRIPTION
NCBS	Number of cycles between particle sorting
BOXID	SPH approximations are computed inside a specified BOX. When a particle has gone outside the BOX, it is deactivated. This will save computational time by eliminating particles that no longer interact with the structure.
DT	Death time. Determines when the SPH calculations are stopped.
IDIM	Space dimension for SPH particles: EQ. 3: for 3D problems EQ. 2: for 2D plane strain problems EQ.-2: for 2D axisymmetric problems When a value is not specified LS-DYNA determines the space dimension automatically by checking the use of 3D, 2D or 2D axisymmetric elements.

VARIABLE	DESCRIPTION
MEMORY	Defines the initial number of neighbors per particle (see Remark 1 below).
FORM	Particle approximation theory: EQ.0: default formulation, EQ.1: renormalization approximation EQ.2: symmetric formulation, EQ.3: symmetric renormalized approximation EQ.4: tensor formulation, EQ.5: fluid particle approximation EQ.6: fluid particle with renormalization approximation, EQ. 7: Total Lagrangian formulation EQ. 8: Total Lagrangian formulation with renormalization
START	Start time for particle approximation. Particle approximations will be computed when time of the analysis has reached the value defined in START.
MAXV	Maximum value for velocity for the SPH particles. Particles with a velocity greater than MAXV are deactivated.
CONT	Defines the computation of the particle approximation between two different SPH parts: EQ.0: Particle approximation is defined (default) EQ.1: Particle approximation is not computed. Two different SPH materials will not interact with each others and penetration is allowed.
DERIV	Time integration type for the smoothing length: EQ.0: $\frac{d}{dt}\left(h\left(t\right)\right)=\frac{1}{d}h\left(t\right)div\left(v\right)$ (default), EQ.1: $\frac{d}{dt}\left(h\left(t\right)\right)=\frac{1}{d}h\left(t\right)\left(div\left(v\right)\right)^{1/3}$
INI	Computation of the smoothing length during the initialization: EQ.0: Bucket sort based algorithm (default, very fast) EQ.1: Global computation on all the particles of the model. EQ. 2: Based on the mass of the SPH element.

Remark:

1. This variable is for memory allocation of arrays during the initialization phase. It can be positive or negative. If this value is positive, memory allocation is dynamic. During the calculation, some particles can request more neighbors and LS-DYNA will automatically adapt the size of that variable. Default value should apply for most applications. If this value is negative, memory allocation is static. During the

calculation only the closest SPH elements will be considered as neighbors. Using this option can avoid memory allocation problems.

*CONTROL_SPOTWELD_BEAM

Purpose: Provides factors for scaling the failure force resultants of beam spot welds as a function of their parametric location on the contact segment and the size of the segment. Also, an option is provided to replace beam welds with solid hexahedron element clusters.

Card	1	2	3	4	5	6	7	8
Variable	LCT	LCS	T_ORT	PRTFLG	T_ORS	RPBHX	BMSID	ID_OFF
Type	I	I	I	I	I	I	I	I
Default	0	0	0	0	0	0	0	0

VARIABLE	DESCRIPTION
LCT	Load curve ID for scaling the response in tension based on the shell element size.
LCS	Load curve ID for scaling the response in shear based on the shell element size.
T_ORT	Table ID for scaling the tension response (and shear response if T_ORS=0) based on the location of the beam node relative to the centroid of the shell.
PRTFLG	Set this flag to 1 to print for each spot weld attachment: the beam, node, and shell ID's, the parametric coordinates that define the constraint location, the angle used in the table lookup, and the three scale factors obtained from the load curves and table lookup. See Figure 8.10.
T_ORS	Optional table ID for scaling the shear response based on the location of the beam node relative to the centroid of the shell.
RPBHX	Replace each spot weld beam element with a cluster of RPBHX solid elements. RPBHX may be set to 1, 4, or 8. When RPBHX is set to 4 or 8, a table is generated to output the force and moment resultants into the SWFORC file, if this file is active. This table is described by the keyword: *DEFINE_HEX_SPOTWELD_ASSEMBLY. The ID's of the beam elements are used as the cluster spot weld ID's so the ID's in the SWFORC file are unchanged. The beam elements are automatically deleted from the calculation, and the section and material data is automatically changed to be used with solid elements. See Figure 11.8.

VARIABLE	DESCRIPTION
BMSID	Optional beam set ID defining the beam element ID's that are to be converted to hex assemblies. If zero, all spot weld beam elements are converted to hex assemblies. See the keyword, *SET_BEAM_ GENERAL for an efficient way of defining beam sets.
ID_OFF	This optional ID offset applies if and only if BMSID is nonzero. Beams, which share part ID's with beams that are converted to hex assemblies, will be assigned new part ID's by adding to the original part ID the value of ID_OFF. If ID_OFF, is zero the new part ID for such beams will be assigned to be larger than the largest part ID in the model.

Remarks:

The load curves and table provide a means of scaling the response of the beam spot welds to reduce any mesh dependencies for failure model 6 in *MAT_SPOTWELD. Figure 8.11 shows such dependencies that can lead to premature spot weld failure. Separate scale factors are calculated for each of the beam's nodes. The scale factors s_T, s_S, s_{OT} , and s_{OS} are calculated using the load curves LCT, LCS, table T_ORT, and table T_ORS, respectively, and are introduced in the failure criteria,

$$\left(\frac{s_T s_{OT}\sigma_{rr}}{\sigma_{rr}^F\left(\dot{\varepsilon}_{eff}\right)}\right)^2+\left(\frac{s_S s_{OS}\tau}{\tau^F\left(\dot{\varepsilon}_{eff}\right)}\right)^2-1=0$$

If a curve or table is given an ID of 0, its scale factor is set to 1.0. The load curves LCT and LCS are functions of the characteristic size of the shell element used in the time step calculation at the start of the calculation. The orientation table is a function of the spot weld's isoparametric coordinate location on the shell element. A vector *V=(s,t)* is defined from the centroid of the shell to the contact point of the beam's node. The arguments for the orientation table are the angle:

$$\Theta=\tan^{-1}\left(\frac{\min(|s|,|t|)}{\max(|s|,|t|)}\right),$$

and the normalized distance $\bar{d}=d/D=\max\left(|s|,|t|\right)$. See Figure 8.10. The table is periodic over a range of 0 (*V* aligned with either the *s* or *t* axis) to 45 degrees (*V* is along the diagonal of the element). The table is specified by the angle of V in degrees, ranging from 0 to 45, and the individual curves give the scale factor as a function of the normalized distance of the beam node, $\bar{d}$, for a constant angle.

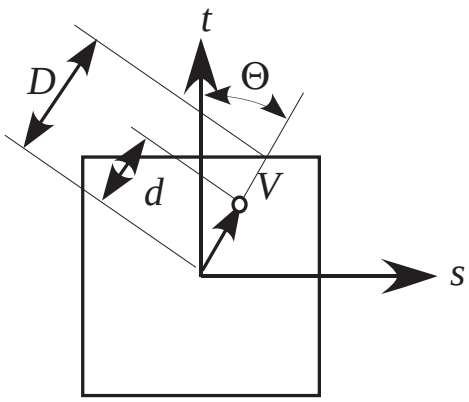


Figure 8.10. Definition of parameters for table definition.

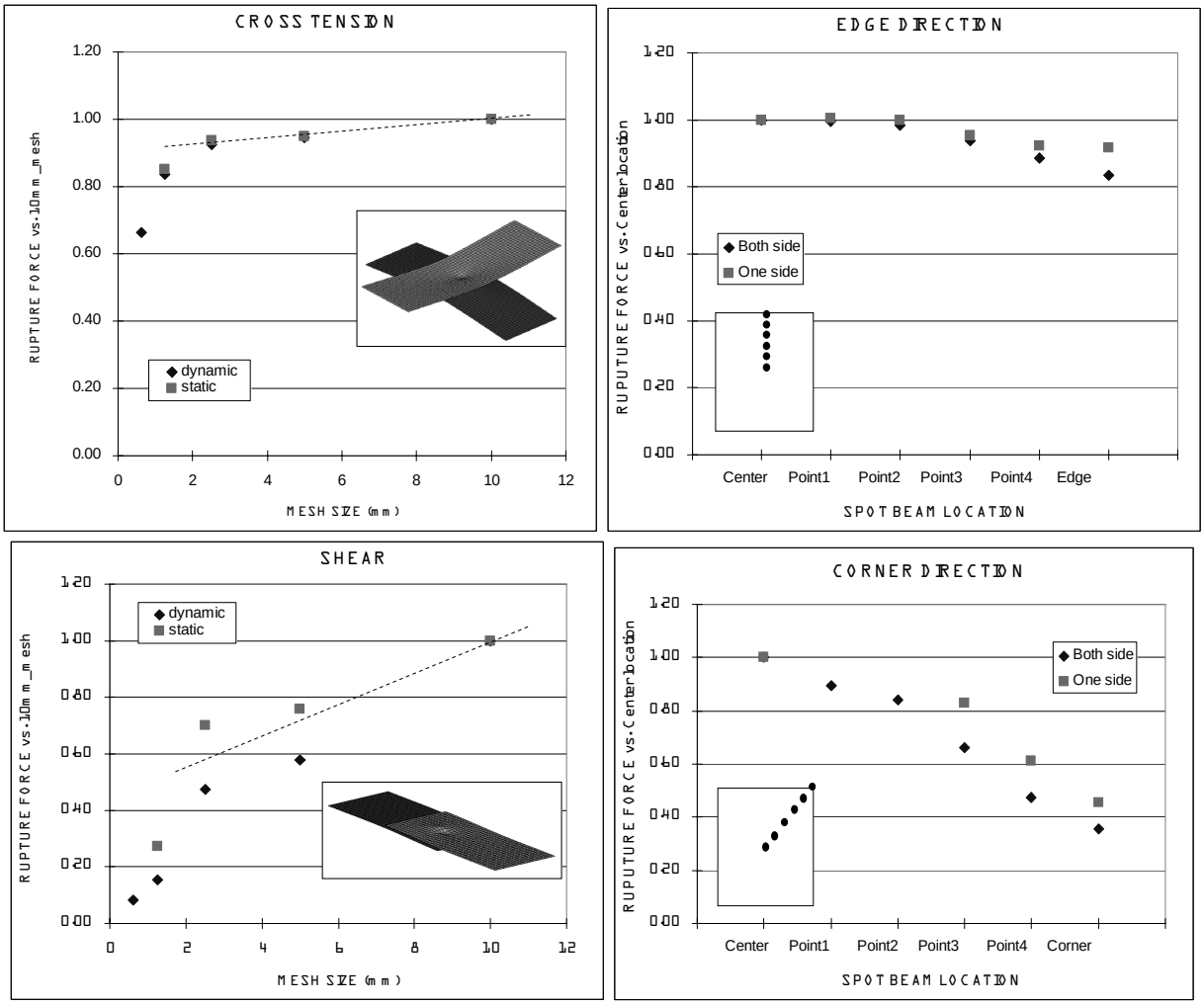


Figure 8.11. The failure force resultants can depend both on mesh size and the location of weld relative to the center of the contact segment.

*CONTROL_STAGED_CONSTRUCTION

This control card is used to help break down analyses of construction processes into stages.

Note: This keyword card will be available starting in release 3 of version 971.

Card	1	2	3	4	5	6	7	8
Variable	TSTART	STGS	STGE	ACCEL	FACT	STREF		
Type	F	I	I	F	F	I		
Default	0	0	0	0.0	1.e-6	0		

VARIABLE	DESCRIPTION
TSTART	Time at start of analysis (normally leave blank)
STGS	Construction stage at start of analysis
STGE	Construction stage at end of analysis
ACCEL	Default acceleration for gravity loading
FACT	Default stiffness and gravity factor for parts before they are added
STREF	Reference stage for displacements in d3plot file

Remarks:

See also *DEFINE_CONSTRUCTION_STAGES and *DEFINE_STAGED_CONSTRUCTION_PART.

The staged construction options offer flexibility to carry out the whole construction simulation in one analysis, or to run it stage by stage. Provided that at least one construction stage is defined (*DEFINE_CONSTRUCTION_STAGES), a dynain file will be written at the end of each stage (file names are end_stage001_dynain, etc). These contain node and element definitions and the stress state; the individual stages can then be re-run without re-running the whole analysis. To do this, make a new input file as follows:

- Copy the original input file, containing *DEFINE_CONSTRUCTION_STAGES and *DEFINE_STAGED_CONSTRUCTION_PART.

- Delete node and element definitions as these will be present in the dynain file (*NODE, *ELEMENT_SOLID, *ELEMENT_SHELL, and *ELEMENT_BEAM).

- Delete any *INITIAL cards; the initial stresses in the new analysis will be taken from the dynain file.

- On *CONTROL_STAGED_CONSTRUCTION set STGS to start at the desired stage

- Add an *INCLUDE statement referencing, for example, end_stage002_dynain if starting the new analysis from Stage 3.

- Move or copy the dynain file into the same directory as the new input file.

When STGS is >1 the analysis starts at a non-zero time (the start of stage STGS). In this case a dynain file must be included to start the analysis from the stress state at the end of the previous stage. The end time for stage STGE overrides the termination time on *CONTROL_TERMINATION. A new dynain file will be written at the end of all stages from STGS to STGE.

ACCEL and FACT are used with *STAGED_CONSTRUCTION_PART for simpler input definition of the parts present at different construction stages.

If STGS>1 and elements have been deleted in a previous stage, these elements will be absent from the new analysis and should not be referred to (e.g. *DATABASE_HISTORY_SOLID) in the new input file.

TSTART can be used to set a non-zero start time (again, assuming a compatible dynain file is included). This option is used only if construction stages have not been defined.

STREF allows the user to set a construction stage at the start of which displacements are considered to be zero – e.g. so that initial analysis stages that achieve a pre-construction equilibrium do not contribute to contour plots of displacement. The current coordinates are not modified, only the “initial geometry” coordinates in the d3plot file. If this analysis starts from a stage later than STREF, the reference geometry will be taken from the dynain file that was written at the end of the stage previous to STREF – this dynain file must be in the same directory as the current model for this process to occur.

*CONTROL_STEADY_STATE_ROLLING

Card	1	2	3	4	5	6	7	8
Variable	IMASS	LCDMU	LCDMUR	IVEL	SCL_K			
Type	I	I	I	I	I			
Default	0	0	0	0				

VARIABLE **DESCRIPTION**

IMASS Inertia switching flag
EQ.0: include inertia during an implicit dynamic simulation.
EQ.1: treat steady state rolling subsystems as quasi-static during implicit dynamic simulations.

LCDMU Optional load curve for scaling the friction forces in contact.

LCDMUR Optional load curve for scaling the friction forces in contact during dynamic relaxation. If LCDMUR isn't specified, LCDMU is used.

IVEL Velocity switching flag.
EQ.0: eliminate the steady state rolling body forces and set the velocities of the nodes after dynamic relaxation.
EQ.1: keep the steady state rolling body forces after dynamic relaxation instead of setting the velocities.

SCL_K Scale factor for the friction stiffness during contact loading and unloading. The default values are 1.0 and 0.01 for explicit and implicit, respectively. Any scaling applied here applies only to contact involving the subsystem of parts defined for steady state rolling.

NOTES:

1. Treating the steady state rolling subsystems as quasi-static during an implicit simulation may eliminate vibrations in the system that are not of interest and is generally recommended.
2. Ramping up the friction by scaling it with LCDMU and LCDMUR may improve the convergence behavior of implicit calculations. The values of the load curves should be 0.0 at initial contact and ramp up smoothly to a value of 1.0.
3. After dynamic relaxation, the default behavior is to initialize the nodes with the velocities required to generate the body forces on elements and remove the body

forces. This initialization is skipped, and the body forces retained, after dynamic relaxation if IVEL=1.

4. The friction model in contact is similar to plasticity, where there is an elastic region during the loading and unloading of the friction during contact. The elastic stiffness is scaled from the normal contact stiffness. For implicit calculations, the default scale factor is 0.01, which results in long periods of time being required to build the friction force, and, in some cases, oscillations in the contact forces. A value between 10 and 100 produces smoother solutions and a faster build-up and decay of the friction force as the tire velocity or slip angle is varied, allowing a parameter study to be performed in a single run.

***CONTROL_STRUCTURED_**{*OPTION*}

Available options include:

<BLANK>

TERM

Purpose: Write out a LS-DYNA structured input deck for Version 970. The name of this structured file is “dyna.str”. This input deck will not support all capabilities that are available in Version 970. As a result some data such as load curve numbers will be output in an internal numbering system. If the TERM option is activated termination will occur after the structured input file is written. This option is useful in debugging especially if problems occur in reading the input file.

*CONTROL_SUBCYCLE

Purpose: Control time step subcycling. This feature is described in the LS-DYNA Theory Manual, Section 21.2, and its use may be detrimental in cases of vectorized computation. This keyword activates subcycling. The use of mass scaling to preserve a reasonable time step size often works better than subcycling. To use mass scaling set the input parameter, DT2MS, to the negative value of the minimum acceptable time step size. See the keyword, *CONTROL_TIMESTEP.

*CONTROL_TERMINATION

Purpose: Stop the job.

Card	1	2	3	4	5	6	7	8
Variable	ENDTIM	ENDCYC	DTMIN	ENDENG	ENDMAS	NOSOL		
Type	F	I	F	F	F	I		
Default	0.0	0	0.0	0.0	0.0	0		
Remarks	1		2					

VARIABLE	DESCRIPTION
ENDTIM	Termination time. Mandatory.
ENDCYC	Termination cycle. The termination cycle is optional and will be used if the specified cycle is reached before the termination time. Cycle number is identical with the time step number.
DTMIN	Reduction (or scale) factor for initial time step size to determine minimum time step, TSMIN. TSMIN=DTSTART*DTMIN where DTSTART is the initial step size determined by LS-DYNA. When TSMIN is reached, LS-DYNA terminates with a restart dump.
ENDENG	Percent change in energy ratio for termination of calculation. If undefined, this option is inactive.
ENDMAS	Percent change in the total mass for termination of calculation. This option is relevant if and only if mass scaling is used to limit the minimum time step size, see *CONTROL_TIMESTEP variable name "DT2MS".
NOSOL	Flag for a non-solution run, i.e. normal termination directly after initialization. EQ.0: off (default), EQ.1: on.

Remarks:

1. Termination by displacement may be defined in the *TERMINATION section.

2. If the erosion flag on *CONTROL_TIMESTEP is set (ERODE=1), then the shell elements and solid elements with time steps falling below TSMIN will be eroded.

*CONTROL_THERMAL_NONLINEAR

Purpose: Set parameters for a nonlinear thermal or coupled structural/thermal analysis. The control card, *CONTROL_SOLUTION, is also required.

Card	1	2	3	4	5	6	7	8
Variable	REFMAX	TOL	DCP	LUMPBC	THLSTL	NLTHPR	PHCHPN	
Type	I	F	F	I	F	I	F	
Default	10	1.e-04	1.0 / 0.5	0	0.	0	100.	

VARIABLE	DESCRIPTION
REFMAX	Maximum number of matrix reformations per time step: EQ.0: set to 10 reformations.
TOL	Convergence tolerance for temperature: EQ.0.0: set to 1000 * machine roundoff.
DCP	Divergence control parameter: steady state problems $0.3 \le DCP \le 1.0$ default 1.0 transient problems $0.0 < DCP \le 1.0$ default 0.5
LUMPBC	Lump enclosure radiation boundary condition: EQ.0: off (default) EQ.1: on
THLSTL	Line search convergence tolerance: EQ.0.0: No line search GT.0.0: Line search convergence tolerance
NLTHPR	Thermal nonlinear print out level: EQ.0: No print out EQ.1: 1 Print convergence parameters during solution of nonlinear system
PHCHPN	Phase change penalty parameter: EQ.0.0: Set to default value 100. GT.0.0: Penalty to enforce constant phase change temperature

*CONTROL_THERMAL_SOLVER

Purpose: Set options for the thermal solution in a thermal only or coupled structural-thermal analysis. The control card, *CONTROL_SOLUTION, is also required.

Card 1

Variable	ATYPE	PTYPE	SOLVER	CGTOL	GPT	EQHEAT	FWORK	SBC
Type	I	I	I	F	I	F	F	F
Default	0	0	3	1.0e-04	8	1.	1.	0.

Optional Card (Define if SOLVER = 11, 12, 13, 14, 15 or 16)

Card 2

Variable	MSGLVL	MAXITR	ABSTOL	RELTOL	OMEGA			TSF
Type	I	I	F	F	F			F
Default	0	500	1.0e-10	1.0e-04	1.0 or 0.			1.

VARIABLE	DESCRIPTION
ATYPE	Thermal analysis type: EQ.0: Steady state analysis, EQ.1: transient analysis.
PTYPE	Thermal problem type: (see *CONTROL_THERMAL_NONLINEAR if no-zero) EQ.0: linear problem, EQ.1: nonlinear problem with material properties evaluated at gauss point temperature. EQ.2: nonlinear problem with material properties evaluated at element average temperature.

VARIABLE	DESCRIPTION
SOLVER	Thermal analysis solver type: EQ.1:using solver 11 (enter -1 to use the old ACTCOL solver), EQ.2:nonsymmetric direct solver, EQ.3:diagonal scaled conjugate gradient iterative (default), EQ.4:incomplete choleski conjugate gradient iterative, EQ.5:nonsymmetric diagonal scaled bi-conjugate gradient EQ.11:symmetric direct solver (recommended over #1), For MPP executions: EQ.11:symmetric direct solver, EQ.12:diagonal scaling (default for mpp) conjugate gradient iterative, EQ.13:symmetric Gauss-Siedel conjugate gradient iterative, EQ.14:SSOR conjugate gradient iterative, EQ.15: ILDLT0 (incomplete factorization) conjugate gradient iterative, EQ.16:modified ILDLT0 (incomplete factorization) conjugate gradient iterative.
CGTOL	Convergence tolerance for SOLVER = 3 and 4. EQ.0.0:use default value 1.e-04 single or 1.e-06 double precision
GPT	Number of Gauss points to be used in the solid elements: EQ.0.0:use default value 8, EQ.1.0:one point quadrature is used.
EQHEAT	Mechanical equivalent of heat (e.g., 1 J / N m). EQ.0.0:use default value 1.0, LT.0.0:designates a load curve number for EQHEAT versus time.
FWORK	Fraction of mechanical work converted into heat. EQ.0.0:use default value 1.0.
SBC	Stefan Boltzmann constant. Value is used with enclosure radiation surfaces, see *BOUNDARY_RADIATION_..... LT.0.0:use a smoothing algorithm when calculating view factors to force the row sum=1.
MSGLVL	Output message level (For SOLVER > 10) EQ.0:no output (default), EQ.1:summary information, EQ.2:detailed information, use only for debugging.
MAXITR	Maximum number of iterations. For SOLVER >11. EQ.0:use default value 500,
ABSTOL	Absolute convergence tolerance. For SOLVER >11. EQ.0.0:use default value 1.e-10

VARIABLE	DESCRIPTION
RELTOL	Relative convergence tolerance. Replaces CGTOL for SOLVER >11. EQ.0.0:use default value 1.e-06
OMEGA	Relaxation parameter omega for SOLVER 14 and 16. EQ.0.0:use default value 1.0 for Solver 14, use default value 0.0 for Solver 16.
TSF	Thermal Speedup Factor – this factor multiplies all thermal velocity terms (i.e., those with 1/sec) for artificial time scaling for metal stamping when the puch speed is artificially increased.

Remarks:

1. Solvers 1, 2, 3 and 4 are only for SMP environments. Solvers 11, 12, 13, 14, 15 and 16 are for SMP and MPP.

2. Solver 11 is the preferred direct solver. Solver 11 uses sparse matrix storage and requires much less memory than Solver 1.

3. Use of a direct solver (e.g., SOLVER = 1, 2 or 11) is usually less efficient than using an iterative solver (SOLVER = 3, 4, 12, 13, 14, 15 or 16). Consider using a direct solver to get the model running and then switch to an iterative solver to decrease execution time (particularly for large models). Direct solvers should be used when experiencing slow or no convergence.

4. For transient problems, diagonal scaling conjugate gradient (SOLVER = 3 or 12) should be adequate.

5. For steady state problems, convergence may be slow or unacceptable, so consider using direct solver (SOLVER = 1, 2 or 11) or a more powerful preconditioner (SOLVER = 4, 13, 14, 15 or 16).

6. Solver 13 (symmetric Gauss-Seidel) and solver 14 (SSOR) are related. When OMEGA = 1, solver 14 is equivalent to solver 13. The optimal omega value for SSOR is problem dependent but lies between 1 and 2.

7. Solver 15 (incomplete LDLT0) and solver 16 (modified incomplete LDLT0) are related. Both are no-fill factorizations that require one extra n-vector of storage. The sparsity pattern of the preconditioner is exactly the same as that of the thermal stiffness matrix. Solver 16 uses the relaxation parameter OMEGA. The optimal OMEGA value is problem dependent, but lies between 0 and 1.

8. Solvers 12, 13, 14, 15 and 16 terminate the iterative solution process when (1) the number of iterations exceeds MAXITR or (2) the 2-norm of the residual drops below ABSTOL + RELTOL*2-norm of the initial residual.

*CONTROL_THERMAL_TIMESTEP

Purpose: Set time step controls for the thermal solution in a thermal only or coupled structural/ thermal analysis. Also *CONTROL_SOLUTION, *CONTROL_THERMAL_SOLVER needed.

Card	1	2	3	4	5	6	7	8
Variable	TS	TIP	ITS	TMIN	TMAX	DTEMP	TSCP	LCTS
Type	I	F	F	F	F	F	F	I
Default	0	0.5	none	-	-	1.0	0.5	0

VARIABLE	DESCRIPTION
TS	Time step control: EQ.0: fixed time step, EQ.1: variable time step (may increase or decrease).
TIP	Time integration parameter: EQ.0.0: set to 0.5 - Crank-Nicholson scheme, EQ 1.0: fully implicit.
ITS	Initial thermal time step
TMIN	Minimum thermal time step: EQ.0.0: set to structural explicit time step.
TMAX	Maximum thermal time step: EQ.0.0: set to 100 * structural explicit time step.
DTEMP	Maximum temperature change in each time step above which the thermal time step will be decreased: EQ.0.0: set to a temperature change of 1.0.
TSCP	Time step control parameter. The thermal time step is decreased by this factor if convergence is not obtained. 0. < TSCP < 1.0: EQ.0.0: set to a factor of 0.5.
LCTS	LCTS designates a load curve number which defines data pairs of (thermal time breakpoint, new time step). The time step will be adjusted to hit the time breakpoints exactly. After the time breakpoint, the time step will be set to the 'new time step' ordinate value in the load curve.

*CONTROL_TIMESTEP

Purpose: Set structural time step size control using different options.

Card 1

Variable	DTINIT	TSSFAC	ISDO	TSLIMT	DT2MS	LCTM	ERODE	MS1ST
Type	F	F	I	F	F	I	I	I
Default	-	0.9/0.67	0	0.0	0.0	0	0	0

(This card is optional).

Card 2

Variable	DT2MSF	DT2MSLC	IMSCL			RMSCL		
Type	F	I	I			F		
Default	not used	not used	0			0.0		

VARIABLE **DESCRIPTION**

DTINIT Initial time step size:
EQ.0.0: LS-DYNA determines initial step size.

TSSFAC Scale factor for computed time step (old name SCFT). See Remark 1 below. (Default = .90; if high explosives are used, the default is lowered to .67).

ISDO Basis of time size calculation for 4-node shell elements. 3-node shells use the shortest altitude for options 0,1 and the shortest side for option 2. This option has no relevance to solid elements, which use a length based on the element volume divided by the largest surface area.
EQ.0: characteristic length=area/(minimum of the longest side or the longest diagonal).

VARIABLE	DESCRIPTION
	EQ.1: characteristic length=area/(longest diagonal). EQ.2: based on bar wave speed and MAX **[**shortest side, area/(minimum of the longest side or the longest diagonal).**]**. THIS LAST OPTION CAN GIVE A MUCH LARGER TIME STEP SIZE THAT CAN LEAD TO INSTABILITIES IN SOME APPLICATIONS, ESPECIALLY WHEN TRIANGULAR ELEMENTS ARE USED. EQ.3: time step size is based on the maximum eigenvalue. This option is okay for structural applications where the material sound speed changes slowly. The cost related to determining the maximum eigenvalue is significant, but the increase in the time step size often allows for significantly shorter run times without using mass scaling.
TSLIMT	Shell element minimum time step assignment, TSLIMT. When a shell controls the time step, element material properties (moduli not masses) will be modified such that the time step does not fall below the assigned step size. This option is applicable only to shell elements using material models: *MAT_PLASTIC_KINEMATIC, *MAT_POWER_LAW_PLASTICITY, *MAT_STRAIN_RATE_DEPENDENT_PLASTICITY, *MAT_PIECE- WISE_LINEAR_PLASTICITY. This so-called stiffness scaling option is NOT recommended. The DT2MS option below applies to all materials and element classes and is preferred. If both TSLIMT and DT2MS below are active and if TSLIMT is input as a positive number, then TSLIMT is set to 1.E-18, which makes it inactive. If TSLIMT is negative and less than |DT2MS|, then |TSLIMT| is applied prior to the mass being scaled. If |DT2MS| exceeds the magnitude of TSLIMT, then TSLIMT is set to 1.E-18.
DT2MS	Time step size for mass scaled solutions, DT2MS. Positive values are for quasi-static analyses or time history analyses where the inertial effects are insignificant. Default = 0.0. If negative, TSSFAC*|DT2MS| is the minimum time step size permitted and mass scaling is done if and only if it is necessary to meet the Courant time step size criterion. This latter option can be used in transient analyses if the mass increases remain insignificant. See *CONTROL_TERMINATION variable name "ENDMAS". WARNING: Superelements, *ELEMENT_DIRECT_MATRIX_INPUT, are not mass scaled; consequently, DT2MS does not affect their time step size. In this case an error termination will occur, and DT2MS will need to be reset to a smaller value.
LCTM	Load curve ID that limits the maximum time step size (optional). This load curve defines the maximum time step size permitted versus time. If the solution time exceeds the final time value defined by the curve the computed step size is used. If the time step size from the load curve is exactly zero, the computed time step size is also used.

VARIABLE	DESCRIPTION
ERODE	Erosion flag for solid and t-shell elements when TSMIN (see *CONTROL_TERMINATION) is reached. If this flag is not set the calculation will terminate. For solid elements the PSFAIL option is available and can reduce CPU time, see *CONTROL_SOLID. EQ.0: no, EQ.1: yes. If ERODE=1, and TSMIN>0 (See *CONTROL_TERMINATION), all solid elements are checked at the beginning of element processing to check for negative volumes. The solid elements, which are found with negative volumes, are eroded and the calculation continues.
MS1ST	Limit mass scaling to the first step and fix the mass vector according to the time steps once. The time step will not be fixed but may drop during the calculation from the specified minimum: EQ.0: no, EQ.1: yes.
DT2MSF	Reduction (or scale) factor for initial time step size to determine the minimum time step size permitted. Mass scaling is done if it is necessary to meet the Courant time step size criterion. If this option is used DT2MS= –DT2MSF multiplied by the initial time step size, Δt, before Δt is scaled by TSSFAC. This option is active if and only if DT2MS=0 above.
DT2MSLC	Load curve specifying DT2MS as a function of time during the explicit solutions phase. The load curve can only be used for increasing the magnitude of DT2MS. Consequently, the magnitude of DT2MS is taken as the maximum of the current value and the value from the load curve.
IMSCL	Flag for selective mass scaling if and only if mass scaling active. Selective mass scaling does not scale the rigid body mass and is therefore more accurate. Since it is memory and CPU intensive, it should be applied only to small finely meshed parts. This option is available starting with the third revision of version 971. EQ.0: no selective mass scaling. EQ.1: all parts undergo selective mass scaling. LT.0: recommended. |IMSCL| is the part set ID of the parts that undergo selective mass scaling; all other parts are mass scaled the usual way.
RMSCL	Flag for using rotational option in selective mass scaling. EQ.0.: Only translational inertia are selectively mass scaled NE.0.: Both translational and rotational inertia are selectively mass scaled

Remarks:

1. During the solution we loop through the elements and determine a new time step size by taking the minimum value over all elements.

$$\Delta t^{n+1} = TSSFAC \cdot \min\{\Delta t_1, \Delta t_2, ..., \Delta t_N\}$$

where N is the number of elements. The time step size roughly corresponds to the transient time of an acoustic wave through an element using the shortest characteristic distance. For stability reasons the scale factor TSSFAC is typically set to a value of .90 (default) or some smaller value. To decrease solution time we desire to use the largest possible stable time step size. Values larger than .90 will often lead to instabilities. Some comments follow:

- The sound speed in steel and aluminum is approximately 5mm per microsecond; therefore, if a steel structure is modeled with element sizes of 5mm, the computed time step size would be 1 microsecond. Elements made from materials with lower sound speeds, such as foams, will give larger time step sizes. Avoid excessively small elements and be aware of the effect of rotational inertia on the time step size in the Belytschko beam element. Sound speeds differ for each material, for example, consider:

AIR	331 m/s
WATER	1478
STEEL	5240
TITANIUM	5220
PLEXIGLAS	2598

- Model stiff components with rigid bodies, not by scaling Young's modulus which can substantially reduce the time step size.

- The altitude of the triangular element should be used to compute the time step size. Using the shortest side is okay only if the calculation is closely examined for possible instabilities. This is controlled by parameter ISDO.

- In the explicit time integration context and in contrast to conventional mass scaling,selective mass scaling (SMS) is a well thought out scheme that not only reduces the number of simulation cycles but that also does not significantly affect the dynamic response of the system under consideration. The drawback is that a linear system of equations must be solved in each time step for the accelerations, in this implementation a preconditioned conjugate gradient method (PCG) is used. An unfortunate consequence of this choice of solver is that the efficiency will worsen when attempting large time steps since the condition number of the assembled mass matrix increases with the added mass. Therefore caution should be taken when choosing the desired time step size. For large models it is also recommended to only use SMS on critical parts since it is otherwise likely to slow down execution, once again the bottle neck being the solution of the linear system of equations. Finally, one should be aware that all constraints and boundary conditions available in LS-DYNA may not be supported for SMS but are continuously implemented when requested by users. Up to date and in brief, the following features are supported in this context

Pointwise nodal constraints in global and local directions

Prescribed motion in global and local directions

Adaptivity

Rigid walls

Deformable elements merged with rigid bodies

Regarding contacts only penalty based algorithms are supported, hence constraint contacts cannot be used with SMS. An attempt is being made to automatically switch from constraint based contacts to penalty based when necessary and possible, and this is accompanied with a proper warning message in the standard out, messag and d3hsp files for the user's convenience. By default, only the translational dynamic properties are treated. This means that only rigid body translation will be unaffected by the mass scaling imposed. There is an option to also properly treat rigid body rotation in this way, this is invoked by flagging the parameter RMSCL. A penalty in computational expense is incurred but the results could be improved if rotations are dominating the simulation.

*CONTROL_VIBRO_ACOUSTIC

Purpose: Set vibro-acoustic structural analysis control using different options. Modal stress and strain output can also be activated through this card.

Card 1	1	2	3	4	5	6	7	8
Variable	VAFLAG	VAPRLD	VASTRS	VAPSD	VARMS	VAPLOT	IPANELU	IPANELV
Type	I	I	I	I	I	I	I	I
Default	-	0	0	0	0	0	0	0
Remarks	1						2	

(This card is optional).

Card 2	1	2	3	4	5	6	7	8
Variable	RESTRT	NMDSTR						
Type	I	I						
Default	0	0						
Remarks	3							

VARIABLE	DESCRIPTION
VAFLAG	Loading type: EQ.0: No vibro-acoustic structural analysis. EQ.1: Base acceleration. EQ.2: Random pressure. EQ.3: Plane wave. EQ.4: Shock wave. EQ.5: Progressive wave. EQ.6: Reverberant wave. EQ.7: Turbulent boundary layer wave. EQ.8: Nodal force.

EQ.9: Modal stresses/strains output only

VARIABLE	DESCRIPTION
VAPRLD	Flag for including preload: EQ.0: No preload. EQ.1: Thermal preload due to temperature difference from the neutral temperature. EQ.2: Mechanical preload due to static pressure. EQ.3: Mechanical preload due to concentrated nodal force.
VASTRS	Flag for including stress analysis: EQ.0: No stress analysis, only displacement analysis is requested. EQ.1: Both stress and displacement analyses are requested.
VAPSD	Flag for PSD output: EQ.0: No PSD output is requested. EQ.1: PSD output is requested.
VARMS	Flag for RMS output: EQ.0: No RMS output is requested. EQ.1: RMS output is requested.
VAPLOT	Flag for PSD broadband plots: EQ.0: No PSD broadband plot is requested. EQ.1: PSD broadband plots are requested.
IPANELU	Number of strips in U direction
IPANELV	Number of strips in V direction
RESTART	Restart option: EQ.0: No restart will be requested. All intermediate output is deleted. EQ.1: Intermediate output is retained for restart. EQ.2: Restart based on intermediate output in last run. All intermediate output is deleted after the current run. EQ.3: Restart based on intermediate output in last run. All intermediate output is retained for next restart run.
NMODSTR	Number of modes in modal stresses/strains output.

Remarks:

9. This command evaluates the structural response due to aero acoustic loads, or random excitation from base acceleration or nodal force.
10. The Numbers of strip in U and V direction are used to group the elements into smaller number of integration domains to reduce computational time. This option is only available for VAFLAG=5, 6, and 7.

11. Restart feature allows fast calculation of power spectral density and root mean square results for a new set of target nodes and elements, or a new range of frequencies, without repeating the whole process of modal analysis and modal stress computation.
12. To print out the modal stresses/strains, select VAFLAG=9. The modal stresses/strains for the first NMODSTR modes will be printed out. If NMODSTR is not given, the number of modes in modal stresses/strains output is same as the number of modes, NEIG, from the *CONTROL_IMPLICIT_EIGENVALUE keyword card.
13. For cases with preload, for cases 1, 2, 3, 4 and 8, the load curves defining the acceleration, pressure and nodal force must have starting time larger than the termination time. For example, in the *DEFINE_CURVE keyword, OFFA must be larger than ENDTIM used in *CONTROL_TERMINATION. This makes sure that these loading will not participate in the modal analysis. In the static computation after modal analysis, LS-DYNA will change OFFA to 0.0, to activate these load curves.

VARIABLE	DESCRIPTION
	EQ.1: characteristic length=area/(longest diagonal). EQ.2: based on bar wave speed and MAX **[**shortest side, area/(minimum of the longest side or the longest diagonal).**]**. THIS LAST OPTION CAN GIVE A MUCH LARGER TIME STEP SIZE THAT CAN LEAD TO INSTABILITIES IN SOME APPLICATIONS, ESPECIALLY WHEN TRIANGULAR ELEMENTS ARE USED. EQ.3: time step size is based on the maximum eigenvalue. This option is okay for structural applications where the material sound speed changes slowly. The cost related to determining the maximum eigenvalue is significant, but the increase in the time step size often allows for significantly shorter run times without using mass scaling.

*DAMPING

The Keyword options in this section in alphabetical order are:

***DAMPING_FREQUENCY_RANGE**

***DAMPING_GLOBAL**

***DAMPING_PART_MASS**

***DAMPING_PART_STIFFNESS**

***DAMPING_RELATIVE**

*DAMPING_FREQUENCY_RANGE

Purpose: This feature provides approximately constant damping (i.e. frequency-independent) over a range of frequencies.

Card	1	2	3	4	5	6	7	8
Variable	CDAMP	FLOW	FHIGH	PSID				
Type	F	F	F	I				
Default	0.0	0.0	0.0	0				

VARIABLE	DESCRIPTION
CDAMP	Damping in fraction of critical.
FLOW	Lowest frequency in range of interest (cycles per unit time, e.g. Hz if time unit is seconds)
FHIGH	Highest frequency in range of interest (cycles per unit time, e.g. Hz if time unit is seconds)
PSID	Part set ID. The requested damping is applied only to the parts in the set. If PSID = 0, the damping is applied to all parts except those referred to by other *DAMPING_FREQUENCY_RANGE cards.

This feature provides approximately constant damping (i.e. frequency-independent) over a range of frequencies $F_{low} < F < F_{high}$. It is intended for small damping ratios (e.g. < 0.05) and frequency ranges such that F_{high}/ F_{low} is in the range 10-300. The drawback to this method of damping is that it reduces the dynamic stiffness of the model, especially at low frequencies. This effect is predictable: the natural frequencies of modes close to F_{low} are reduced by 3% for a damping ratio of 0.01 and F_{high}/ F_{low} in the range 10-30. Near F_{high} the error is between zero and one third of the error at F_{low}. Estimated frequency errors are shown in the table below.

Frequency Error at F_{low}	F_{high}/ F_{low}			
		3 to 30	30 to 300	300 to 3000
Damping Ratio	0.01	3%	4.5%	6%
	0.02	6%	9%	12%
	0.04	12%	18%	24%

It is recommended that the elastic stiffnesses in the model be increased slightly to account for this, e.g. for 0.01 damping across a frequency range of 30 to 600Hz, the average error across the frequency range is about 2%. Increase the stiffness by $(1.02)^2$, i.e. by 4%.

*DAMPING_GLOBAL

Purpose: Define mass weighted nodal damping that applies globally to the nodes of deformable bodies and to the mass center of the rigid bodies.

Card	1	2	3	4	5	6	7	8
Variable	LCID	VALDMP	STX	STY	STZ	SRX	SRY	SRZ
Type	I	F	F	F	F	F	F	F
Default	0	0.0	0.0	0.0	0.0	0.0	0.0	0.0
Remarks	1		2	2	2	2	2	2

VARIABLE	DESCRIPTION
LCID	Load curve ID which specifies the system damping constant: EQ.0: a constant damping factor as defined by VALDMP is used, EQ.n: system damping is given by load curve n. The damping force applied to each node is f=-d(t) mv, where d(t) is defined by load curve n.
VALDMP	System damping constant, D_s (this option is bypassed if the load curve number defined above is non zero).
STX	Scale factor on global x translational damping forces.
STY	Scale factor on global y translational damping forces.
STZ	Scale factor on global z translational damping forces.
SRX	Scale factor on global x rotational damping moments.
SRY	Scale factor on global y rotational damping moments.
SRZ	Scale factor on global z rotational damping moments.

Remarks:

1. This keyword is also used for the restart, see *RESTART.

2. If STX=STY=STZ=SRX=SRY=SRZ=0.0 in the input above, all six values are defaulted to unity.

   With mass proportional system damping the acceleration is computed as:

$$a^n = M^{-1}\left(P^n - F^n - F^n_{damp} \right)$$

   where, M is the diagonal mass matrix, P^n is the external load vector, F^n is the internal load vector, and F^n_{damp} is the force vector due to system damping. This latter vector is defined as:

$$F^n_{damp} = D_s m v$$

   The best damping constant for the system is usually some value approaching the critical damping factor for the lowest frequency mode of interest.

$$(D_s)_{critical} = 2\omega_{min}$$

   The natural frequency ω_{min} (given in radians per unit time) is generally taken as the fundamental frequency of the structure. This frequency can be determined from an eigenvalue analysis or from an undamped transient analysis. Note that this damping applies to both translational and rotational degrees of freedom. Also note that mass proportional damping will damp rigid body motion as well as vibration.

   Energy dissipated by through mass weighted damping is reported as system damping energy in the ASCII file GLSTAT. This energy is computed whenever system damping is active.

DAMPING_PART_MASS_*{OPTION}

OPTION specifies that a part set ID is given with the single option:

SET

If not used a part ID is assumed.

Purpose: Define mass weighted damping by part ID. Parts may be either rigid or deformable. In rigid bodies the damping forces and moments act at the center of mass.

Card 1	1	2	3	4	5	6	7	8
Variable	PID/PSID	LCID	SF	FLAG				
Type	I	I	F	I				
Default	0	0	1.0	0				

(This card is optional and is read if and only if FLAG=1. If this card is not read STX, STY, STZ, SRX, SRY, and SRZ default to unity.)

Card 2	1	2	3	4	5	6	7	8
Variable	STX	STY	STZ	SRX	SRY	SRZ		
Type	F	F	F	F	F	F		
Default	0.0	0.0	0.0	0.0	0.0	0.0		

VARIABLE	DESCRIPTION
PID/PSID	Part ID, see *PART or part set ID, see *SET_PART.
LCID	Load curve ID which specifies system damping for parts.
SF	Scale factor for load curve. This allows a simple modification of the load curve values.
FLAG	Set this flag to unity if the global components of the damping forces require separate scale factors.

VARIABLE	DESCRIPTION
STX	Scale factor on global x translational damping forces.
STY	Scale factor on global y translational damping forces.
STZ	Scale factor on global z translational damping forces.
SRX	Scale factor on global x rotational damping moments.
SRY	Scale factor on global y rotational damping moments.
SRZ	Scale factor on global z rotational damping moments.

Remarks:

Mass weighted damping damps all motions including rigid body motions. For high frequency oscillatory motion stiffness weighted damping may be preferred. With mass proportional system damping the acceleration is computed as:

$$a^n = M^{-1}\left(P^n - F^n - F^n_{damp}\right)$$

where, M is the diagonal mass matrix, P^n is the external load vector, F^n is the internal load vector, and F^n_{damp} is the force vector due to system damping. This latter vector is defined as:

$$F^n_{damp} = D_s m v$$

The best damping constant for the system is usually based on the critical damping factor for the lowest frequency mode of interest. Therefore,

$$D_s = 2\omega_{min}$$

is recommended where the natural frequency (given in radians per unit time) is generally taken as the fundamental frequency of the structure. The damping is applied to both translational and rotational degrees of freedom. The component scale factors can be used to limit which global components see damping forces.

Energy dissipated by through mass weighted damping is reported as system damping energy in the ASCII file GLSTAT. This energy is computed whenever system damping is active.

DAMPING_PART_STIFFNESS_*{OPTION}

OPTION specifies that a part set ID is given with the single option:

SET

If not used a part ID is assumed.

Purpose: Assign Rayleigh stiffness damping coefficient by part ID.

Card	1	2	3	4	5	6	7	8
Variable	PID/PSID	COEF						
Type	I	F						
Default	none	0.0						

VARIABLE	DESCRIPTION
PID/PSID	Part ID, see *PART or part set ID, see *SET_PART.
COEF	Rayleigh damping coefficient. Two methods are now available: LT.0.0: Rayleigh damping coefficient is set based on a given frequency and applied uniformly to each element in the part ID. This approach is used in versions of LS-DYNA prior to version 960. See notes below. EQ.0.0: Inactive. GT.0.0: Rayleigh damping coefficient for stiffness weighted damping. Values between 0.01 and 0.25 are recommended. Higher values are strongly discouraged, and values less than 0.01 may have little effect. The damping coefficient is uniquely defined for each element of the part ID.

Remarks:

The damping matrix in Rayleigh damping is defined as:

$$C = \alpha M + \beta K$$

where C, M, and K are the damping, mass, and stiffness matrices, respectively. The constants α. and β are the mass and stiffness proportional damping constants. The mass proportional damping can be treated by system damping, see keywords: *DAMPING_GLOBAL and DAMPING_PART_ MASS. Transforming C with the ith eigenvector ϕ_i gives:

$$\phi_i^t C \phi_i = \phi_i^t \left(\alpha M + \beta K\right) \phi_i = \alpha + \beta \omega_i^2 = 2\omega_i \xi_i \delta_{ij}$$

where ω_i is the ith frequency (radians/unit time) and ξ_i is the corresponding modal damping parameter.

Generally, the stiffness proportional damping is effective for high frequencies and is orthogonal to rigid body motion. Mass proportional damping is more effective for low frequencies and will damp rigid body motion. If a large value of the stiffness based damping coefficient is used, it may be necessary to lower the time step size significantly. This must be done manually by reducing the time step scale factor on the *CONTROL_TIMESTEP control card. Since a good value of β is not easily identified, the coefficient, COEF, is defined such that a value of .10 roughly corresponds to 10% damping in the high frequency domain.

In versions prior to 960, one damping coefficient is defined that applies to all elements of the entire part. With this older approach if 10% of critical damping is sought in the ith mode then set:

$$\beta = \frac{.20}{\omega_i}$$

and input β as a negative number. Typically, β is some fraction of the time step size.

Energy dissipated by Rayleigh damping is computed if and only if the flag, RYLEN, on the control card, *CONTROL_ENERGY is set to 2. This energy is accumulated as element internal energy and is included in the energy balance. In the GLSTAT file this energy will be lumped in with the internal energy.

*DAMPING_RELATIVE

Purpose: Apply damping relative to the motion of a rigid body.

Card	1	2	3	4	5	6	7	8
Variable	CDAMP	FREQ	PIDRB	PSID				
Type	F	F	F	I				
Default	0	0	0	0				

VARIABLE	DESCRIPTION
CDAMP	Fraction of critical damping.
FREQ	Frequency at which CDAMP is to apply (cycles per unit time, e.g. Hz if time unit is seconds).
PIDRB	Part ID of rigid body, see *PART. Motion relative to this rigid body will be damped.
PSID	Part set ID. The requested damping is applied only to the parts in the set.

Remarks:

1. This feature provides damping of vibrations for objects that are moving through space. The vibrations are damped, but not the rigid body motion. This is achieved by calculating the velocity of each node relative to that of a rigid body, and applying a damping force proportional to that velocity. The forces are reacted onto the rigid body such that overall momentum is conserved. It is intended that the rigid body is embedded within the moving object.

2. Vibrations at frequencies below FREQ are damped by more than CDAMP, while those at frequencies above FREQ are damped by less than CDAMP. It is recommended that FREQ be set to the frequency of the lowest mode of vibration.

*DATABASE

The database definitions are optional, but are necessary to obtain output files containing results information. In this section the database keywords are defined in alphabetical order:

DATABASE_OPTION*

***DATABASE_ADAMS**

DATABASE_BINARY_OPTION*

***DATABASE_CPM_SENSOR**

DATABASE_CROSS_SECTION_OPTION1*_{*OPTION2*}

DATABASE_EXTENT_OPTION*

***DATABASE_FORMAT**

***DATABASE_FSI**

***DATABASE_FSI_SENSOR**

DATABASE_HISTORY_OPTION*

***DATABASE_MASSOUT**

***DATABASE_NODAL_FORCE_GROUP**

***DATABASE_PWP_FLOW**

***DATABASE_PWP_OUTPUT**

***DATABASE_SPRING_FORWARD**

***DATABASE_SUPERPLASTIC_FORMING**

***DATABASE_TRACER**

The ordering of the database definition cards in the input file is completely arbitrary.

DATABASE_*OPTION

Options for ASCII files include (if the file is not specified it will not be created):

ABSTAT	Airbag statistics.
AVSFLT	AVS database. See *DATABASE_EXTENT_*OPTION*.
BNDOUT	Boundary condition forces and energy
CURVOUT	Output from *DEFINE_CURVE_FUNCTION.
DEFGEO	Deformed geometry file. (Note that to output this file in Chrysler format insert the following line in your *.cshrc* file: "setenv LSTC_DEFGEO chrysler") The NASBDF file (NASTRAN Bulk Data) is created whenever the DEFGEO file is requested.
DCFAIL	Failure function data for *MAT_SPOTWELD_DAIMLERCHRYSLER
DEFORC	Discrete elements.
ELOUT	Element data. See *DATABASE_HISTORY_*OPTION*. Also, see Card 3 of the *DATABASE_EXTENT_BINARY parameters INTOUT and NODOUT. This latter option will output all integration point data or extrapolated data to the connectivity nodes in a file call ELOUTDET.
GCEOUT	Geometric contact entities.
GLSTAT	Global data. Always obtained if SSSTAT file is activated.
H3OUT	HybridIII rigid body dummies.
JNTFORC	Joint force file
MATSUM	Material energies. See Remarks 1 and 2 below.
MOVIE	MOVIE. See *DATABASE_EXTENT_*OPTION*.
MPGS	MPGS. See *DATABASE_EXTENT_*OPTION*.
NCFORC	Nodal interface forces. See *CONTACT - Card 1 (SPR and MPR)
NODFOR	Nodal force groups. See *DATABASE_NODAL_FORCE_GROUP.
NODOUT(HF)	Nodal point data. See *DATABASE_HISTORY_*OPTION*.
RBDOUT	Rigid body data. See Remark 2 below.
RCFORC	Resultant interface forces. Output in a local coordinate system is available, see *CONTACT, Optional Card C.
RWFORC	Wall forces.
SBTOUT	Seat belt output file
SECFORC	Cross section forces. See *DATABASE_CROSS_SECTION_*OPTION*.
SLEOUT	Sliding interface energy. See *CONTROL_ENERGY
SPCFORC	SPC reaction forces.
SPHOUT	SPH data. See *DATABASE_HISTORY_*OPTION*.
SSSTAT	Subsystem data. See *DATABASE_EXTENT_SSSTAT.
SWFORC	Nodal constraint reaction forces (spot welds and rivets).
TPRINT	Thermal output from a coupled structural/thermal or thermal only analysis.
TRHIST	Tracer particle history information. See *DATABASE_TRACER.

To include global and subsystem mass and inertial properties in the GLSTAT and SSSTAT files add the option _MASS_PROPERTIES as show below. If this option is active the current mass and inertia properties are output including the principle inertias and their axes. Mass of deleted nodes and rigid bodies are not included in the calculated properties.

GLSTAT_MASS_PROPERTIES This is an option for the glstat file to include mass and inertial properties.

SSSTAT_MASS_PROPERTIES This is an option for the ssstat file to include mass and inertial properties for the subsystems.

Card	1	2	3	4	5	6	7	8
Variable	DT	BINARY	LCUR	IOOPT	DTHFF	BINHF		
Type	F	I	I	I	F	I		
Default	0.	1 or 2	none	0.	0.	1 or 2		

VARIABLE	DESCRIPTION
DT	Time interval between outputs. If DT is zero, no output is printed.
BINARY	Flag for binary file EQ.1: ASCII file is written. This is the default on serial and shared memory computers. EQ.2: Data written to a binary database, which contains data that would otherwise be output to the ASCII file. The ASCII file in this case is not created. This is the default on distributed memory computers. EQ.3: ASCII file is written and the data is also written to the binary database (NOTE: this option is only valid for serial and shared memory computers – distributed memory computers will only produce the binary database).
LCUR	Optional load curve ID specifying time interval between dumps.
IOOPT	Flag to govern behavior of the plot frequency load curve defined by LCUR: EQ.1: At the time each plot is generated, the load curve value is added to the current time to determine the next plot time.(this is the default behavior) EQ.2: At the time each plot is generated, the next plot time T is computed so that T = the current time plus the load curve value at time T. EQ.3: A plot is generated for each abscissa point in the load curve definition. The actual value of the load curve is ignored.
DTHF	Optional input for the NODOUT file option only. Time interval between outputs for the high frequency file, NODOUTHF. If DTHF is

zero, no output is printed. Nodal points that are to be output at a higher frequency are flagged in the DATABASE_HISTORY input.

VARIABLE	DESCRIPTION
BINHF	Optional input for the NODOUTHF file only. Flag for binary file for the high frequency NODOUTHF file. See BINARY above.

The file names and corresponding unit numbers are:

	I/O UNIT #	FILE NAME
Airbag statistics	i/o unit #43	ABSTAT
ASCII database	i/o unit #44	AVSFLT
Boundary conditions	i/o unit #46	BNDOUT (nodal forces and energies)
Smug animator database	i/o unit#40	DEFGEO
Discrete elements	i/o unit#36	DEFORC
Element data	i/o unit#34	ELOUT
Contact entities	i/o unit #48	GCEOUT
Global data	i/o unit#35	GLSTAT
Joint forces	i/o unit #53	JNTFORC
Material energies	i/o unit#37	MATSUM
MOVIE file family	i/o unit #50	MOVIEnnn.xxx where.nnn=001-999
MPGS file family	i/o unit #50	MPGSnnn.xxx where nnn=001-999
Nastran/BDF file	i/o unit#49	NASBDF (see comment below)
Nodal interface forces	i/o unit#38	NCFORC
Nodal force group	i/o unit #45	NODFOR
Nodal point data	i/o unit#33	NODOUT
Rigid body data	i/o unit #47	RBDOUT
Resultant interface forces	i/o unit#39	RCFORC
Rigidwall forces	i/o unit#32	RWFORC
Seat belts	i/o unit #52	SBTOUT
Cross-section forces	i/o unit#31	SECFORC
Interface energies	i/o unit #51	SLEOUT
SPC reaction forces	i/o unit#41	SPCFORC
SPH element data	i/o unit#68	SPHOUT
Subsystems statistics	i/o unit#58	SSSTAT
Nodal constraint resultants	i/o unit #42	SWFORC (spot welds/rivets)
Thermal output	i/o unit #73	TPRINT
Tracer particles	i/o unit #70	TRHIST

Output Components for ASCII Files

ABSTAT	BNDOUT	DCFAIL	DEFORC
volume	x, y, z force	failure function	x, y, z force
pressure		normal term	
internal energy		bending term	
input mass flow rate		shear term	
output mass flow rate		weld area	
mass		effective strain rate	
temperature			
density			

ELOUT			
Beam	Stress Shell	Brick	Strain Shell
axial force resultant	xx, yy, zz stress	xx, yy, zz stress	xx, yy, zz strain
s shear resultant	xy, yz, zx stress	xy, yz, zx stress	xy, yz, zx strain
t shear resultant	plastic strain	effective stress	lower surface strain
s moment resultant		yield function	upper surface strain
t moment resultant			
torsional resultant			

GCEOUT	
x, y, z force	x, y, z moment

GLSTAT	
time step	total energy
kinetic energy	external work
internal energy	total energy / initial energy
spring & damper energy	energy ratio w/o eroded energy
hourglass energy	element id controlling time step
system damping energy	global x, y, z velocity
sliding interface energy	time per zone cycle

eroded kinetic energy	**joint internal energy**
eroded internal energy	**stonewall energy**
eroded hourglass energy	**rigid body stopper energy**

JNTFORC	
x, y, z force	**x, y, z moment**

MATSUM	
kinetic energy	**x, y, z rigid body velocity**
internal energy	**eroded internal energy**
hourglass energy	**eroded kinetic energy**
x, y, z momentum	**added mass**

NCFORC	NODOUT	NODFOR
x force	**x, y, z displacement**	**x, y, z force**
y force	**x, y, z velocity**	
z force	**x, y, z acceleration**	
	x, y, z rotation	
	x, y, z rotational velocity	
	x, y, z rotational acceleration	

RBDOUT	RCFORC	RWFORC
x, y, z displacement	**x, y, z force**	**normal**
x, y, z velocity	**Mass of nodes in contact**	**x, y, z force**
x, y, z acceleration		

SECFORC	SLEOUT	SPCFORC	SWFORC
x, y, z force	**slave energy**	**x, y, z force**	**axial force**
x, y, z moment	**master energy**	**x, y, z moment**	**shear force**
x, y, z center	**frictional energy**		
area			
resultant force			

Remarks:

1. The kinetic energy quantities in the MATSUM and GLSTAT files may differ slightly in values for several reasons. First, the energy associated with added mass (from mass-scaling) is included in the GLSTAT calculation, but is not included in MATSUM. Secondly, the energies are computed element by element in MATSUM for the deformable

materials and, consequently, nodes which are merged with rigid bodies will also have their kinetic energy included in the rigid body total. Furthermore, kinetic energy is computed from nodal velocities in GLSTAT and from element midpoint velocities in MATSUM.

2. The PRINT option in the part definition allows some control over the extent of the data that is written into the MATSUM and RBDOUT files. If the print option is used the variable PRBF can be defined such that the following numbers take on the meanings:

   EQ.0: default is taken from the keyword *CONTROL_OUTPUT,

   EQ.1: write data into RBDOUT file only,
   EQ.2: write data into MATSUM file only,
   EQ.3: do not write data into RBDOUT and MATSUM.

   Also see CONTROL_OUTPUT and PART_PRINT.

3. This keyword is also used in the restart phase, see *RESTART. Thus, the output interval can be changed when restarting.

4. All information in the files except in AVSFLT, MOVIE, AND MPGS can also be plotted using the post-processor LS-PREPOST. Arbitrary cross plotting of results between ASCII files is easily handled.

5. Resultant contact forces reported in RCFORC are averaged over the preceding output interval.

6. "Spring and damper energy" reported in GLSTAT is a subset of "Internal energy". The "Spring and damper energy" includes internal energy of discrete elements, seatbelt elements, and that associated with joint stiffness (see *CONSTRAINED_JOINT_STIFFNESS_...).

*DATABASE_RCFORC_MOMENTS

Purpose: Define contact ID and nodes for moment calculations..

Card	1	2	3	4	5	6	7	8
Variable	CID	NODES	NODEM					
Type	I	I	I					

VARIABLE	DESCRIPTION
CID	Contact ID
NODES	Node for moment calculation on slave surface.
NODEM	Node for moment calculation on master surface.

*DATABASE_ADAMS

Purpose: Request output of an MDI Modal Neutral File for later use in the ADAMS software.

Card	1	2	3	4	5	6	7	8
Variable	IFLAG	M_UNITS	L_UNITS	T_UNITS				
Type	I	F	F	F				
Default	0	none	none	none				

VARIABLE **DESCRIPTION**

IFLAG Flag controlling write of modal neutral file after eigenvalue analysis
- EQ.0: do not write (default),
- EQ.1: write to file "d3mnf"

M_UNITS Mass units of measure used in this model.
- EQ.-1: kilogram
- EQ.-2: gram
- EQ.-3: megagram (metric ton)
- EQ.-4: lbf*sec**2/in (psi-compatible)
- EQ.-5: slug
- EQ.-6: pound-mass

L_UNITS Length units of measure used in this model.
- EQ.-1: meter
- EQ.-2: centimeter
- EQ.-3: millimeter
- EQ.-4: inch
- EQ.-5: foot

T_UNITS Time units of measure used in this model.
- EQ.-1: second
- EQ.-2: millisecond
- EQ.-3: minute
- EQ.-4: hour

Remarks:

1. This option is not available for every platform. Check LS-DYNA Banner upon execution of the program to see if this feature is enabled.
2. Models must be created using a combination of the above units.

DATABASE_BINARY_OPTION*

Options for binary output files with the default names given include:

BLSTFOR	Blast pressure database. See also *LOAD_BLAST_ENHANCED.
D3DRLF	Dynamic relaxation database.
D3DUMP	Binary output restart files. Define output frequency in cycles.
D3PART	Dt for partial output states See also *DATABASE_EXTENT_BINARY.
D3PLOT	Dt for complete output states. See also *DATABASE_EXTENT_BINARY.
D3PROP	Output property data.
D3THDT	Dt for time history data of element subsets. See *DATABASE_HISTORY.
FSIFOR	ALE interface force database (please see Remark 1).
RUNRSF	Binary output restart file. Define output frequency in cycles.
INTFOR	Dt for output of contact interface data (file name must be given on the execution line using "S="). Also see *CONTACT variables mpr and spr.
XTFILE	Flag to specify output of extra time history data to XTFILE at same time as D3THDT file. The following card is left blank for this option.
D3CRACK	Dt for output of crack data file for the Winfrith concrete model (file name must be given on the execution line using "q="). This file can be used with the D3PLOT file to show crack formation of the deformed concrete materials.

The D3DUMP and the RUNRSF options create complete databases which are necessary for restarts, see *RESTART. When RUNRSF is specified, the same file is overwritten after each interval, an option allows a series of files to be overwritten in a cyclic order. When D3DUMP is specified, a new restart file is created after each interval. When D3DUMP is specified, a new restart file is created after each interval, thus a “family” of files is created numbered sequentially D3DUMP01, D3DUMP02, etc. The default file names are RUNRSF and D3DUMP unless other names are specified on the execution line, see the INTRODUCTION, EXECUTION SYNTAX. Since all data held in memory is written into the restart files, these files can be quite large and care should be taken with the D3DUMP files not to create too many. If *DATABASE_BINARY_D3PLOT is not specified in the keyword deck then a complete output state will be written ever time step.

The D3PLOT, D3PART, D3DRLF, and the INTFOR files contain plotting information to plot data over the three dimensional geometry of the model. These databases can be plotted with LS-PREPOST. The D3THDT file contains time history data for element subsets as well as global information, see *DATABASE_HISTORY. This data can be plotted with LS-PREPOST. The default names for the D3PLOT, D3PART, D3DRLF, and the D3THDT files are D3PLOT, D3PART, D3DRLF, and D3THDT. For INTFOR a unique name must be specified on the execution line with S=iff, (iff=file name), for FSIFOR a unique name must be specified on the execution line with h=iff, (iff=file name), see the INTRODUCTION, EXECUTION SYNTAX. The file structure is such that each file contains the full geometry at the beginning, followed by the analysis generated output data at the specified time intervals. For the contents of the D3PLOT, D3PART and D3THDT files see also the *DATABASE_EXTENT_BINARY definition. It is possible to severely restrict the information that is dumped and consequently reduce the size of the databases. The contents of the D3THDT file are also specified with the

*DATABASE_HISTORY definition. It should also be noted in particular that the databases can be considerably reduced for models with rigid bodies containing many elements.

Card	1	2	3	4	5	6	7	8
Variable	DT/CYCL	LCDT/NR	BEAM	NPLTC	PSETID			
Type	F	I	I	I	I			
Default	-	-	-	-	-			
Remarks								

Optional Card that only applies to the D3PLOT database

Card	1	2	3	4	5	6	7	8
Variable	IOOPT							
Type	I							
Default	0							
Remarks								

Use only for D3PROP option (No other cards are necessary)

Card	1	2	3	4	5	6	7	8
Variable	IFILE	IMATL	IWALL					
Type	I	I	I					
Default	1	0	0					

VARIABLE	DESCRIPTION
DT	Time interval between outputs.
CYCL	Output interval in time steps (a time step is a cycle). For the D3DRFL file a positive number 'n' will cause plot dumps to be written at every n'th convergence check interval specified on the *CONTROL_ DYNAMIC_RELAXATION card.
NR	Number of Running Restart Files, RUNRSF, written in a cyclical fashion. The default number is one, i.e. the same file is overwritten each time.
LCDT	Optional load curve ID specifying time interval between dumps. This option is only available for the D3PLOT, D3PART, D3THDT and INTFOR files.
BEAM	Option flag for *DATABASE_BINARY_D3PLOT or D3PART. EQ.0: Discrete spring and damper elements are added to the D3PLOT or D3PART database where they are display as beam elements. The element global X, global Y, global Z and resultant forces are written to the database, EQ.1: No discrete spring and damper elements are added to the D3PLOT or D3PART database. This option is useful when translating old LS-DYNA input decks to KEYWORD input. In older input decks there is no requirement that beam and spring elements have unique ID's, and beam elements may be created for the spring and dampers with identical ID's to existing beam elements causing a fatal error. Contact interfaces which are based on part IDs of seatbelt elements will not be properly generated if this option is used. EQ.2: Discrete spring and damper elements are added to the D3PLOT or D3PART database where they are displayed as beam elements (similar to option 0). In this option the element resultant force is written to its first database position allowing beam axial forces and spring resultant forces to be plotted at the same time. This can be useful during some post-processing applications.
NPLTC	DT=ENDTIME/NPLTC applies to D3PLOT and D3PART only. This overrides the DT specified in the first field.
PSETID	SET_PART ID for D3PART only.
IOOPT	This option applies to the D3PLOT file only. Flag to govern behavior of the plot frequency load curve defined by LCDT: EQ.1: At the time each plot is generated, the load curve value is added to the current time to determine the next plot time (this is the default behavior).

VARIABLE	DESCRIPTION
	EQ.2: At the time each plot is generated, the next plot time T is computed so that T = the current time plus the load curve value at time T. EQ.3: A plot is generated for each abscissa point in the load curve definition. The actual value of the load curve is ignored.
IFILE	Specify file for D3PROP output. (This can also be defined on the command line by adding d3prop = 1 or d3prop = 2 which also sets IMATL = IWALL = 1) EQ.1: Output data at the end of the first d3plot file. EQ.2: Output data to the file d3prop.
IMATL	Output *EOS, *HOURGLASS, *MAT, *PART and *SECTION data. EQ.0: No EQ.1: Yes
IWALL	Output *RIGIDWALL data. EQ.0: No EQ.1: Yes

Remarks:

1. When *DATABASE_FSI is defined, a few pieces of coupling information of some Lagrangian surface entities interacting with the ALE materials may be output as history parameters into a file called "dbfsi". Coupling pressure is one of the output variables. This coupling pressure is averaged over each surface entity. To obtain coupling pressure contour variations over each segment, use *DATABASE_BINARY_FSIFOR. To use it, three things must be done:

   1) The INTFORC parameter (*CONSTRAINED_LAGRANGE_IN_SOLID, 4^{th} row, 3^{rd} column) must be turned ON (INTFORC=1).

   2) A *DATABASE_BINARY_FSIFOR card is defined controlling the output interval. The time interval between output is defined by the parameter DT in this card.

   3) This interface force file is activated by executing ls970 as follow:

      ls970 i=inputfilename.k … h=interfaceforcefilename

      LSDYNA will then writes out the segment coupling pressure and forces to a binary interface force file for contour plotting over the whole simulation interval.

      To plot the binary data in this file, type: lsprepost interfaceforcefilename.

For example, when all 3 of the above actions are taken, and let's assume we define the interfaceforcefilename =fsifor ➔ a series of "fsifor##" binary files are output for contour plotting. To plot this, type "lsprepost fsifor" (without the double quotes).

*DATABASE_CPM_SENSOR

Purpose: This card activates an ASCII file "cpm_sensor". Its input defines sensors' locations based on the positions of some Lagrangian segments. The output gives the history of the velocity, temperature, density and pressure averaged on the number of particles contained in the sensors. This card is activated only when the *AIRBAG_PARTICLE card is used.

Card 1 Format

Card 1	1	2	3	4	5	6	7	8
Variable	DT	BINARY						
Type	F	I						

Card 2,…, Define one segment set per line. Input stops when the next "*" Keyword is found.

Card 2,3,...	1	2	3	4	5	6	7	8
Variable	SEGSID	OFFSET	RADIUS					
Type	I	F	F					

VARIABLE	DESCRIPTION
DT	Output interval
BINARY	Flag for the binary file EQ.1: ASCII file is written, EQ.2: Data written to the binary file "binout", EQ.3: ASCII file is written and the data written to the binary file "binout"
SEGSID	Segment set ID
OFFSET	Offset distance between the center of the sensor and the segment center. If it is positive, it is on the side pointed to by the segment normal vector. See remarks1 and 3.
RADIUS	Radius of the sensor. See remarks 2 and 3.

Remarks:

1. Each segment has a sensor. The distance between the segment center and the sensor center is defined by OFFSET (2nd parameter on the 2nd line) in the normal direction defined by the segment. This distance is constant: the sensor moves along with the segment.

2. The sensor is a sphere with a radius given by RADIUS (3rd parameter on the 2nd line).

3. OFFSET should be larger than RADIUS to prevent the segment from cutting the sphere.

4. The output parameters in the "cpm_sensor" file are:

   velx = x-velocity
   vely = y-velocity
   velz = z-velocity
   velr = velocity
   temp = temperature
   dens =density
   pres =pressure

   These values are averaged on the number of particles in the sensor. RADIUS should be large enough to contain a reasonable number of particles for the averages.

```
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|.
$ INPUT:
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|.
*DATABASE_CPM_SENSOR
      0.01
$   SEGSID    OFFSET    RADIUS
       123       5.0       5.0
       124      -0.2       0.1
       125       0.7       0.6
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|..
$ The segment set id: 123 has 1 segment.
$ The segment set id: 123 has 1 segment.
$ The segment set id: 123 has 11 segments.
$ Each segment has an ID defined in D3HSP
$ The D3HSP file looks like the following:
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|..
Segments for sensor          1
  Sensor id        n1        n2        n3        n4
         1      3842      3843      3848      3847
```

Segments for sensor 2

Sensor id	n1	n2	n3	n4
2	3947	3948	3953	3952

Segments for sensor 3

Sensor id	n1	n2	n3	n4
3	3867	3868	2146	2145
4	3862	3863	3868	3867
5	3857	3858	3863	3862
6	3852	3853	3858	3857
7	3847	3848	3853	3852
8	3837	3838	3843	3842
9	3842	3843	3848	3847
10	3832	3833	3838	3837
11	3827	3828	3833	3832
12	3822	3823	3828	3827
13	1125	1126	3823	3822

```
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|..
```

DATABASE_CROSS_SECTION_OPTION1_*{*OPTION2*}

Option 1 includes:

PLANE

SET

To define an ID and heading for the database cross section use the option:

ID

Purpose: Define a cross section for resultant forces written to ASCII file SECFORC. For the PLANE option, a set of two cards is required for each cross section. Then a cutting plane has to be defined, see Figure 10.1. If the SET option is used, just one card is needed which identifies a node set and at least one element set. In this latter case the node set defines the cross section and the forces from the elements belonging to the element set(s) are summed up to form the section forces. Thus the element set(s) should include elements to only one side (not both sides) of the cross section. The cross-section should cut through deformable elements only, not rigid bodies. Cutting through master segments for deformable solid element spot welds can lead to incorrect section forces since the constraint forces are not accounted for in the force and moment summations. Beam element modeling of welds does *not* require any special precautions.

The following card is read if and only if the ID option is specified.

Optional 1 2-8

Variable	CSID	HEADING
Type	I	A70

The heading is picked up by some of the peripheral LS-DYNA codes to aid in post-processing.

VARIABLE	DESCRIPTION
CSID	Cross section ID. This must be a unique number.
HEADING	Cross section descriptor. It is suggested that unique descriptions be used.

Card (1 of 2) for the PLANE option

Card	1	2	3	4	5	6	7	8
Variable	PSID	XCT	YCT	ZCT	XCH	YCH	ZCH	RADIUS
Type	I	F	F	F	F	F	F	F
Default	0	0.	0.	0.	0.	0.	0.	0.

Card (2 of 2) for the PLANE option

Card	1	2	3	4	5	6	7	8
Variable	XHEV	YHEV	ZHEV	LENL	LENM	ID	ITYPE	
Type	F	F	F	F	F	I	I	
Default	0.	0.	0.	infinity	infinity	global	0	

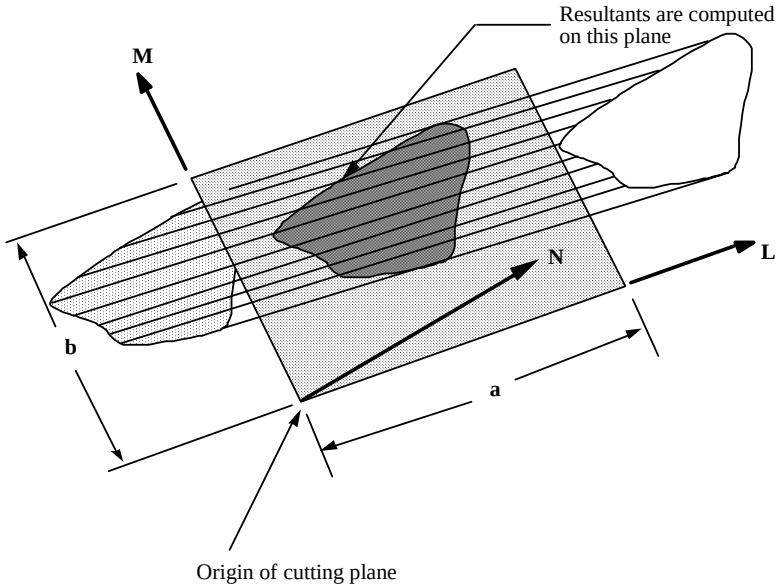


Figure 10.1. Definition of cutting plane for automatic definition of interface for cross-sectional forces. The automatic definition does not check for springs and dampers in the section. For best results the cutting plane should cleanly pass through the middle of the elements, distributing them equally on either side. Elements that intersect the edges of the cutting plane are deleted from the cross-section.

The set option requires that the equivalent of the automatically generated input via the cutting plane be identified manually and defined in sets. All nodes in the cross-section and their related elements that contribute to the cross-sectional force resultants should be defined.

Card (1 of 1) for the SET option

Card	1	2	3	4	5	6	7	8
Variable	NSID	HSID	BSID	SSID	TSID	DSID	ID	ITYPE
Type	I	I	I	I	I	I	I	I
Default	required	0	0	0	0	0	global	0

VARIABLE	DESCRIPTION
CSID	Optional ID for cross section. If not specified cross section ID is taken to be the cross section order in the input deck.
PSID	Part set ID. If zero all parts are included.
XCT	x-coordinate of tail of any outward drawn normal vector, **N**, originating on wall (tail) and terminating in space (head), see Figure 10.1.
YCT	y-coordinate of tail of normal vector, **N**.
ZCT	z-coordinate of tail of normal vector, **N**.
XCH	x-coordinate of head of normal vector, **N**.
YCH	y-coordinate of head of normal vector, **N**.
ZCH	z-coordinate of head of normal vector, **N**.
RADIUS	Optional radius. If RADIUS > 0., a circular cut plane centered at (XCT, YCT ,ZCT) of radius=RADIUS, with the normal vector originating at (XCT, YCT, ZCT) and pointing towards (XCH, YCH, ZCH) will be created. In this case the variables XHEV, YHEV, ZHEV, LENL, and LENM, which are defined on the 2nd card will be ignored.
XHEV	x-coordinate of head of edge vector, **L**.
YHEV	y-coordinate of head of edge vector, **L**.
ZHEV	z-coordinate of head of edge vector, **L**.

VARIABLE	DESCRIPTION
LENL	Length of edge a, in **L** direction.
LENM	Length of edge b, in **M** direction.
NSID	Nodal set ID, see *SET_NODE_*OPTION*.
HSID	Solid element set ID, see *SET_SOLID.
BSID	Beam element set ID, see *SET_BEAM.
SSID	Shell element set ID, see *SET_SHELL_*OPTION*.
TSID	Thick shell element set ID, see *SET_TSHELL.
DSID	Discrete element set ID, see *SET_DISCRETE.
ID	Rigid body (see *MAT_RIGID, type 20), accelerometer ID (see *ELEMENT_ SEATBELT_ACCELEROMETER) or coordinate ID, see *DEFINE_COORDINATE_NODES. The force resultants are output in the updated local system of the rigid body, accelerometer, or coordinate system.
ITYPE	Flag for local system type: EQ.0: rigid body, EQ.1: accelerometer, EQ.2: coordinate ID.

DATABASE_EXTENT_OPTION*

Available options include:

AVS

BINARY

MOVIE

MPGS

SSSTAT

Purpose: Specify output database to be written. Binary applies to the data written to the D3PLOT, D3PART, and D3THDT files. See *DATABASE_BINARY_*OPTION*.

For the AVS, MPGS, and MOVIE options the following cards apply:

Define as many cards as necessary. The created MPGS and MOVIE databases consist of a geometry file and one file for each output database.

Card	1	2	3	4	5	6	7	8
Variable	VTYPE	COMP						
Type	I	I						

VARIABLE	DESCRIPTION
VTYPE	Variable type: EQ.0: node, EQ.1: brick, EQ.2: beam, EQ.3: shell, EQ.4: thick shell.
COMP	Component ID. For the corresponding VTYPE, integer components from the following tables can be chosen: VTYPE.EQ.0: Table 10.1, VTYPE.EQ.1: Table 10.2, VTYPE.EQ.2: not supported, VTYPE.EQ.3: Table 10.3, VTYPE.EQ.4: not supported.

Remarks:

The AVS database consists of a title card, then a control card defining the number of nodes, brick-like elements, beam elements, shell elements, and the number of nodal vectors, NV, written for each output interval. The next NV lines consist of character strings that describe the nodal vectors. Nodal coordinates and element connectivity follow. For each state the solution time is written, followed by the data requested below. The last word in the file is the number of states. We recommend creating this file and examining its contents, since the organization is relatively transparent. The MOVIE and MPGS database are widely used and will be familiar with users who are currently using these databases.

Table 10.1. Nodal Quantities

Component ID	Quantity
1	x, y, z-displacements
2	x, y, z-velocities
3	x, y, z-accelerations

Table 10.2. Brick Element Quantities

Component ID	Quantity
1	x-stress
2	y-stress
3	z-stress
4	xy-stress
5	yz-stress
6	zx-stress
7	effective plastic strain

Table 10.3. Shell and Thick Shell Element Quantities

Component ID	Quantity
1	midsurface x-stress
2	midsurface y-stress
3	midsurface z-stress
4	midsurface xy-stress
5	midsurface yz-stress
6	midsurface xz-stress
7	midsurface effective plastic strain
8	inner surface x-stress
9	inner surface y-stress
10	inner surface z-stress
11	inner surface xy-stress
12	inner surface yz-stress
13	inner surface zx-stress
14	inner surface effective plastic strain
15	outer surface x-stress
16	outer surface y-stress

Table 10.3. Shell and Thick Shell Element Quantities (cont.).

Component ID	Quantity
17	outer surface z-stress
18	outer surface xy-stress
19	outer surface yz-stress
20	outer surface zx-stress
21	outer surface effective plastic strain
22	bending moment-mxx (4-node shell)
23	bending moment-myy (4-node shell)
24	bending moment-mxy (4-node shell)
25	shear resultant-qxx (4-node shell)
26	shear resultant-qyy (4-node shell)
27	normal resultant-nxx (4-node shell)
28	normal resultant-nxx (4-node shell)
29	normal resultant-nxx (4-node shell)
30	thickness (4-node shell)
31	element dependent variable
32	element dependent variable
33	inner surface x-strain
34	inner surface y-strain
35	inner surface z-strain
36	inner surface xy-strain
37	inner surface yz-strain
38	inner surface zx-strain
39	outer surface x-strain
40	outer surface y-strain
41	outer surface z-strain
42	outer surface xy-strain
43	outer surface yz-strain
44	outer surface zx-strain
45	internal energy
46	midsurface effective stress
47	inner surface effective stress
48	outer surface effective stress
49	midsurface max. principal strain
50	through thickness strain
51	midsurface min. principal strain
52	lower surface effective strain
53	lower surface max. principal strain
54	through thickness strain
55	lower surface min. principal strain
56	lower surface effective strain
57	upper surface max. principal strain
58	through thickness strain
59	upper surface min. principal strain
60	upper surface effective strain

Table 10.4. Beam Element Quantities

Component ID	Quantity
1	x-force resultant
2	y-force resultant
3	z-force resultant
4	x-moment resultant
5	y-moment resultant
6	z-moment resultant

For the BINARY option the following cards apply (Card 3 is optional):

Card 1

Variable	1	2	3	4	5	6	7	8
	NEIPH	NEIPS	MAXINT	STRFLG	SIGFLG	EPSFLG	RLTFLG	ENGFLG
Type	I	I	I	I	I	I	I	I
Default	0	0	3	0	1	1	1	1
Remarks			1					

Card 2

Variable	1	2	3	4	5	6	7	8
	CMPFLG	IEVERP	BEAMIP	DCOMP	SHGE	STSSZ	N3THDT	IALEMAT
Type	I	I	I	I	I	I	I	I
Default	0	0	0	0	0	0	2	1
Remarks			2					

Card 3

Variable	1	2	3	4	5	6	7	8
	NINTSLD	PKP_SEN	SCLP	HYDRO	MSSCL	THERM	INTOUT	NODOUT
Type	I	I	F	I	I	I	A	A
Default	1	0	1.0	0	0	0	none	none
Remarks							4-10	4-10

Card 4

Variable	DTDT							
Type	I							
Default	1							
Remarks								

VARIABLE	DESCRIPTION
NEIPH	Number of additional integration point history variables written to the binary database for solid elements. The integration point data is written in the same order that it is stored in memory-each material model has its own history variables that are stored. For user defined materials it is important to store the history data that is needed for plotting before the data which is not of interest.
NEIPS	Number of additional integration point history variables written to the binary database for both shell and thick shell elements for each integration point, see NEIPH above.
MAXINT	Number of shell integration points written to the binary database, see also *INTEGRATION_SHELL. If the default value of 3 is used then results are output for the outermost (top) and innermost (bottom) integration points together with results for the neutral axis. If MAXINT is set to 3 and the element has 1 integration point then all three results will be the same. If a value other than 3 is used then results for the first MAXINT integration points in the element will be output. Note: If the element has an even number of integration points and MAXINT is not set to 3 then you will not get mid-surface results. See Remarks below. If MAXINT is set to a negative number, MAXINT integration points are output for each in plane integration point location and no averaging is used. This can greatly increase the size of the binary databases D3PLOT, D3THDT, and D3PART.
STRFLG	Set to 1 to dump strain tensors for solid, shell and thick shell elements for plotting by LS-PREPOST and ASCII file ELOUT. For shell and thick shell elements two tensors are written, one at the innermost and one at the outermost integration point. For solid elements a single strain tensor is written.

VARIABLE	DESCRIPTION
SIGFLG	Flag for including stress tensor in the shell LS-DYNA database: EQ.1: include (default), EQ.2: exclude.
EPSFLG	Flag for including the effective plastic strains in the shell LS-DYNA database: EQ.1: include (default), EQ.2: exclude.
RLTFLG	Flag for including stress resultants in the shell LS-DYNA database: EQ.1: include (default), EQ.2: exclude.
ENGFLG	Flag for including shell internal energy density and thickness in the LS-DYNA database: EQ.1: include (default), EQ.2: exclude.
CMPFLG	Orthotropic and anisotropic material stress and strain output in local material coordinate system for solids, shells and thick shells. This option applies to material types 22, 23, 33, 34, 36, 40, 41-50, 54-56, 58, 59, 103, 116, and 194. It does not apply to material type 2. EQ.0: global, EQ.1: local.
IEVERP	Every plot state for "d3plot" database is written to a separate file. This option will limit the database to 1000 states: EQ.0: more than one state can be on each plotfile, EQ.1: one state only on each plotfile.
BEAMIP	Number of beam integration points for output. This option does not apply to beams that use a resultant formulation.
DCOMP	Data compression to eliminate rigid body data: EQ.1: off (default), no rigid body data compression, EQ.2: on, rigid body data compression active, EQ.3: off, no rigid body data compression, but nodal velocities and accelerations are eliminated from the database. EQ.4: on, rigid body data compression active and nodal velocities and accelerations are eliminated from the database.
SHGE	Output shell hourglass energy density: EQ.1: off (default), no hourglass energy written, EQ.2: on.
STSSZ	Output shell element time step, mass, or added mass: EQ.1: off (default), EQ.2: output time step size,

VARIABLE	DESCRIPTION
STSSZ	Output shell element time step, mass, or added mass: EQ.1: off (default), EQ.2: output time step size, EQ.3: output mass, added mass, or time step size. See remark 3 below.
N3THDT	Material energy write option for D3THDT database EQ.1: off, energy is NOT written to D3THDT database, EQ.2: on (default), energy is written to D3THDT database.
IALEMAT	Output solid part ID list containing ale materials. EQ.1: on (default)
NINTSLD	Number of solid element integration points written to the LS-DYNA database. The default value is 1. For solids with multiple integration points NINTSLD may be set to 8. Currently, no other values for NINTSLD are allowed. For solids with multiple integration points, an average value is output if NINTSLD is set to 1.
PKP_SEN	Flag to output the peak pressure and surface energy computed by each contact interface into the interface force database. To obtain the surface energy, FRCENG, must be sent to 1 on the control contact card. When PKP_SEN=1, it is possible to identify the energies generated on the upper and lower shell surfaces, which is important in metal forming applications. This data is mapped after each H-adaptive remeshing. EQ.0: No data is written EQ.1: Output the peak pressures and surface energy by contact interface
SCLP	A scaling parameter used in the computation of the peak pressure. This parameter is generally set to unity (the default), but it must be greater than 0.
HYDRO	Either 3 or 5 additional history variables useful to shock physics are output as the last history variables. For HYDRO=1, the internal energy per reference volume, the reference volume, and the value of the bulk viscosity are added to the database, and for HYDRO=2, the volume strain and current density are also added.
MSSCL	Output nodal information related to mass scaling into the D3PLOT database. This option can be activated if and only if DT2MS < 0.0, see control card *CONTROL_TIMESTEP. This option is available starting with the second release of Version 971. EQ.0: No data is written EQ.1: Output incremental nodal mass EQ.2: Output percentage increase in nodal mass

VARIABLE	DESCRIPTION
THERM	Output of thermal data to d3plot. The use of this option (THERM>0) may make the database incompatible with other 3rd party software. EQ.0: (default) output temperature EQ.1: output temperature EQ.2: output temperature and flux EQ.3: output temperature, flux, and shell bottom and top surface temperature
INTOUT	Output stress/strain at all integration points for detailed element output in the file ELOUTDET. DT and BINARY of *DATABASE_ELOUT apply to ELOUTDET. See remarks 4-10 below. EQ.STRESS: when stress output is required EQ.STRAIN when strain output is required EQ.ALL when both stress and strain output are required
NODOUT	Output extrapolated stress/strain at connectivity nodes for detailed element output in the file ELOUTDET. DT and BINARY of *DATABASE_ELOUT apply to ELOUTDET. EQ.STRESS when stress output is required EQ.STRAIN when strain output is required EQ.ALL when both stress and strain output are required EQ.STRESS_GL when nodal averaged stress output along the global coordinate system is required EQ.STRAIN_GL when nodal averaged strain output along the global coordinate system is required EQ.ALL_GL for global nodal averaged stress and strain output
DTDT	Output of node point Δtemperature/Δtime data to d3plot EQ.0: (default) no output EQ.1: output ΔT/Δt

Remarks:

1. If MAXINT is set to 3 then mid-surface, inner-surface and outer-surface stresses are output at the center of the element to the LS-DYNA database. For an even number of integration points, the points closest to the center are averaged to obtain the midsurface values. If multiple integration points are used in the shell plane, the stresses at the center of the element are found by computing the average of these points. For MAXINT equal to 3 LS-DYNA assumes that the data for the user defined integration rules are ordered from bottom to top even if this is not the case. If MAXINT is not equal to 3, then the stresses at the center of the element are output in the order that they are stored for the selected integration rule. If multiple points are used in plane the stresses are first averaged.

2. Beam stresses are output to the LS-DYNA database if and only if BEAMIP is greater than zero. In this latter case the data that is output is written in the same order that the integration points are defined. The data at each integration point consists of the following five values for elastic-plastic Hughes-Liu beams: the normal stress, σ_{rr}; the transverse shear stresses, σ_{rs} and σ_{tr}; the effective plastic strain, and the axial strain which is logarithmic. For beams that are not elastic-plastic, the first history variable, if any, is output instead of the plastic strain. For the beam elements of Belytschko and his co-workers, the transverse shear stress components are not used in the formulation. No data is output for the Belytschko-Schwer resultant beam.

3. If mass scaling is active, the output of the time step size reveals little information about the calculation. If global mass scaling is used for a constant time step, the total element mass is output; however, if the mass is increased so that a minimum time step size is maintained (DT2MS is negative), the added mass is output. Also, see the control card *CONTROL_TIMESTEP.

4. Output coordinate system used. When the parameters: INTOUT or NODOUT is set to STRESS, STRAIN, or ALL, the output coordinate system the data, similar to the file ELOUT, is determined by "cmpflg", the 1st column of the 2nd card in *DATABASE_ EXTENT_BINARY.

5. Nodal output when NODOUT=STRESS, STRAIN , or ALL. Each node of the element nodal connectivity will be output.

Example 1:

ELOUTDET for an shell elem. of two, 2, through-thickness integration points and four, 4, in-plane integration points, with INTOUT=STRESS and NODOUT=STRESS

element materl

ipt stress sig-xx sig-yy sig-zz sig-xy sig0yz sig-zx yield location

1- 1

1- 10	elastic	4.4104E-02	2.5141E-01	0.0000E+00	7.7640E-08	0.0000E+00	0.0000E+00	0.0000E+00	int. point	1
1- 10	elastic	4.4104E-02	2.5141E-01	0.0000E+00	7.7640E-08	0.0000E+00	0.0000E+00	0.0000E+00	int. point	2
1- 10	elastic	4.4104E-02	2.5141E-01	0.0000E+00	7.7640E-08	0.0000E+00	0.0000E+00	0.0000E+00	int. point	3
1- 10	elastic	4.4104E-02	2.5141E-01	0.0000E+00	7.7640E-08	0.0000E+00	0.0000E+00	0.0000E+00	int. point	4
1- 10	elastic	4.4104E-02	2.5141E-01	0.0000E+00	7.7640E-08	0.0000E+00	0.0000E+00	0.0000E+00	node	21
1- 10	elastic	4.4104E-02	2.5141E-01	0.0000E+00	7.7640E-08	0.0000E+00	0.0000E+00	0.0000E+00	node	22
1- 10	elastic	4.4104E-02	2.5141E-01	0.0000E+00	7.7640E-08	0.0000E+00	0.0000E+00	0.0000E+00	node	20
1- 10	elastic	4.4104E-02	2.5141E-01	0.0000E+00	7.7640E-08	0.0000E+00	0.0000E+00	0.0000E+00	node	19
2- 10	elastic	4.4104E-02	2.5141E-01	0.0000E+00	7.7640E-08	0.0000E+00	0.0000E+00	0.0000E+00	int. point	1
2- 10	elastic	4.4104E-02	2.5141E-01	0.0000E+00	7.7640E-08	0.0000E+00	0.0000E+00	0.0000E+00	int. point	2
2- 10	elastic	4.4104E-02	2.5141E-01	0.0000E+00	7.7640E-08	0.0000E+00	0.0000E+00	0.0000E+00	int. point	3
2- 10	elastic	4.4104E-02	2.5141E-01	0.0000E+00	7.7640E-08	0.0000E+00	0.0000E+00	0.0000E+00	int. point	4
2- 10	elastic	4.4104E-02	2.5141E-01	0.0000E+00	7.7640E-08	0.0000E+00	0.0000E+00	0.0000E+00	node	21
2- 10	elastic	4.4104E-02	2.5141E-01	0.0000E+00	7.7640E-08	0.0000E+00	0.0000E+00	0.0000E+00	node	22
2- 10	elastic	4.4104E-02	2.5141E-01	0.0000E+00	7.7640E-08	0.0000E+00	0.0000E+00	0.0000E+00	node	20
2- 10	elastic	4.4104E-02	2.5141E-01	0.0000E+00	7.7640E-08	0.0000E+00	0.0000E+00	0.0000E+00	node	19

6. Nodal output when NODOUT=STRESS_GL, STRAIN_GL, or ALL_GL. Averaged nodal results are calculated by summing up all contributions from elements sharing the common node, and then dividing the total by the number of contributing elements. Averaged nodal values are always output in the global coordinate system.

Example 2.

ELOUTDET for averaged nodal strain

```
n o d a l  s t r a i n  c a l c u l a t i o n s  f o r  t i m e  s t e p     24  ( at time 9.89479E+01 )


node (global)
      strain          eps-xx         eps-yy         eps-zz         eps-xy         eps-yz         eps-zx
    1-
lower surface   2.0262E-01 -2.6058E-02 -7.5669E-02 -5.1945E-03  0.0000E+00  0.0000E+00
upper surface   2.0262E-01 -2.6058E-02 -7.5669E-02 -5.1945E-03  0.0000E+00  0.0000E+00
    2-
lower surface   1.9347E-01  2.3728E-04 -8.3019E-02 -1.4484E-02  0.0000E+00  0.0000E+00
upper surface   1.9347E-01  2.3728E-04 -8.3019E-02 -1.4484E-02  0.0000E+00  0.0000E+00
    3-
lower surface   2.0541E-01 -5.7521E-02 -6.3383E-02 -1.7668E-03  0.0000E+00  0.0000E+00
upper surface   2.0541E-01 -5.7521E-02 -6.3383E-02 -1.7668E-03  0.0000E+00  0.0000E+00
    4-.....
```

7. Available stress/strain components in eloutdet stress components includes 6 stress components (sig-xx, sig-yy, sig-zz, sig-xy, sig-yz, sig-zx), yielding status, and effective plastic strain. Strain components includes 6 strain components

8. Shell element output at integration point: stresses at all integration points can be output. The strain at the top and bottom integration layer can be output. At a connective node the extrapolated stress and strain at the top and bottom layer can be output

9. Thick shell element output includes the six stress components at each integration point. Strain at the top and bottom layer can be output. At the element node, values at the bottom layer are extrapolated to yield the values of nodes 1-4, and values at the top layer are extrapolated to yield values of nodes 5-8.

10. Solid element output at integration point: stresses and strain at all integration points can be output. For the nodal points, values at the integration points are extrapolated.

For the SSSTAT option the following card(s) apply:

Define as many cards as necessary.

(Define one part set ID for each subsystem. Use as many cards as necessary.)

Card	1	2	3	4	5	6	7	8
Variable	PSID1	PSID2	PSID3	PSID4	PSID5	PSID6	PSID7	PSID8
Type	I	I	I	I	I	I	I	I

VARIABLE	DESCRIPTION
PSIDn	Part set ID for subsystem n.:, see *SET_PART.

*DATABASE_FORMAT

Purpose: Define the output format for binary files.

Card	1	2	3	4	5	6	7	8
Variable	IFORM	IBINARY						
Type	I	I						
Default	0	0						
Remarks	1	2						

VARIABLE	DESCRIPTION
IFORM	Output format for D3PLOT and D3THDT files EQ.0: LS-DYNA database format (default), EQ.1: ANSYS database format, EQ.2: Both LS-DYNA and ANSYS database formats.
IBINARY	Word size of the binary output files (D3PLOT, D3THDT, D3DRLF and interface files for 64 bit computer such as CRAY and NEC. EQ.0: default 64 bit format, EQ.1: 32 bit IEEE format

Remarks:

1. This option is not available for every platform. Check LS-DYNA Banner upon execution of the program

2. By using this option one can reduce the size of the binary output files which are created by 64 bits computer such as CRAY and NEC.

*DATABASE_FSI

Purpose: This card may be used to output information about certain coupled Lagrangian surfaces. The Lagrangian shell/segment entity to be monitored must be included in a *CONSTRAINED_LAGRANGE_IN_SOLID (CLIS) card. *DATABASE_FSI activates the output of an ASCII file called “dbfsi”. This file contains some coupling information (force, pressure, accumulated mass flowing over some surfaces, etc.) for the coupled Lagrangian surface.

Card 1	1	2	3	4	5	6	7	8
Variable	DT							
Type	F							

Define one surface per card

Card 2,3,...	1	2	3	4	5	6	7	8
Variable	DBFSI_ID	SID	SIDTYPE	SWID	CONVID			
Type	I	I	I	I	I			

VARIABLE	DESCRIPTION
DT	Output interval
DBFSI_ID	Surface ID (for reference purposes only) or a DATABASE_FSI entity ID. It consists of a geometric entity defined by the set ID below.
SID	Set ID defining the geometrical surface(s) through/upon which some data is to be tracked and output to an ASCII file called “dbfsi”. This set ID can be a (1) PID or (2) PSID or (3) SGSID. This Lagrangian SID must be contained in a Lagrangian slave SID defined in a corresponding coupling card, *CONSTRAINED_LAGRANGE_IN_SOLID.
SIDTYPE	Set type: EQ.0: Part set, EQ.1: Part, EQ.2: Segment set.

VARIABLE	DESCRIPTION
SWID	Switch ID from a corresponding *ALE_FSI_SWITCH_MMG_ID card. If defined, the accumulative mass of the "switched" ALE multi-material group (AMMG) is written out under the "pleak" parameter in the "dbfsi" file.
CONVID	For airbag application only: Convection ID from a corresponding *LOAD_ALE_CONVECTION_ID card (which computes the heat transfer between inflator gas and the inflator canister). If defined, the temperature of the Lagrangian part having heat transfer with the gas, and its change in temperature as function of time in the "dbfsi" file.

Remarks:

1. When a Lagrangian mesh overlaps with an Eulerian or ALE mesh, the fluid-structure (or ALE-Lagrangian) interaction may be modeled via a *CONSTRAINED_LAGRANGE_IN_SOLID (CLIS) card. This database command allows for the tracking of certain coupling information related to the flow across, and the load on some selected Lagrangian surfaces defined in corresponding CLIS card.

2. The output parameters in the dbfsi ASCII file are:

   p = Averaged pressure on the surface being tracked (Pa)

   fx,fy,fz = Total force components (N) over the entity(ies) defined (acting at centroid of each surface)

   pleak = Under LS-Prepost ASCII plotting, this is labeled "POROSITY". See remark 3 below.

   Mflux = If a shell part or part set is included in the coupling, and the normal of this shell structure points away from the fluid to be coupled to, then the fluid will flow across this surface since it is not detected by this surface. For example, this may be done for an airbag vent hole. Then, the amount of accumulated mass flowing across this surface may be output via the "mflux" parameter in the "dbfsi" ASCII output file. Under LS-Prepost ASCII plotting, it is labeled "Outlet_mass_Flux". This value is only an approximation as the relative velocity between the shell and the fluid group is used to compute the mass, but not precisely at the coupling points. The relative velocity vector dots with the normal vector of the Lagrangian surface to give the "sign" of the mass variable.

   {fx-lc,fy-lc,fz-lc} = Average x|y|z leakage control force component over the surface entity. This is used for debugging only. Too high leakage control forces (relative to coupling forces) may indicate that alternate coupling approach should be considered since the main coupling force is putting out too little resistance to leakage.

   Ptemp = Lagrangian part Temperature (Activated only when the *LOAD_ALE_CONVECTION card is used).

Pdtemp = Lagrangian part Temperature increase (Activated only when the *LOAD_ALE_CONVECTION card is used).

3. **PLEAK** parameter in the "dbfsi" ASCII output file from this keyword contains the accumulated **mass** (for example, Kg) for 4 different cases:

a) When LCIDPOR is defined in the coupling card (CLIS), porous flow across a Lagrangian shell surface may be monitored and output in PLEAK.

b) Porous flow across Lagrangian shell may also be defined via a load curve in the *MAT_FABRIC card, and similar result will be tracked and output.

c) When NVENT in the CLIS card is defined (isentropic venting), the flow across the isentropic vent hole may be output in PLEAK.

d) When an *ALE_FSI_SWITCH_MMG_ID card is defined, and the SWID parameter specifies this ID to be tracked, then the amount of accumulated mass that has been switched when flowing across a monitoring surface is output.

Example:

Consider a model with a Lagrangian mesh overlaps with an Eulerian or ALE mesh. On the Lagrangian mesh, there are 3 Lagrangian surface sets over which some data is to be written out.

```
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|....8
$ INPUT:
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|....8
*DATABASE_FSI
$       dt
  2.97E-06
$ DBFSI_ID       SID     STYPE      swid    convid [STYPE: 0=PSID;1=PID;2=SGSID]
        11         1         2
        12         2         2
        13         3         1
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|....8
$ This reads:
$ DBFSI_ID 11 is defined by a SID=1: a SGSID = as specified by STYPE=2
$ DBFSI_ID 12 is defined by a SID=2: a SGSID = as specified by STYPE=2
$ DBFSI_ID 13 is defined by a SID=3: a PID    = as specified by STYPE=1
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|....8
$ An OUTPUT file called "dbfsi" looks like the following:
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|....8
     Fluid-structure interaction output
     Number of surfaces:       3

        id          p            fx            fy            fz           pleak
                  mflux         fx-lc         fy-lc         fz-lc         Ptemp
PDtmp
         time=  0.00000E+00
        11    0.0000E+00    0.0000E+00    0.0000E+00    0.0000E+00    0.0000E+00
              0.0000E+00    0.0000E+00    0.0000E+00    0.0000E+00    0.0000E+00
0.0000E+00

        12    0.0000E+00    0.0000E+00    0.0000E+00    0.0000E+00    0.0000E+00
              0.0000E+00    0.0000E+00    0.0000E+00    0.0000E+00    0.0000E+00
0.0000E+00

        13    0.0000E+00    0.0000E+00    0.0000E+00    0.0000E+00    0.0000E+00
              0.0000E+00    0.0000E+00    0.0000E+00    0.0000E+00    0.0000E+00
0.0000E+00

         time=  0.29709E-05
        11    0.0000E+00    0.0000E+00    0.0000E+00    0.0000E+00    0.0000E+00
              0.0000E+00    0.0000E+00    0.0000E+00    0.0000E+00    0.0000E+00
0.0000E+00

        12    0.0000E+00    0.0000E+00    0.0000E+00    0.0000E+00    0.0000E+00
              0.0000E+00    0.0000E+00    0.0000E+00    0.0000E+00    0.0000E+00
0.0000E+00

        13    0.0000E+00    0.0000E+00    0.0000E+00    0.0000E+00    0.0000E+00
              0.1832E-06    0.0000E+00    0.0000E+00    0.0000E+00    0.0000E+00
0.0000E+00
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|....8
```

*DATABASE_FSI_SENSOR

Purpose: This card activates the output of an ASCII file called "dbsensor". Its input defines the pressure sensors' locations which follow the positions of some Lagrangian segments during the simulation. Its ASCII output file, dbsensor, contains the spatial position of the sensor and its recorded pressure from the ALE elements containing the sensors. This card is activated when a *CONSTRAINED_LAGRANGE_IN_SOLID card is used and the Lagrangian shell elements defining the locations of the sensors must be included in the slave or structure coupling set.

Card 1 Format

Card 1	1	2	3	4	5	6	7	8
Variable	DT							
Type	F							

Card(s) 2+ Format: Define one surface per line

Card 2,3,...	1	2	3	4	5	6	7	8
Variable	DBFSI_ID	NID	SEGMID	OFFSET				
Type	I	I	I	F				

VARIABLE	DESCRIPTION
DT	Output interval
DBFSI_ID	Pressure-Sensor ID.
NID	An optional Lagrangian node ID defining an approximate pressure sensor location with respect to a Lagrangian shell element. This is not a required input.
SEGMID	A required Lagrangian shell element ID for locating the pressure sensor. If NID=0 or blank, the sensor will be automatically placed in the center of this SEGMID, accounting for the offset distance.
OFFSET	Offset distance between the pressure sensor and the Lagrangian segment surface. If it is positive, it is on the side pointed to by the segment normal vector and vice versa.

Remarks:

1. The output parameters in the "dbsensor" ASCII file are:

   ID = Sensor ID.
   x,y,z = Sensor spatial location.
   P = Sensor recorded pressure (Pa) from the ALE fluid element containing the sensor.

   For example to plot the sensor pressure in LS-Prepost, select:
   ASCII → dbsensor → LOAD → (select sensor ID) → Pressure → PLOT

Example 1:

```
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|....8
$ INPUT:
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|....8
*DATABASE_FSI_SENSOR
      0.01
$ DBFSI_ID       NID SEGMENTID    OFFSET
        10       360       355      -0.5
        20       396       388      -0.5
        30       324       332      -0.5
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|....8
$ The 1st line reads:
$ SENSOR_ID 10 is located by segment-ID=355.  Node-ID=360 precisely locate this
$ sensor (if NID=0, then the sensor is located at the segment center).  This
$ sensor is located 0.5 length unit away from the segment surface.  Negative
$ sign indicates a direction opposite to the segment normal vector.
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|....8
$ An OUTPUT file called "dbsensor" looks like the following:
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|....8
     ALE sensors output
     Number of sensors:  3

        id          x            y            z            p
         time=  0.17861E-02
        10    0.0000E+00    0.0000E+00   -0.3900E+00    0.1085E-03
        20   -0.2250E+02    0.2250E+02   -0.3900E+00    0.1085E-03
        30    0.2250E+02   -0.2250E+02   -0.3900E+00    0.1085E-03
         time=  0.20081E-02
        10    0.0000E+00    0.0000E+00   -0.3900E+00    0.1066E-03
        20   -0.2250E+02    0.2250E+02   -0.3900E+00    0.1066E-03
        30    0.2250E+02   -0.2250E+02   -0.3900E+00    0.1066E-03
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|....8
$ ID    = DBFSI_ID
$ x,y,z = Sensor location (defined based on a Lagrangian segment)
$ p     = Sensor pressure as taken from the fluid element containing the sensor.
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|....8
```

DATABASE_HISTORY_OPTION*

Available options include:

BEAM

BEAM_SET

BEAM_ID

DISCRETE

DISCRETE_ID

DISCRETE_SET

NODE

NODE

NODE_ID

NODE_LOCAL

NODE_LOCAL_ID

NODE_SET

NODE_SET_LOCAL

SEATBELT

SEATBELT_ID

SHELL

SHELL_ID

SHELL_SET

SOLID

SOLID_ID

SOLID_SET

SPH

SPH_SET

TSHELL

TSHELL_ID

TSHELL_SET

Purpose: Control which nodes or elements are output into the binary history file, D3THDT, the ASCII file NODOUT, the ASCII file ELOUT and the ASCII file SPHOUT. Define as many cards as necessary. The next "*" card terminates the input. See also *DATABASE_BINARY_*OPTION* and *DATABASE_*OPTION*.

For options BEAM, BEAM_SET, DISCRETE, DISCRETE_SET, NODE, NODE_SET, SEATBELT, SHELL, SHELL_SET, SOLID, SOLID_SET, SPH, SPH_SET, TSHELL, and TSHELL_SET

Cards 1,2,... 1 2 3 4 5 6 7 8

Variable	ID1	ID2	ID3	ID4	ID5	ID6	ID7	ID8
Type	I	I	I	I	I	I	I	I

VARIABLE	DESCRIPTION
IDn	NODE/NODE_SET or element/element set ID n. Elements may be BEAM/BEAM_SET, DISCRETE/DISCRETE_SET, SEATBELT, SHELL/SHELL_SET, SOLID/SOLID_SET, or TSHELL/ TSHELL_SET. The contents of the files are given in Table 10.1 for nodes, Table 10.2 for solid elements, Table 10.3 for shells and thick shells, and Table 10.4 for beam elements. In the binary file, D3THDT, the contents may be extended or reduced with the *DATABASE_EXTENT_BINARY definition.

For options BEAM_ID, NODE_ID, SEATBELT_ID, SHELL_ID, SOLID_ID, and TSHELL_ID

Card 1,2,... 1 2-8

Variable	ID	HEADING
Type	I	A70

VARIABLE	DESCRIPTION
ID	Node or element ID
HEADING	A description of the node or element. It is suggested that unique descriptions be used. This description is written into the D3HSP file and into the ASCII databases NODOUT and ELOUT.

For options NODE_LOCAL, NODE_LOCAL_ID, and NODE_SET_LOCAL

Cards 1,..	1	2	3	4	5	6	7	8
Variable	ID	CID	REF	HFO				
Type	I	I	I	I				

Read the second card for NODE_LOCAL_ID, otherwise, skip.

Card 1,2,...	1-7	8
Variable	HEADING	
Type	A70	

VARIABLE	DESCRIPTION
ID	NODE/NODE_SET set ID. The contents of the files are given in Table 9.1 for nodes. See the remark below concerning accelerometer nodes.
CID	Coordinate system ID for nodal output. See DEFINE_COORDINATE options.
REF	Output reference: EQ.0: Output is in the local system fixed for all time from the beginning of the calculation. EQ.1: Output is in the local system which is defined by the DEFINE_ COORDINATE_NODES. The local system can change orientation depending on the movement of the three defining nodes. The defining nodes can belong to either deformable or rigid parts. EQ.2: Output is relative to the local system which is defined by the DEFINE_COORDINATE_NODES option. The local system can change orientation depending on the movement of the three defining nodes. If dynamic relaxation is used, the reference location is reset when convergence is achieved.
HFO	Flag for high frequency output into NODOUTHF EQ.0: Nodal data written to NODOUT file only EQ.1: Nodal data also written NODOUTHF at the higher frequency
HEADING	A description of the nodal point. It is suggested that unique description be used. This description is written into the D3HSP file and into the ASCII database NODOUT.

Remarks:

1. If a node belongs to an accelerometer, see *ELEMENT_SEATBELT_ ACCELEROMETER, and if it also appears as an active node in the NODE_LOCAL or NODE_SET_LOCAL keyword, the coordinate system, CID, transformations will be skipped and the LOCAL option will have no effect.

*DATABASE_MASSOUT

Purpose: Output nodal masses into ASCII file MASSOUT.

Card	1	2	3	4	5	6	7	8
Variable	SETID	NDFLG	RBFLG					
Type	I	I	I					
Default	0	1	0					

VARIABLE	DESCRIPTION
SETID	Optional set ID. EQ.0: mass output for all nodes, LT.0: no output, GT.0: set ID identifying nodes whose mass will be output.
NDFLG	Database extent: EQ.1: output translational mass for nodes identified by SETID (default), EQ.2: output translational mass and rotary inertias for the nodes identified by the SETID.
RBFLG	Rigid body data: EQ.0: no output for rigid bodies, EQ.1: output rigid body mass and inertia.

Remarks:

1. Nodes and rigid bodies with no mass are not output. By inference, when the set ID is zero and no output shows up for a node, then the mass of that node is zero.

***DATABASE_NODAL_FORCE_GROUP**

Purpose: Define a nodal force group for output into ASCII file NODFOR and the binary file XTFILE. See also *DATABASE_*OPTION* and *DATABASE_BINARY_*OPTION*.

Card	1	2	3	4	5	6	7	8
Variable	NSID	CID						
Type	I	I						
Default	none	none						

VARIABLE	DESCRIPTION
NSID	Nodal set ID, see *SET_NODE_*OPTION*.
CID	Coordinate system ID for output of data in local system, see *DEFINE_ COORDINATE_*OPTION*.

Remarks:

1. The nodal reaction forces in the global or local (if CID is defined above) x, y, and z directions are printed into the NODFOR ascii file along with the external work which is a result of these reaction forces. The resultant force vector found by summing the reaction forces over the nodes is also written into this file. These forces can be a result of applied boundary forces such as nodal point forces and pressure boundary conditions, body forces, and contact interface forces. In the absence of body forces, interior nodes would always yield a null force resultant vector. In general this option would be used for surface nodes.

*DATABASE_PWP_FLOW

Purpose: Request output containing nett inflow of fluid at a set of nodes.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	NSET							
Type	I							
Default	0							

VARIABLE	DESCRIPTION
NSET	Node set ID

Remarks:

Any number of these cards can be used. Nett inflow or outflow arises when maintaining an applied PWP boundary condition implies addition or removal of water.

Output is written to a file named database_pwp_flow.csv, a comma-separated ascii file. Each line consists of (time, flow1, flow2…) where flow1 is the total inflow at the node set for the first DATABASE_PWP_FLOW request, flow2 is for the second, etc.

*DATABASE_PWP_OUTPUT

Purpose: Set contents of output files for pore pressure calculations.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	IVEL	IACCX	IACCY	IACCZ	NCYOUT			
Type	I	I	I	I	I			
Default	0	0	0	0	100			

VARIABLE	DESCRIPTION
IVEL	Meaning of "Velocity" in d3plot and d3thdt output files 0: Nodal velocity vector 1: Seepage velocity vector
IACCX,Y,Z	Meaning of "X/Y/Z-Acceleration" in d3plot and d3thdt output files 0: Not written 1: Total pwp head 2: Excess pwp head (this is also written as temperature) 3: Target rate of volume change 4: Actual rate of volume change 7: Hydraulic pwp head 8: Error in rate of volume change (calculated from seepage minus actual) 9: Volume at node 10: Rate of volume change calculated from seepage 14: Void volume (generated at suction limit) 17: NFIXCON (e.g. +4/-4 for nodes on suction limit)
NCYOUT	Number of cycles between outputs of calculation status to d3hsp, log, and tdc_control_output.csv files (time-dependent and steady-state analysis types)

*DATABASE_SPRING_FORWARD

Purpose: Create spring forward nodal force file. This option is to output resultant nodal force components of sheet metal at the end of the forming simulation into an ASCII file, "SPRING-FORWARD", for spring forward and die corrective simulations.

Card	1	2	3	4	5	6	7	8
Variable	IFLAG							
Type	I							

VARIABLE	DESCRIPTION
IFLAG	Output type: EQ.0: off, EQ.1: output element nodal force vector for deformable nodes.

*DATABASE_SUPERPLASTIC_FORMING

Purpose: Specify the output intervals to the superplastic forming output files. The option *LOAD_SUPERPLASTIC_FORMING must be active.

Card	1	2	3	4	5	6	7	8
Variable	DTOUT							
Type	F							

VARIABLE	DESCRIPTION
DTOUT	Output time interval for output to “pressure”, “curve1” and “curve2” files. The “pressure” file contains general information from the analysis and the files “curve1” and “curve2” contain pressure versus time from phases 1 and 2 of the analysis. The data in the pressure and curve files may be plotted using ASCII > superpl in LS-Prepost.

*DATABASE_TRACER

Purpose: Tracer particles will save a history of either a material point or a spatial point into an ASCII file, TRHIST. This history includes positions, velocities, and stress components. The option *DATABASE_TRHIST must be active. This option applies to ALE and SPH problems.

Card	1	2	3	4	5	6	7	8
Variable	TIME	TRACK	X	Y	Z			
Type	F	I	F	F	F			
Default	0.0	Lagrangian	0	0	0			

VARIABLE	DESCRIPTION
TIME	Start time for tracer particle
TRACK	Tracking option: EQ.0: particle follows material, EQ.1: particle is fixed in space.
X	Initial x-coordinate
Y	Initial y-coordinate
X	Initial z-coordinate

*DEFINE

The keyword *DEFINE provides a way of defining boxes, coordinate systems, load curves, tables, and orientation vectors for various uses. The keyword cards in this section are defined in alphabetical order:

***DEFINE_ALEBAG_BAG**

***DEFINE_ALEBAG_HOLE**

***DEFINE_ALEBAG_INFLATOR**

***DEFINE_BOX**

***DEFINE_BOX_ADAPTIVE**

***DEFINE_BOX_COARSEN**

***DEFINE_BOX_DRAWBEAD**

***DEFINE_BOX_SPH**

***DEFINE_CONNECTION_PROPERTIES_{***OPTION***}**

***DEFINE_CONSTRUCTION_STAGES**

***DEFINE_CONTACT_VOLUME**

***DEFINE_COORDINATE_NODES**

***DEFINE_COORDINATE_SYSTEM**

***DEFINE_COORDINATE_VECTOR**

***DEFINE_CURVE_{***OPTION***}**

***DEFINE_CURVE_COMPENSATION**

***DEFINE_CURVE_DRAWBEAD**

***DEFINE_CURVE_DUPLICATE**

***DEFINE_CURVE_ENTITY**

***DEFINE_CURVE_FEEDBACK**

***DEFINE_CURVE_FUNCTION**

***DEFINE_CURVE_SMOOTH**

***DEFINE_CURVE_TRIM_{***OPTION***}**

***DEFINE_DEATH_TIMES_{***OPTION***}**

***DEFINE_ELEMENT_DEATH _{***OPTION***}**

***DEFINE_FRICTION**

***DEFINE_FUNCTION**

***DEFINE_HEX_SPOTWELD_ASSEMBLY_**{*OPTION*}

***DEFINE_SD_ORIENTATION**

***DEFINE_SET_ADAPTIVE**

***DEFINE_SPOTWELD_FAILURE_RESULTANTS**

***DEFINE_SPOTWELD_RUPTURE_PARAMETER**

***DEFINE_SPOTWELD_RUPTURE_STRESS**

***DEFINE_STAGED_CONSTRUCTION_PART**

***DEFINE_TABLE**

***DEFINE_TABLE_2D**

***DEFINE_TABLE_3D**

***DEFINE_TRANSFORMATION**

***DEFINE_VECTOR**

An additional option **_TITLE** may be appended to all the ***DEFINE** keywords. If this option is used then an addition line is read for each section in 80a format which can be used to describe the defined curve, table, etc. At present LS-DYNA does make use of the title. Inclusion of titles gives greater clarity to input decks.

Examples for the *DEFINE keyword can be found at the end of this section.

*DEFINE_ALEBAG_BAG

Purpose: This card defines information about the Lagrangian airbag structure that is required as complementary definition to the *AIRBAG_ADVANCED_ALE (AAA) card. It contains information about (a) one Lagrangian shell structure representing an airbag (or part of one), (b) venting characteristics of this airbag, and (c) its main coupling control features (Remark 1). The AAA card is used to model the airbag-to-inflator-gas interaction via the ALE method (Remark 2).

Card 1 Format

Card 1	1	2	3	4	5	6	7	8
Variable	BAGID	SID	SIDTYPE	CVBAG	IBLOCK	VTCOEF	VENTSID	VENTYP
Type	I	I	I	I	I	F	I	I
Default	none	none	1	none	none	none	0	none
Remarks	1			3	4	5	6	

Card 2 Format (All coupling parameters are also defined similarly under the *CONSTRAINED_LAGRANGE_IN_SOLID card)

Card 2	1	2	3	4	5	6	7	8
Variable	NQUAD	CTYPE	PFAC	FRIC	FRCMIN	NORMTYP	ILEAK	PLEAK
Type	I	I	F	F	F	I	I	F
Default	none	none	0.1	none	none	2	0.1	none
Remarks	7		8				9	

Card 3 Format (All coupling parameters are also defined similarly under the *CONSTRAINED_LAGRANGE_IN_SOLID card)

Card 3	1	2	3	4	5	6	7	8
Variable	NORM	START	END					
Type	I	F	F					
Default	1	0.0	1.0E10					
Remarks	10							

VARIABLE	DESCRIPTION
BAGID	An ID associated with the airbag mesh definition defined to be used in a corresponding *AIRBAG_ADVANCED_ALE card (See Remark 1).
SID	A set ID defining the Lagrangian airbag structure that is to be coupled to the inflator gas in a corresponding *AIRBAG_ADVANCED_ALE card. BAGID points to SID.
SIDTYP	Set type for the SID above: EQ.0: for a part set ID (PSID) EQ.1: for a part ID (PID)
CVBAG	Flag for including the SID above in the control volume (CV) analysis phase (See Remark 3): EQ.0: Do not include the above SID in subsequent CV analysis EQ.1: Include the above SID in subsequent CV analysis
IBLOCK	Flag for considering contact blockage for venting holes and porous flow (See Remark 4): EQ.0: Do not consider contact blockage EQ.1: Consider contact blockage
VTCOEF	Flow coefficient for each vent surface area defined by VENTSID (See Remark 5).
VENTSID	Set ID defining the vent hole surface (shape, See Remark 6). This venting estimate is sometimes referred to as “isentropic venting” as the flow is estimated via isentropic flow correlations. The amount of mass that escapes out of the vent hole is simply subtracted (deleted) from the mass inside the airbag, reducing the inflating potential of the inflator gas. The vented flow AMMG cannot be visualized in LS-PrePost.

VARIABLE	DESCRIPTION
VENTYP	Set ID type of the vent surface area defined by VENTSID: EQ.0: Part set ID (PSID). EQ.1: Part ID (PID). EQ.2: Segment set ID (SGSID).
NQUAD	Number of (quadrature) coupling points distributed over each coupled Lagrangian surface segment (See Remark 7). EQ.0: NQUAD will be set by default to 4, EQ.n: An NQUAD*NQUAD coupling points distribution over each Lagrangian segment is defined, EQ.-n: NQUAD is reset to a positive value. Coupling at nodes is obsolete.
CTYPE	Fluid-Structure coupling method: EQ.4: penalty coupling for shell (with or without erosion) and solid elements (without erosion). DIREC is set to 2 (default). EQ.6: penalty coupling designed for airbag modeling which automatically controls the DIREC parameter internally. It is equivalent to setting {CTYPE=4; DIREC=1} for unfolded region; and {CTYPE=4; DIREC=2}; in folded region. For both cases: {ILEAK=2; FRCMIN=0.3}.
PFAC	Penalty factor (CTYPE 4 and 6). PFAC is a scale factor for scaling the estimated stiffness of the interacting (coupling) system. It is used to compute the coupling forces to be distributed on the slave and master parts. If positive real: Fraction of estimated critical stiffness. If negative integer, -n: Refers to load curve ID n. The curve defines the coupling pressure (y-axis) as a function of the penetration (x-axis). (See Remark 8).
FRIC	Coefficient of friction (used with DIREC 2 only).
FRCMIN	Minimum volume fraction of a coupled ALE multi-material group (AMMG), or fluid, in a multi-material ALE element to activate coupling. Default value is 0.5. Reducing FRCMIN (typically, between 0.1 and 0.3) would turn on coupling earlier to prevent leakage in hypervelocity impact cases.
NORMTYP	Penalty coupling spring (or force) direction (DIREC 1 and 2): EQ.0: normal vectors are interpolated from nodal normals (default). EQ.1: normal vectors are interpolated from segment normals. This is sometimes a little more robust for sharp Lagrangian corners, and folds.

VARIABLE	DESCRIPTION
ILEAK	Coupling leakage control flag (See Remark 9): EQ.0: none (default), EQ.1: weak, leakage control is turned off if the penetrating volfrac > FRCMIN+0.1. EQ.2: strong, with improved energy consideration. Leakage control is turned off if the penetrating volfrac > FRCMIN+0.3.
PLEAK	Leakage control penalty factor, 0 < PLEAK < 0.2. This factor influences the additional coupling force magnitude to prevent leakage. It is conceptually similar to PFAC. Most of the time, the default value (0.1) is adequate.
NORM	A flag indicating the rule for defining which side of the Lagrangian segment the fluid is supposed to be coupled to. By default (NORM=0) the fluid on the side pointed to by the Lagrangian segment normal (head-side) is coupled to. To couple to the fluid on the side not pointed to by the segment normals (tail-side), set NORM=1 (See Remark 10). "Head" and "tail" refer to the normal vector of the coupling segment. EQ.0: Couple fluid to head-side of Lagrangian segment. EQ.1: Couple fluid to tail-side of Lagrangian segment.
START	Start time for coupling.
END	End time for coupling.

Remarks:

1. This command provides supplemental information for the main *AIRBAG_ADVANCED_ALE (AAA) card. Specifically, it defines (a) a Lagrangian airbag structure that couples with the inflator gas, (b) its venting characteristics, and (c) some of the main coupling parameters. The information input for the AAA set is translated into the ALE keywords internally in LS-DYNA. This input approach does not include all the control features available in the regular ALE modeling approach, especially for couplings. Therefore highly complex fluid-structure interactions may require going back to the traditional ALE input approach. For more details on all coupling parameters, please see *CONSTRAINED_LAGRANGE_IN_SOLID card.

2. The airbag inflation process may be modeled in 2 stages. In the 1st stage the bag-gas interaction is modeled via ALE method where a variable pressure field inside the airbag may be simulated. In the 2nd stage, after the airbag has opened up sufficiently, the uniform pressure inflation method (also called control volume, or CV, approach) is used to inflate the bag. Please refer to the AAA card for more information.

3. If the Lagrangian structure is to be included in the 1st stage only (deployment using ALE method) but to be excluded from the 2nd stage analysis (uniform pressure analysis), then set CVBAG=0. One such example may be the inner bag of a bag-in-bag model. If the

defined Lagrangian structure is to be included in both computational phases, set CVBAG=1.

4. Fabric venting and porous flow behaviors for ALE and CV phases (see *CONSTRAINED_LAGRANGIAN_IN_SOLID (CLIS) and *AIRBAG_HYBRID) is defined under the *MAT_FABRIC card {FLC(t), FAC(P), FVOPT}. The FLC(t) is the orifice flow coefficient (typically is close to 1.0). FAC(P) is a relative porous gas speed curve as a function of absolute upstream pressure. For AAA application, FVOPT must be either 7 or 8. FVOPT 7 and 8 will be used for both ALE and CV phases (*AIRBAG_HYBRID). Blockage consideration for both venting and porous flows is accounted for by the IBLOCK flag. If IBLOCK=0 then in the CV phase FVOPT is set to 7, no contact blockage consideration. If IBLOCK=1 then in the CV phase FVOPT is set to 8, with contact blockage consideration. IBLOCK overwrites FVOPT.

5. VTCOEF will be used to scale the vent area for ALE venting. Upon switching to the CV phase, this coefficient will be used in place of the vent coefficient “C23” of the *AIRBAG_HYBRID card.

6. VENTSID defines the isentropic venting area definition which is used for ALE venting. Upon switching to the CV phase, the venting area will be used for venting in place of parameter “A23” of the *AIRBAG_HYBRID card.

7. See remark 2 under *CONSTRAINED_LAGRANGIAN_IN_SOLID.

8. See remark 6 under *CONSTRAINED_LAGRANGIAN_IN_SOLID

9. See remark 10 under *CONSTRAINED_LAGRANGIAN_IN_SOLID.

10. See remark 7 under *CONSTRAINED_LAGRANGIAN_IN_SOLID. Typically, if the airbag shell elements have their normal vectors pointing outward. Then to couple to the inflator gas from the inside of the bag NORM is set to 1 (the default value for NORM in this card).

*DEFINE_ALEBAG_HOLE

Purpose: This optional card defines information about a physical vent hole structure of a Lagrangian airbag. This card is only used with an associated *AIRBAG_ADVANCED_ALE (AAA) card to provide supplemental venting flow information for an airbag deployment simulation (Remark 1). This card defines (a) geometry of a vent hole, (b) information for switching the ALE multi-material group (AMMG) ID of the inflator gas when it passes through a vent hole, and (c) type of vent hole. The information input here is translated into a *ALE_FSI_SWITCH_MMG_ID card (Remark 2).

Card 1 Format

Card	1	2	3	4	5	6	7	8
Variable	HOLEID	SID	SIDTYPE	NQUAD	XOFF	NFOLD	XCLEN	
Type	I	I	I	I	F	I	F	
Default	none	none	none	none	none	none	none	
Remarks	1						3	

VARIABLE	DESCRIPTION
HOLEID	An ID associated with a vent hole surface in an airbag structure definition. It is defined to be used in a corresponding *AIRBAG_ ADVANCED_ALE card. (See Remark 1.)
SID	A set ID defining the physical geometry a vent hole in the Lagrangian airbag structure. The inflator gas is supposed to pass through this hole.
SIDTYP	Set type for the SID above: EQ.0: for a part set ID (PSID) EQ.1: for a part ID (PID)
NQUAD	The number of flow-sensor points to be distributed over each monitoring surface or segment (defined by SID). There should be enough sensor points, distributed in each ALE element, to monitor the flow across this monitoring surface (see remark 3). At least 1 or 2 sensor points are needed in each ALE element to monitor the flow through that element. An alternate method for defining flow-sensor points is by defining XCLEN parameter below.

VARIABLE	DESCRIPTION
XOFF	An offset distance away from the monitoring surface, beyond which the AMMGID switching occurs. A positive value of XOFF means the offset distance is along the normal vector direction of the monitoring segment, and vice versa. This offset distance, in general, should be about 1.5 to 2 times the ALE element widths where the vent flow is occurring (default=0.0).
NFOLD	Flag for checking folding logic (default=0=off). If NFOLD=1=on, then LS-DYNA will check if the shell elements defining the monitoring surface are in the folded region or not. If the monitoring segment is still located within a folded region, then no AMMG switching is allowed yet until it has unfolded.
XCLEN	This is an absolute distance for distributing the flow sensor points over each monitoring segment (surface). To make sure that at least 1 or 2 sensor points are present in each ALE element to track the flow of an AMMG, XCLEN may be roughly estimated as a third or one-half the width of the smallest ALE element in the mesh (see Remark 3).

Remarks:

1. A vent hole associated with an airbag structure may be defined for modeling physical venting (actual flow of material across the hole surface may be monitored, in contrast to isentropic venting where vented material is simply deleted). The primary function of this card is to define the vent hole geometry and the AMMG switching as the gas passes through it.

   This card provides supplemental information for an associated *AIRBAG_ADVANCED_ALE (AAA) card. The vent hole structure is associated with and airbag ID under AAA so that the coupling and AMMGID switching may be performed. This is equivalent to modeling "physical venting holes" via the *CONSTRAINED_LAGRANGE_IN_SOLID card. It is used only when users define fine enough ALE mesh to resolve the flow across the vent (physical venting, see remark 12 under *CONSTRAINED_LAGRANGE_IN_SOLID card).

   The information input for the AAA card and all its supplemental commands are converted into the ALE keywords internally in LS-DYNA. The corresponding ALE keywords are written out to a file called "advalebag.kw" for checking. For an in-depth understanding of the interaction modeling, it is recommended that the users understand how to set up the coupling with the traditional ALE method.

2. For more detailed information, the user may review the *ALE_FSI_SWITCH_MMG_ID card since it is the actions of this card that are being executed. The AMMGIDs of the gases involved in the switching, upstream and downstream of the hole are defined automatically.

3. When both NQUAD and XCLEN are defined, whichever gives smaller distance between sensor-points will be used. XCLEN may give better control as in the case of a null shell acting as the monitoring surface. As this null shell is stretched, NQUAD distribution of sensor-points may not be adequate, but XCLEN would be.

***DEFINE_ALEBAG_INFLATOR**

Purpose: This card defines supplemental information about an airbag inflator that is required as complementary definition to the *AIRBAG_ADVANCED_ALE (AAA) card. It defines the (a) orifices, (b) gas properties, and (c) inlet conditions (Remark 1). This command is only used with an associated *AIRBAG_ADVANCED_ALE card. This card contains basic information that is required by the ALE command *SECTION_POINT_SOURCE_MIXTURES. It is only used for modeling airbag deployment process (Remark 2) via the ALE method.

Card 1

Variable	INFLAID	unused	unused	unused	NGAS	NORIF	LCIDVEL	LCIDT
Type	I				I	I	I	I
Default	none				none	none	0	none
Remarks	1							

Repeat this card "NGAS" times, one for each species in the mixture.
See *AIRBAG_HYBRID

Card 2

Variable	LCIDMD	unused	unused	MWGAS	unused	A	B	C
Type	I			F		F	F	F
Default	none			none		none	0.0	0.0
Remarks						3	3	3

Repeat this card "NORIF" times.
See *SECTION_POINT_SOURCE_MIXTURE

Card 3	1	2	3	4	5	6	7	8
Variable	NODEID	VECID	ORIFAREA	unused	unused	unused	unused	unused
Type	I	I	F					
Default	0	0	0.0					
Remarks	4							

VARIABLE	DESCRIPTION
INFLAID	An inflator ID. It is defined to be used in a corresponding *AIRBAG_ADVANCED_ALE card. (See Remark 1).
NGAS	Number of thermally equilibrated ideal gas species making up one inflator gas mixture.
NORIF	Number of inflator inlet orifices through which the inflator gas mixture is injected into the airbag. Each orifice requires a node ID to define its location (see NODEID below).
LCIDVEL	User-estimated inflator gas mixture average velocity load curve ID. If LCIDVEL=0 or blank, LSDYNA will estimate the inlet gas velocity.
LCIDT	Inflator gas mixture thermally equilibrated stagnation temperature load curve ID. All species of the mixture are assumed to have the same average stagnation temperature.
LCIDMD	The inlet mass flow rate load curve ID of this species of the inflator gas mixture.
MWGAS	Molecular weight of this species (e.g., kg/mole).
A	Nominal constant-pressure heat capacity (per-mole unit) at STP (see Remark 3).
B	1^{st} order (linear) coefficient of temperature dependent heat capacity of the inflator gas. (see Remark 3).
C	2^{nd} order (quadratic) coefficient of temperature dependent heat capacity of the inflator gas. (see Remark 3).

VARIABLE	DESCRIPTION
NODEID	A node ID defining the location of a point source (see Remark 4).
VECID	A vector ID defining the direction of flow at each point source.
ORIFAREA	The orifice area at each point source.

Remarks:

1. This card defines the inflator inflow information for one inflator referred to by an *AIRBAG_ADVANCED_ALE (AAA) card. The basic information includes (a) gas properties, (b) orifices, and (c) inlet conditions. The gas properties are defined similarly to that of the *AIRBAG_HYBRID card. The orifices, and inlet conditions (see ***SECTION_POINT_SOURCE_MIXTURE**), consist of $\dot{m}(t), T_{stag}(t)$, and maybe an estimated inlet gas velocity curve, $\tilde{v}_{gas}(t)$, if available. The information input for the AAA card and all its associates are translated into ALE keywords internally in LS-DYNA. The ALE keywords translated by AAA are output to the "advalebag.kw" file for review. For in depth understanding of the interaction modeling, it is recommended that the users understand how to set up the coupling with the traditional ALE method. The AAA card requires 2 additional cards for complementary definitions, *DEFINE_ALEBAG_BAG and *DEFINE_ALEBAG_INFLATOR. One optional card, *DEFINE_ALEBAG_HOLE, may be used to define the vent hole of the airbag.

2. The airbag inflation process may be modeled in 2 stages. In the 1st stage the bag-gas interaction is modeled via ALE method where a variable pressure field inside the airbag may be simulated. In the 2nd stage, after the airbag has opened up sufficiently, the uniform pressure inflation method (also called control volume, or CV, approach) is used to inflate the bag.

3. The per-mass-unit, temperature-dependent, constant-pressure heat capacity is

$$C_p(T) = \frac{\left[A + B*T + C*T^2\right]}{MW} \sim \frac{J}{kg*K} \qquad B \sim J/\left(mole*K^2\right)$$

$$A = \tilde{C}_{P0} \sim J/\left(mole*K\right) \qquad C \sim J/\left(mole*K^3\right)$$

   The units shown are only for demonstration of the equation. Please see the *MAT_GAS_MIXTURE card definition.

4. In general, it is best to locate a point source near the center of an ALE element. Associated width each point source is an area and a vector indicating flow direction. Each point source should occupy 1 ALE element by itself, and there should be at least 2 empty ALE elements between any 2 point-sources. A point source should be located at least 2 or 3 elements away from the free surface of an ALE mesh or a Lagrangian surface to prevent interaction with the mesh boundary or coupling boundary (see ***SECTION_POINT_SOURCE_MIXTURE**).

*DEFINE_BOX

Purpose: Define a box-shaped volume. Two diagonally opposite corner points of a box are specified in global coordinates. The box volume is then used for various specifications, e.g., velocities, contact, etc.

Card	1	2	3	4	5	6	7	8
Variable	BOXID	XMN	XMX	YMN	YMX	ZMN	ZMX	
Type	I	F	F	F	F	F	F	
Default	0	0.0	0.0	0.0	0.0	0.0	0.0	
Remarks								

VARIABLE	DESCRIPTION
BOXID	Box ID. Define unique numbers.
XMN	Minimum x-coordinate.
XMX	Maximum x-coordinate.
YMN	Minimum y-coordinate.
YMX	Maximum y-coordinate.
ZMN	Minimum z-coordinate.
ZMX	Maximum z-coordinate.

*DEFINE_BOX_ADAPTIVE

Purpose: Define a box-shaped volume enclosing the elements where the adaptive level is to be specified. If the midpoint of the element falls within the box the adaptive level is reset. Elements falling outside of this volume use the value, MAXLVL, on the *CONTROL_ADAPTIVE control cards.

Card 1

Variable	BOXID	XMN	XMX	YMN	YMX	ZMN	ZMX	
Type	I	F	F	F	F	F	F	
Default	none	0.0	0.0	0.0	0.0	0.0	0.0	

Card 2

Variable	PID	LEVEL						
Type	I	I						
Default	0	none						

VARIABLE	DESCRIPTION
BOXID	Box ID. Define unique numbers.
XMN	Minimum x-coordinate.
XMX	Maximum x-coordinate.
YMN	Minimum y-coordinate.
YMX	Maximum y-coordinate.
ZMN	Minimum z-coordinate.
ZMX	Maximum z-coordinate.
PID	Part ID. If zero, all active elements within box are considered.
LEVEL	Maximum number of refinement levels for elements that are contained in the box. Values of 1, 2, 3, 4,... allow a maximum of 1, 4, 16, 64, ... elements, respectively, to be created for each original element.

*DEFINE_BOX_COARSEN

Purpose: Define a specific box-shaped volume indicating elements which are protected from mesh coarsening. See also *CONTROL_COARSEN.

Card	1	2	3	4	5	6	7	8
Variable	BOXID	XMN	XMX	YMN	YMX	ZMN	ZMX	IFLAG
Type	I	F	F	F	F	F	F	I
Default	none	0.0	0.0	0.0	0.0	0.0	0.0	0

VARIABLE	DESCRIPTION
BOXID	Box ID. Define unique numbers.
XMN	Minimum x-coordinate.
XMX	Maximum x-coordinate.
YMN	Minimum y-coordinate.
YMX	Maximum y-coordinate.
ZMN	Minimum z-coordinate.
ZMX	Maximum z-coordinate.
IFLAG	Flag for protecting elements inside or outside of box. EQ.0: elements inside the box cannot be coarsened EQ.1: elements outside the box cannot be coarsened

Remarks:

1. Many boxes may be defined. If an element is protected by any box then it may not be coarsened.

*DEFINE_BOX_DRAWBEAD

Purpose: Define a specific box or tube shaped volume around a draw bead. This option is useful for the draw bead contact. If box shaped, the volume will contain the draw bead nodes and elements between the bead and the outer edge of the blank. If tubular, the tube is centered around the draw bead. All elements within the tubular volume are included in the contact definition.

Card	1	2	3	4	5	6	7	8
Variable	BOXID	PID	SID	IDIR	STYPE	RADIUS	CID	
Type	I	F	F	F	I	F	I	
Default	0	0.0	0.0	0.0	4	0.0	0	
Remarks						optional	optional	

VARIABLE	DESCRIPTION
BOXID	Box ID. Define unique numbers.
PID	Part ID of blank.
SID	Set ID that defines the nodal points that lie along the draw bead. If a node set is defined, the nodes in the set must be consecutive along the draw bead. If a part or part set is defined, the set must consist of beam or truss elements. Within the part set, no ordering of the elements is assumed, but the number of nodes must equal the number of beam elements plus 1.
IDIR	Direction of tooling movement. The movement is in the global coordinate direction unless the tubular box option is active and CID is nonzero. In this latter case, the movement is in the local coordinate direction. EQ.1: tooling moves in x-direction, EQ.2: tooling moves in y-direction, EQ.3: tooling moves in z-direction.
STYPE	Set type: EQ.2: part set ID, EQ.3: part ID, EQ.4: node set ID.

VARIABLE	DESCRIPTION
RADIUS	The radius of the tube, which is centered around the draw bead. Elements of part ID, PID, that lie within the tube will be included in the contact. If the radius is not defined, a rectangular box is used instead. This option is recommended for curved draw beads and for draw beads that are not aligned with the global axes.
CID	Optional coordinate system ID. *This option is only available for the tubular drawbead. This option is available starting in the third release of version 971.*

***DEFINE_BOX_SPH**

Purpose: Define a box-shaped volume. Two diagonally opposite corner points of a box are specified in global coordinates. Particle approximations of SPH elements are computed when particles are located inside the box. The load curve describes the motion of the maximum and minimum coordinates of the box.

Card 1

Variable	BOXID	XMN	XMX	YMN	YMX	ZMN	ZMX	VID
Type	I	F	F	F	F	F	F	I
Default	none	0.0	0.0	0.0	0.0	0.0	0.0	0

Card 2

Variable	LCID	VD						
Type	I	I						
Default	0	0						

VARIABLE	DESCRIPTION
BOXID	Box ID. Define unique numbers.
XMN	Minimum x-coordinate.
XMX	Maximum x-coordinate.
YMN	Minimum y-coordinate.
YMX	Maximum y-coordinate.
ZMN	Minimum z-coordinate.
ZMX	Maximum z-coordinate.
VID	Vector ID for DOF, see *DEFINE_VECTOR.

VARIABLE	DESCRIPTION
LCID	Load curve ID to describe motion value versus time, see *DEFINE_CURVE
VD	Velocity/Displacement flag: EQ.0: velocity, EQ.1: displacement

***DEFINE_CONNECTION_PROPERTIES_**{*OPTION*}

Available options include:

<BLANK>

ADD

Purpose: Define failure related parameters for solid element spot weld failure by *MAT_SPOTWELD_DAIMLERCHRYSLER. For each connection identifier, CON_ID, a separate *DEFINE_CONNECTION_PROPERTIES section must be included. The **ADD** option allows material specific properties to be added to an existing connection ID. See remark 2.

:

Card 1

1	2	3	4	5	6	7	8
CON_ID	PROPRUL	AREAEQ		DG_TYP			
F	I	I		I			
0	0	0		0	.		

Card 2

Variable	D_SIGY	D_ETAN	D_DG_PR	D_RANK	D_SN	D_SB	D_SS
Type	F	F	F	F	F	F	F
Default	none	none	1.0e+10	none	none	none	none

Card 3

Variable	D_EXSN	D_EXSB	D_EXSS	D_LCSN	D_LCSB	D_LCSS	D_GFAD	
Type	F	F	F	I	I	I	F	
Default	1.0	1.0	1.0	0	0	0	none	

Define the following 2 cards for each shell material that will have material specific data defined for this CON_ID. The input is terminated by the next "*" keyword card.

Card 4

Variable	MID	SGIY	ETAN	DG_PR	RANK	SN	SB	SS
Type	A8	F	F	F	F	F	F	F
Default				1.0e+10				

Card 5

Variable	EXSN	EXSB	EXSS	LCSN	LCSB	LCSS	GFAD	
Type	F	F	F	I	I	I	F	
Default								

VARIABLE	DESCRIPTION
CON_ID	Connection ID, referenced on *MAT_SPOTWELD_DAIMLERCHRYSLER. Multiple sets of connection data may be used by assigning different connection IDs.
PROPRUL	The failure rule number for this connection.
AREAEQ	Area equation number for the connection area calculation. EQ.0: (default) area_true=area_modeled EQ.1: millimeter form; see Remark 4 EQ.-1: meter form; see Remark 4
DG_TYP	Damage type EQ.0: no damage function is used EQ.1: strain based damage EQ.2: failure function based damage EQ.3 or 4: fading energy based damage; see Remark 4
D_SIGY	Default yield stress for the spot weld element.
D_ETAN	Default tangent modulus for the spot weld element.
D_DG_PR	Default damage parameter for hyperbolic based damage function.

VARIABLE	DESCRIPTION
D_RANK	Default rank value.
D_SN	Default normal strength.
D_SB	Default bending strength.
D_SS	Default shear strength.
D_EXSN	Default exponent on normal stress term.
D_EXSB	Default exponent on bending stress term.
D_EXSS	Default exponent on shear stress term.
D_LCSN	Default curve ID for normal strength scale factor as a function of strain rate.
D_LCSB	Default curve ID for bending strength scale factor as a function of strain rate.
D_LCSS	Default curve ID for shear strength scale factor as a function of strain rate.
D_GFAD	Default fading energy for damage type 3 and type 4.
MID	Material ID of the shell material for which properties are defined.
SIGY	Yield stress to be used in the spot weld element calculation.
ETAN	Tangent modulus to be used in the spot weld element calculation.
DG_PR	Damage parameter for hyperbolic based damage function.
RANK	Rank value. See Remark 4.
SN	Normal strength.
SB	Bending strength.
SS	Shear strength.
EXSN	Exponent on normal stress term.
EXSB	Exponent on bending stress term.
EXSS	Exponent on shear stress term.
LCSN	Curve ID for normal strength scale factor as a function of strain rate.

VARIABLE	DESCRIPTION
LCSB	Curve ID for bending strength scale factor as a function of strain rate.
LCSS	Curve ID for shear strength scale factor as a function of strain rate.
LCSS	Fading energy for damage type 3.

Remarks:

1. This keyword is used only with *MAT_SPOTWELD_DAIMLERCHRYSLER. The data input is used in a 3 parameter failure model. Each solid spot weld element connects shell elements that may have the same or different materials. The failure model assumes that failure of the spot weld depends on the properties of the welded materials, so this keyword allows shell material specific data to be input for the connection. The default data will be used for any spot weld connected to a shell material that does not have material specific data defined, so it is not necessary to define material specific data for all welded shell materials.

2. To simplify data input, the ADD keyword option allows material specific data to be added to an existing *DEFINE_CONNECTION_PROPERTIES table. To use the ADD option, omit cards 2 and 3, and input only CON_ID on card 1. Then use cards 4 and 5 to input material specific data. For each unique CON_ID, control parameters and default values must be input in one set of *DEFINE_CONNECTION_PROPERTIES data. The same CON_ID may be used for any number of sets of material specific data input with the ADD option.

3. The three parameter failure function is

$$f = \left(\frac{\sigma_n}{\sigma_n^F}\right)^{m_n} + \left(\frac{\sigma_b}{\sigma_b^F}\right)^{m_b} + \left(\frac{\tau}{\tau^F}\right)^{m_\tau} - 1$$

where the three strength terms are SN, SB, and SS, and the three exponents are EXSN, EXSB, and EXSS. The strengths may be a function of strain rate by using the load curves, LCSN, LCSB, and LCSS. The peak stresses in the numerators are calculated from force resultants and simple beam theory.

$$\sigma_n = \frac{N_{rr}}{A} \quad \sigma_b = \frac{\sqrt{M_{rs}^2 + M_{rt}^2}}{Z} \quad \tau = \frac{M_{rr}}{2Z} + \frac{\sqrt{N_{rs}^2 + N_{rt}^2}}{A}$$

where the area is the cross section area of the weld element and Z is given by:

$$Z = \pi \frac{d^3}{32}$$

where *d* is the equivalent diameter of the solid spot weld element assuming a circular cross section.

4. There are three control parameters that define how the table data will be used for the connection, PROPRUL, AREA_EQ, and DG_TYP. PROPRUL determines how the parameters will be used. Because each weld connects two shell surfaces, one weld can have two sets of failure data as well as two values for ETAN and SIGY. At present, a single rule is implemented and the data with the lower RANK will be used.

The second control parameter is AREA_EQ which specifies a rule for calculating a true weld cross section area, A_{true} to be used in the failure function in place of the modeled solid element area, *A*. For AREA_EQ=1, A_{true} is calculated by

$$A_{true} = \frac{\pi}{4}\left(5\sqrt{t_{min\_shell}}\right)^2$$

where t_{min_shell} is the thickness of the welded shell surface that has the smaller thickness. For AREA_EQ=−1, A_{true} is calculated by

$$A_{true} = \frac{\pi}{4}\left(\frac{5}{1000}\sqrt{1000 * t_{min\_shell}}\right)^2$$

The equation for AREA_EQ=1 is valid only for a length unit of millimeters, and AREA_EQ=−1 is valid only for a length unit of meters.

The third control parameter, DG_TYP, chooses from two available damage types. For DG_TYP=0, damage is turned off and the weld fails immediately when $f \ge 0$. For DG_TYP>0, damage is initiated when $f \ge 0$ and complete failure occurs when $\omega \ge 1$. For DG_TYP=1, damage growth is a function of plastic strain:

$$\omega = \frac{\varepsilon^p_{eff} - \varepsilon^p_{failure}}{\varepsilon^p_{rupture} - \varepsilon^p_{failure}} \quad if \quad \varepsilon^p_{failure} \le \varepsilon^p_{eff} \le \varepsilon^p_{rupture}$$

where ε^p_{eff} is the effective plastic strain in the weld material. When the value of the failure function first exceeds zero, the plastic strain at failure $\varepsilon^p_{failure}$ is set to the current plastic strain, and the rupture strain is offset from the plastic strain at failure by

$$\varepsilon^p_{rupture} = \varepsilon^p_{failure} + \text{RS} - \text{EFAIL}$$

where RS and EFAIL are the rupture strain and plastic strain at failure which are input on the *MAT_SPOTWELD_DAIMLERCHRYSLER card. If failure occurs when the plastic strain is zero, the weld material yield stress is reduced to the current effective stress such that damage can progress.

For DG_TYP=2, damage is a function of the failure function, *f*:

$$\omega = \frac{f}{f_{rupture}} \quad \text{if} \quad f \geq 0$$

where $f_{rupture}$ is the value of the failure function at rupture which is defined by

$$f_{rupture} = \text{RS-EFAIL}$$

and RS and EFAIL are input on the *MAT_SPOTWELD_DAIMLERCHRYSLER card.

Because the DG_TYP=1 damage function is scaled by plastic strain, it will monotonically increase in time. The DG_TYP=2 damage function is forced to be a monotonically increasing function in time by using the maximum of the current value and the maximum previous value. For both DG_TYP=1 and DG_TYP=2, the stress scale factor is then calculated by

$$\hat{\sigma} = \frac{DG\_PR(1-\omega)}{\omega\left(\frac{1}{2}+\sqrt{\frac{1}{4}+DG\_PR}\right)+DG\_PR}\sigma$$

This equation becomes nearly linear at the default value of DG_PR which is 1.0e+10.

For DG_TYP=3, damage is a function of total strain:

$$\omega = \frac{\Delta\varepsilon_n}{\Delta\varepsilon_{fading}}$$

where $\Delta\varepsilon_n$ is the accumulated total strain increment between moment of damage initiation (failure) and current time step t_n

$$\Delta\varepsilon_n = \Delta\varepsilon_{n-1} + \Delta t_n \sqrt{2/3\dot{\boldsymbol{\varepsilon}}_n : \dot{\boldsymbol{\varepsilon}}_n} \quad , \qquad \Delta\varepsilon\big|_{t_{failure}} = 0$$

and $\Delta\varepsilon_{fading}$ is the total strain increment for fading (reduction of stresses to zero)

$$\Delta\varepsilon_{fading} = \frac{2\,GFAD}{\sigma_{failure}}$$

where GFAD is the fading energy from input and $\sigma_{failure}$ is the effective stress at failure. The stress scale factor is then calculated by a linear equation

$$\hat{\sigma} = (1-\omega)\sigma$$

where σ is the Cauchy stress tensor at failure and ω is the actual damage value. Problems can occur, if the loading direction changes after the onset of failure, since during the damage process, the components of the stress tensor are kept constant and hence represent the stress state at failure.

Therefore DG_TYP=4 should be used describing the damage behavior of the spotweld in a more realistic way. For DG_TYP=4, damage is a function of the internal work done by the spotweld after failure, i. e.,

$$\hat{\sigma} = (1-\omega)\,\sigma^{ep} \;, \quad \omega = \frac{G_{used}}{2GFAD} \;, \quad G_{used} = G_{used}^{n-1} + \det F_{ij}\,\sigma_{ij}^{ep}\,\Delta\varepsilon_{ij} \;.$$

Therein, F_{ij} is the deformation gradient. σ^{ep} is a scaled Cauchy stress tensor based on the undamaged Cauchy stress tensor σ^{wd} and scaled in such a way that the same internal work is done in the current time step as in the time step before (equipotential):

$$\sigma^{ep} = \alpha\sigma^{wd} \;, \quad \alpha = \frac{\sigma_{ij}^{n-1,ep}\,\Delta\varepsilon_{ij}}{\sigma_{ij}^{wd}\,\Delta\varepsilon_{ij}} \;.$$

*DEFINE_CONSTRUCTION_STAGES

Purpose: Define times and durations of construction stages.

Note: This keyword card will be available starting in release 3 of version 971.

Card	1	2	3	4	5	6	7	8
Variable	ISTAGE	ATS	ATE	ATR	RTS	RTE		
Type	I	F	F	F	F	F		
Default	none	0.0	0.0	none	=ATS	=ATE		

VARIABLE	DESCRIPTION
ISTAGE	Stage ID
ATS	Analysis time at start of stage
ATE	Analysis time at end of stage
ATR	Analysis time duration of ramp
RTS	Real time at start of stage
RTE	Real time at end of stage

Remarks:

See also *CONTROL_CONSTRUCTION_STAGES and *DEFINE_STAGED_CONSTRUCTION_PART.

The first stage should start at time zero. There must be no gaps between stages, i.e. ATS for each stage must be the same as ATE for the previous stage.

The ramp time allows gravity loading and part stiffening/removal to be applied gradually during the first time period ATR of the construction stage.

The analysis always runs in “analysis time” – typically measured in seconds. The “real time” is used only as a number to appear on output plots and graphs, and is completely arbitrary. A dynain file is written at the end of each stage.

*DEFINE_CONTACT_VOLUME

Purpose: Define a rectangular, a cylindrical, or a spherical volume in a local coordinate system. Nodes and segments which belong to specified part ID's and lie inside of the defined volume are used in the treatment of contact.

Card 1	1	2	3	4	5	6	7	8
Variable	CVID	CID	TYPE	XC	YC	ZC		
Type	I	I	I	F	F	F		
Default	0	0	0	0.	0.	0.		

For type=0, rectangular prism

Card 2	1	2	3	4	5	6	7	8
Variable	XMN	XMX	YMN	YMX	ZMN	ZMX		
Type	F	F	F	F	F	F		
Default	0.0	0.0	0.0	0.0	0.0	0.0		

For type=1, cylindrical volume

Card 2	1	2	3	4	5	6	7	8
Variable	LENGTH	RINNER	ROUTER	D_ANGC				
Type	F	F	F	F				
Default	0.0	0.0	0.0	0.0				

For type=3, spherical volume

Card 2	1	2	3	4	5	6	7	8
Variable	RINNER	ROUTER	D_ANGS					
Type	F	F	F					
Default	0.0	0.0	0.0					

VARIABLE	DESCRIPTION
CVID	Contact volume ID
CID	Coordinate system ID. Required for rectangular and cylindrical volumes
TYPE	Volume type. Set to 0 for rectangular, 1 for cylindrical, and 2 for spherical.
XC	x-coordinate which defines the origin of coordinate system or the center of the sphere for type=3 referenced to the global coordinate system.
YC	y-coordinate which defines the origin of coordinate system or the center of the sphere for type=3 referenced to the global coordinate system.
ZC	z-coordinate which defines the origin of coordinate system or the center of the sphere for type=3 referenced to the global coordinate system.
XMN	Minimum x-coordinate in local coordinate system.
XMX	Maximum x-coordinate in local coordinate system.
YMN	Minimum y-coordinate in local coordinate system.
YMX	Maximum y-coordinate in local coordinate system.
ZMN	Minimum z-coordinate in local coordinate system.
ZMX	Maximum z-coordinate in local coordinate system.
LENGTH	Length of cylinder originating at (XC,YC,ZC) and revolving around the local x-axis.
RINNER	Inner radius of cylinder or sphere.
ROUTER	Outer radius of cylinder or sphere.

VARIABLE	DESCRIPTION
D_ANGC	If the included angle between the axis of the cylinder and the normal vector to the contact segment is *less* than this angle, the segment is deleted.
D_ANGS	If the included angle between a line draw from the center of the sphere to the centroid of the segment, and the normal vector to the contact segment is *greater* than this angle, the segment is deleted.

*DEFINE_COORDINATE_NODES

Purpose: Define a local coordinate system with three node numbers. The local cartesian coordinate system is defined in the following steps. If the primary direction is along the x-axis, then the z-axis is computed from the cross product of x and $\bar{y}$, (see Figure 11.2), $z = x \times \bar{y}$, then the y-axis is computed via $y = z \times x$. A similar procedure applies if the local axis is along the y or z axes. The DIR option below applies to the third release of 971 and later versions.

Card	1	2	3	4	5	6	7	8
Variable	CID	N1	N2	N3	FLAG	DIR		
Type	I	I	I	I	I	A		
Default	0	0	0	0	0	X		

VARIABLE	DESCRIPTION
CID	Coordinate system ID. A unique number has to be defined.
N1	ID of node located at local origin.
N2	ID of node located along local x-axis if DIR=X, the y-axis if DIR=Y, and along the z axis if DIR=Z.
N3	ID of node located in local x-y plane if DIR=X, the local y-z plane if DIR=Y, and the local z-x plane if DIR=Z.
FLAG	Set to unity, 1, if the local system is to be updated each time step for the BOUNDARY_SPC nodal constraints and ELEMENT_BEAM type 6, the discrete beam element. Generally, this option when used with nodal SPC's is *not recommended* since it can cause excursions in the energy balance because the constraint forces at the node may go through a displacement if the node is partially constrained
DIR	Axis defined by node N2 moving from the origin node N1. The default direction is the x-axis.

Remarks:

1. The nodes N1, N2, and N3 must be separated by a reasonable distance and not colinear to avoid numerical inaccuracies.

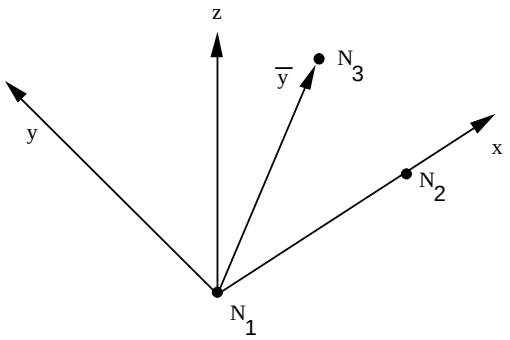


Figure 11.1. Definition of local coordinate system using three nodes when the node N2 lies along the x-axis.

*DEFINE_COORDINATE_SYSTEM

Purpose: Define a local coordinate system with three points. The same procedure as described in Figure 11.1, see *DEFINE_COORDINATE_NODES, is used. The coordinates of the nodes are given instead. N_1 is defined by (X_0,Y_0,Z_0), N_2 is defined by (X_L,Y_L,Z_L), and N_3 by (X_P,Y_P,Z_P).

Card 1 of 2 - Required.

Card 1	1	2	3	4	5	6	7	8
Variable	CID	XO	YO	ZO	XL	YL	ZL	
Type	I	F	F	F	F	F	F	
Default	0	0.0	0.0	0.0	0.0	0.0	0.0	
Remarks								

Card 2 of 2 - Required.

Card 2	1	2	3	4	5	6	7	8
Variable	XP	YP	ZP					
Type	F	F	F					
Default	0.0	0.0	0.0					
Remarks								

VARIABLE	DESCRIPTION
CID	Coordinate system ID. A unique number has to be defined.
XO	X-coordinate of origin
YO	Y-coordinate of origin
ZO	Z-coordinate of origin
XL	X-coordinate of point on local x-axis

VARIABLE	DESCRIPTION
YL	Y-coordinate of point on local x-axis
ZL	Z-coordinate of point on local x-axis
XP	X-coordinate of point in local x-y plane
YP	Y-coordinate of point in local x-y plane
ZP	Z-coordinate of point in local x-y plane

Remarks:

1. The coordinates of the points must be separated by a reasonable distance and not colinear to avoid numerical inaccuracies.

*DEFINE_COORDINATE_VECTOR

Purpose: Define a local coordinate system with two vectors, see Figure 11.2. The vector cross product, $z = x \times xy$, determines the z-axis. The y-axis is then given by $y = z \times x$. If this coordinate system is assigned to a nodal point, then at each time step during the calculation, the coordinate system is incrementally rotated using the angular velocity of the nodal point to which it is assigned.

Card	1	2	3	4	5	6	7	8
Variable	CID	XX	YX	ZX	XV	YV	ZV	NID
Type	I	F	F	F	F	F	F	I
Default	0	0.0	0.0	0.0	0.0	0.0	0.0	0.

VARIABLE	DESCRIPTION
CID	Coordinate system ID. A unique number has to be defined.
XX	X-coordinate on local x-axis. Origin lies at (0,0,0).
YX	Y-coordinate on local x-axis
ZX	Z-coordinate on local x-axis
XV	X-coordinate of local x-y vector
YV	Y-coordinate of local x-y vector
ZV	Z-coordinate of local x-y vector
NID	Optional nodal point ID. The coordinate system rotates with the rotation of this node. If the node is not defined, the coordinate system is stationary.

Remarks:

1. These vectors should be separated by a reasonable included angle to avoid numerical inaccuracies.

2. Ideally, this nodal point should be attached to a rigid body or a structural part where the nodal point angular velocities are meaningful. It should be noted that angular velocities of nodes may not be meaningful if the nodal point is attached only to solid elements and

even to shell elements where the drilling degree of freedom may be singular, which is likely in flat geometries.

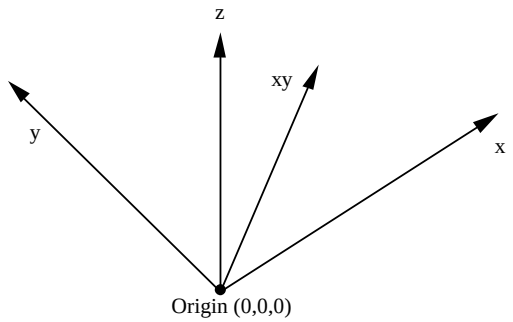


Figure 11.2. Definition of the coordinate system with two vectors.

***DEFINE_CURVE**_{*OPTION*}

Purpose: Define a curve [for example, load (ordinate value) versus time (abscissa value)], often referred to as a load curve.

Curves are discretized internally with equal intervals along the abscissa for fast evaluation in constitutive models. Discretized curves are not used for evaluating loading conditions. Also, see remark 1 below. To improve the accuracy of the discretized curves in later releases of version 970 changes were made in the discretization process. These changes had the unexpected effect of changing the results generated with validated models such as barriers and occupants. Consequently, *OPTION* was added to make available the old discretization if needed for the validated models.

Available options include:

<OPTION>

3858

5434a

which correspond to the first releases of version 970 and the 2005 release, respectively.

Since input errors and wrong results are sometimes related to load curve usage, a "*Load curve usage*" table is printed in the D3HSP file after all the input is read. This table should be checked to insure that each curve ID is referenced by the option for which the curve is intended.

Card 1

Variable	LCID	SIDR	SFA	SFO	OFFA	OFFO	DATTYP	
Type	I	I	F	F	F	F	I	
Default	none	0	1.	1.	0.	0.	0	

Card 2, 3, 4, etc. Put one pair of points per card (2E20.0). Input is terminated when a "*" card is found. (Use only two points for applying loads if the implicit arc-length method is active.)

Card 2…

Variable	A1	O1		
Type	F	F		
Default	0.0	0.0		

VARIABLE	DESCRIPTION
LCID	Load curve ID. Tables (see *DEFINE_TABLE) and load curves may not share common ID's. LS-DYNA allows load curve ID's and table ID's to be used interchangeably. A unique number has to be defined.
SIDR	Stress initialization by dynamic relaxation: EQ.0: load curve used in transient analysis only or for other applications, EQ.1: load curve used in stress initialization but not transient analysis, EQ.2: load curve applies to both initialization and transient analysis.
SFA	Scale factor for abscissa value. This is useful for simple modifications. EQ.0.0: default set to 1.0.
SFO	Scale factor for ordinate value (function). This is useful for simple modifications. EQ.0.0: default set to 1.0.
OFFA	Offset for abscissa values, see explanation below.
OFFO	Offset for ordinate values (function), see explanation below.
DATTYP	Data type. Usually 0, set to 1 only for general xy data. This affects how offsets are applied. General xy data curves refer to curves whose abscissa values do not increase monotonically. Generally, DATTYP=0 for time dependent curves, force versus displacement curves, and stress strain curves.
A1, A2,...	Abscissa values. Only pairs have to be defined, see remarks below.
O1, O2,...	Ordinate (function) values. Only pairs have to be defined, see remarks below.

Remarks:

1. **Warning**: In the definition of Load Curves used in the constitutive models, reasonable spacing of the points should always be observed, i.e., never set a single point off to a value approaching infinity. LS-DYNA uses internally discretized curves to improve efficiency in the constitutive models. Also, since the constitutive models extrapolate the curves, it is important to ensure that extrapolation does not lead to physically meaningless values, such as a negative flow stress.

2. The load curve values are scaled after the offsets are applied, i.e.:

$$Abscissa\ value = SFA \cdot (Defined\ value + OFFA)$$
$$Ordinate\ value = SFO \cdot (Defined\ value + OFFO)$$

3. Positive offsets for the load curves (DATTYP=0) are intended for time versus function curves since two additional points are generated automatically at time zero and at time .999*OFFA with the function values set to zero. If DATTYP=1, then the offsets do not create these additional points. Negative offsets for the abscissa simply shifts the abscissa values without creating additional points.

4. Load curves are not extrapolated by LS-DYNA for applied loads such as pressures, concentrated forces, displacement boundary conditions, etc. Function values are set to zero if the time, etc., goes off scale. Therefore, extreme care must be observed when defining load curves. In the constitutive models, extrapolation is employed if the values on the abscissa go off scale.

5. The load curve offsets and scale factors are ignored during restarts if the curve is redefined. See *CHANGE_CURVE_DEFINITION in the restart section.

*DEFINE_CURVE_COMPENSATION

Purpose: To define a curve for local compensation. All elements inside or outside of a curve can be compensated locally with a transitional region. This keyword must be used with *INTERFACE_COMPENSATION_NEW.

Card 1

Variable	INOUT							
Type	I							
Default	none							

Card 2

Variable	X	Y	Z					
Type	F	F	F					
Default	none	none	none					

VARIABLE	DESCRIPTION
INOUT	EQ.1: elements inside the curve will be compensated EQ.2: elements outside the curve will be compensated
X,Y,Z	Coordinates of curve points

*DEFINE_CURVE_DRAWBEAD

Purpose: To facilitate the definition of drawbead.

Card	1	2	3	4	5	6	7	8
Variable	CID	TCTYPE	VID	PID	BLKID	PERCT		
Type	I	I	I	I	I			
Default								

VARIABLE	DESCRIPTION
CID	Curve ID
TYPE	Bead date type EQ.1: x,y,z data EQ.2: IGES data
VID	Vector ID, See DEFINE_VECTOR. This vector is used to project the bead to the rigid part (PID)
PID	Part ID to attach the drawbead
BLKID	Blank ID
PERCT	Percentage of restraining force (the ratio of restraining force over Lock force). The value should be between 0 and 100.

***DEFINE_CURVE_DUPLICATE**

Purpose: Define a curve by optionally scaling and offsetting the abscissa and ordinates of another curve defined by the *DEFINE_CURVE keyword.

Card 1	1	2	3	4	5	6	7	8
Variable	LCID	RLCID	SFA	SFO	OFFA	OFFO		
Type	I	I	F	F	F	F		
Default	none	none	1.	1.	0.	0.		

VARIABLE	DESCRIPTION
LCID	Load curve ID. Tables (see *DEFINE_TABLE) and load curve ID's must be unique.
RLCID	Reference load curve ID.
SFA	Scale factor for abscissa value of curve ID, RLCID. This value scales the SFA value defined for RLCID. EQ.0.0: default set to 1.0.
SFO	Scale factor for ordinate value (function) of curve ID, RLCID. This value scales the SFO value defined for RLCID. EQ.0.0: default set to 1.0.
OFFA	Offset for abscissa values. This value is added to the OFFA value defined for RLCID.
OFFO	Offset for ordinate values (function). This value is added to the OFFO value defined for RLCID.

*DEFINE_CURVE_ENTITY

Purpose: Define a curve of straight line segments and circular arcs that defines an axisymmetric surface. This curve can only be used with the keyword, *CONTACT_ENTITY for the load curve entity, GEOTYP=11. This option is in the third release.

Card	1	2	3	4	5	6	7	8
Variable	LCID	SFA	SFO	SFR	OFFA	OFFO	OFFR	
Type	I	F	F	F	F	F	F	
Default	none	1.	1.	1.	0.	0.	0.	

Card 2, 3, 4, etc. Put one pair of points per card (3E20.0,I20). Input is terminated when a "*" card is found.

Card	1	2	3	4	5	6	7	8
Variable	Ai		Oi		Ri		IFLAG	
Type	F		F		F		I	
Default	0.0		0.0		optional		Required if |R1|>0	

VARIABLE	DESCRIPTION
LCID	Load curve ID. Tables (see *DEFINE_TABLE) and load curves may not share common ID's. LS-DYNA allows load curve ID's and table ID's to be used interchangeably. A unique number has to be defined.
SFA	Scale factor for axis value. This is useful for simple modifications. EQ.0.0: default set to 1.0.
SFO	Scale factor for radius values. This is useful for simple modifications. EQ.0.0: default set to 1.0.
SFR	Scale factor for circular radius. This is useful for simple modifications. EQ.0.0: default set to 1.0.
OFFA	Offset for axis values, see explanation below.

VARIABLE	DESCRIPTION
OFFO	Offset for radius values, see explanation below.
OFFR	Offset for circular radius, see explanation below.
Ai	Z-axis coordinates along the axis of rotation.
Oi	Radial coordinates from the axis of rotation
Ri	Radius of arc between points (Ai,Oi) and (Ai+1,Oi+1). If zero, a straight line segment is assumed.
IFLAG	Defined if |Ri|>0. Set to 1 if center of arc is inside axisymmetric surface and to -1 if the center is outside the axisymmetric surface.

Remarks:

1. The load curve values are scaled after the offsets are applied, i.e.:

$$\begin{aligned} \textit{Axis value} &= SFA \cdot (\textit{Defined value} + OFFA) \\ \textit{Radius value} &= SFO \cdot (\textit{Defined value} + OFFO) \\ \textit{Circular radius} &= SFR \cdot (\textit{Defined value} + OFFR) \end{aligned}$$

*DEFINE_CURVE_FEEDBACK

Purpose: Define information that is used as the solution evolves to scale the ordinate values of the specified load curve ID. One application for this capability is in sheet metal stamping.

Card 1

Variable	LCID	PID	BOXID	FLDID				
Type	I	I	I	I				
Default	none	none	0	none				

Card 2

Variable	FSL	TSL	SFF	SFT	BIAS			
Type	F	F	F	F	F			
Default	none	none	1.0	1.0	0.0			

VARIABLE	DESCRIPTION
LCID	ID number for load curve to be scaled.
PID	Active part ID for load curve control
BOXID	Box ID. Elements of specified part ID contained in box are checked. If the box ID is set to zero the all elements of the active part are checked.
FLDID	Load curve ID which defines the flow limit diagram as shown in Figure 11.3.
FSL	If the strain ratio, $\varepsilon_{major_{workpiece}} / \varepsilon_{major_{fld}}$ exceeds *FSL*, the scale factor for flow, *SF* , is active.
TSL	Thickness strain limit. If the through thickness strain is exceeded the scale factor for thickening, *ST* , is active.
SFF	Scale factor for the flow limit diagram, *SF* (Default=1.0).

VARIABLE	DESCRIPTION
SFT	Scale factor for thickening, ST (Default=1.0).
BIAS	Bias for combined flow and thickening, S, $-1 \le S \le 1$.

Remarks:

The scale factor for the load curve ordinate value is updated as:

$$S_{load\,curve}^{n+1} = S_{load curve}^{n} \cdot S_{final}$$

where S_{final} is equal to SF if the strain ratio is exceeded or to ST if the thickness strain limit is exceeded. The bias value determines the final scale factor, S_{final}, in the event that the thickness and flow limit diagram criteria both satisfied. In this case the scale factor for the load curve is given by:

$$S_{final} = \frac{1}{2}(1-S)\cdot SF + \frac{1}{2}(1+S)ST$$

Generally, SF is slightly less than unity and ST is slightly greater than unity so that $S_{loadcurve}$ changes insignificantly from time step to time step.

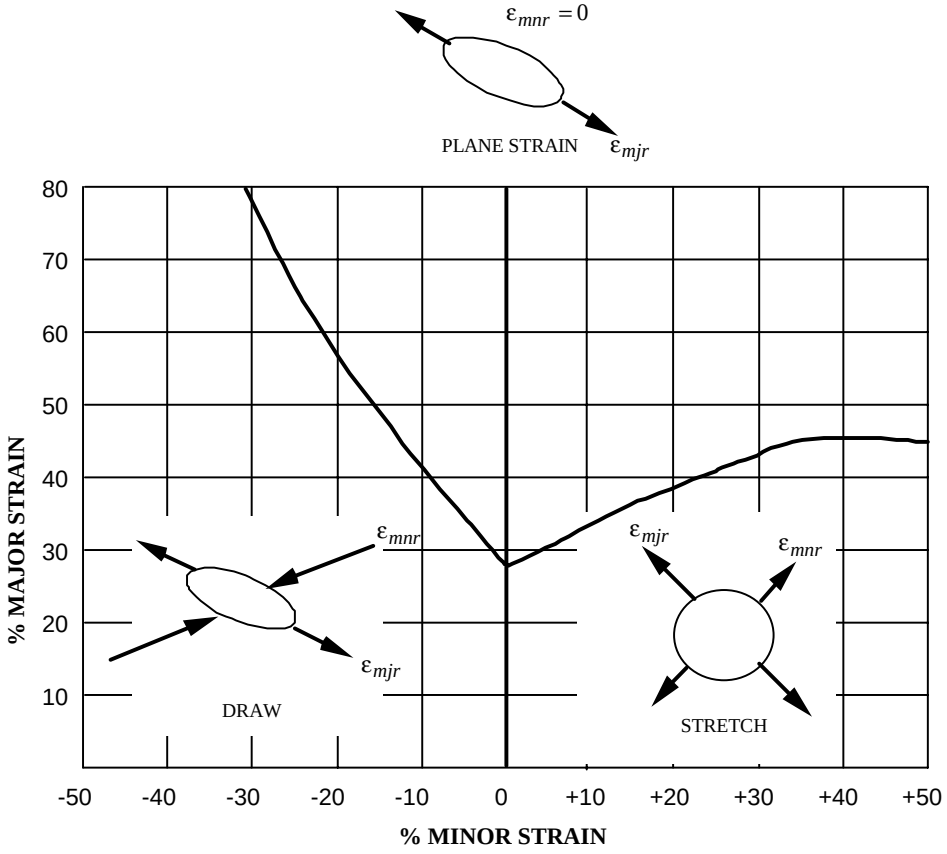


Figure 11.3. Flow limit diagram

*DEFINE_CURVE_FUNCTION

Purpose: Define a curve [for example, load (ordinate value) versus time (abscissa value)] where the ordinate is given by a function expression. The function can reference other curve definition, kinematical quantities, forces, interpolating polynomials, intrinsic functions, and combinations thereof. Please note that many functions require the definition of a local coordinate system (see Remark 1 below).

Card	1	2	3	4	5	6	7	8
Variable	LCID	SIDR						
Type	I	I						
Default	none	0						

Card 2, 3, 4, etc. (not to exceed 10). These cards are combined to form a single line of input. The next "*" terminates the input.

Card

Variable	FUNCTION
Type	C
Remarks	1

VARIABLE	DESCRIPTION
LCID	Load curve ID. Tables (see *DEFINE_TABLE) and load curves may not share common ID's. LS-DYNA allows load curve ID's and table ID's to be used interchangeably. A unique number has to be defined.
SIDR	Stress initialization by dynamic relaxation: EQ.0: load curve used in transient analysis only or for other applications, EQ.1: load curve used in stress initialization but not transient analysis, EQ.2: load curve applies to both initialization and transient analysis.

VARIABLE	DESCRIPTION
FUNCTION	Arithmetic expression involving a combination of the following possibilities.

Constants and Variables

FUNCTION NAME	DESCRIPTION
TIME	Current simulation time
PI	Proportionality constant relating the circumference of a circle to its diameter
DTOR	Degrees to radians conversion factor (PI/180.)
RTOD	Radians to degrees conversion factor (180./PI)

Intrinsic Functions

FUNCTION NAME	DESCRIPTION
ABS(a)	Absolute value of (a)
AINT(a)	Nearest integer whose magnitude is not larger than (a)
ANINT(a)	Nearest whole number to (a)
MOD(a1,a2)	Remainder when a1 is divided by a2
SIGN(a1,a2)	Transfer sign of a2 to magnitude of a1
MAX(a1,a2)	Maximum of a1 and a2
MIN(a1,a2)	Minimum of a1 and a2
SQRT(a)	Square root of (a)
EXP(a)	e raised to the power of (a)
LOG(a)	Natural logarithm of (a)
LOG10(a)	Log base 10 of (a)
SIN(a)	Sine of (a)
COS(a)	Cosine of (a)
TAN(a)	Tangent of (a)

ASIN(a)	Arc sine of (a)
ACOS(a)	Arc cosine of (a)
ATAN(a)	Arc tangent of (a)
ATAN2(a1,a2)	Arc tangent of (a1/a2)
SINH(a)	Hyperbolic sine of (a)
COSH(a)	Hyperbolic cosine of (a)
TANH(a)	Hyperbolic tangent of (a)

Load Curves

FUNCTION NAME	DESCRIPTION
LCn	Ordinate value of curve n defined elsewhere (see *DEFINE_CURVE)

Coordinate Functions

FUNCTION NAME	DESCRIPTION
CX(n1)	Value of x-coordinate for node n1.
CY(n1)	Value of y-coordinate for node n1.
CZ(n1)	Value of z-coordinate for node n1.

Displacement Functions

FUNCTION NAME	DESCRIPTION
DM(n1[,n2])	Magnitude of translational displacement of node n1 relative to node n2. Node n2 is optional and if omitted the displacement is computed relative to ground.
DX(n1[,n2,n3])	x-translational displacement of node n1 relative to node n2 expressed in the local coordinate system of node n3. If node n2 is omitted it defaults to ground. If node n3 is not specified the displacement is reported in the global coordinate system.
DY(n1[,n2,n3])	y-translational displacement of node n1 relative to node n2 expressed in the local coordinate system of node n3. If node n2 is omitted it defaults to ground. If node n3 is not specified the displacement is reported in the global coordinate system.
DZ(n1[,n2,n3])	z-translational displacement of node n1 relative to node n2 expressed in the local coordinate system of node n3. If node n2 is omitted it defaults to ground. If node n3 is not specified the displacement is reported in the global coordinate system.
AX(n1[,n2])	Rotation displacement of node n1 about the local x-axis of node n2. If n2 is not specified then it defaults to ground. In computing this value it is assumed the rotation about the other two axes (y-, z-axes) of node n2 is zero.
AY(n1[,n2])	Rotation displacement of node n1 about the local y-axis of node n2. If n2 is not specified then it defaults to ground. In computing this value it is assumed the rotation about the other two axes (x-, z-axes) of node n2 is zero.
AZ(n1[,n2])	Rotation displacement of node n1 about the local z-axis of node n2. If n2 is not specified then it defaults to ground. In computing this value it is assumed the rotation about the other two axes (x-, y-axes) of node n2 is zero.
PSI(n1[,n2])	First angle in the body2:313 Euler rotation sequence which orients node n1 in the frame of node n2. If n2 is omitted it defaults to ground.
THETA(n1[,n2])	Second angle in the body2:313 Euler rotation sequence which orients node n1 in the frame of node n2. If n2 is omitted it defaults to ground.
PHI(n1[,n2])	Third angle in the body2:313 Euler rotation sequence which orients node n1 in the frame of node n2. If n2 is omitted it defaults to ground.
YAW(n1[,n2])	First angle in the body3:321 yaw-pitch-roll rotation sequence which orients node n1 in the frame of node n2. If n2 is omitted it defaults to ground.

PITCH(n1[,n2])	Second angle in the body3:321 yaw-pitch-roll rotation sequence which orients node n1 in the frame of node n2. If n2 is omitted it defaults to ground.
ROLL(n1[,n2])	Third angle in the body3:321 yaw-pitch-roll rotation sequence which orients node n1 in the frame of node n2. If n2 is omitted it defaults to ground.

Velocity Functions

FUNCTION NAME	DESCRIPTION
VM(n1[,n2])	Magnitude of translational velocity of node n1 relative to node n2. Node n2 is optional and if omitted the velocity is computed relative to ground.
VR(n1[,n2])	Relative radial translational velocity of node n1 relative to node. If node n2 is omitted it defaults to ground.
VX(n1[,n2,n3])	x-component of the difference between the translational velocity vectors of node n1 and node n2 in the local coordinate system of node n3. If node n2 is omitted if defaults to ground. Node n3 is optional and if not specified the global coordinate system is used.
VY(n1[,n2,n3])	y-component of the difference between the translational velocity vectors of node n1 and node n2 in the local coordinate system of node n3. If node n2 is omitted if defaults to ground. Node n3 is optional and if not specified the global coordinate system is used.
VZ(n1[,n2,n3])	z-component of the difference between the translational velocity vectors of node n1 and node n2 in the local coordinate system of node n3. If node n2 is omitted if defaults to ground. Node n3 is optional and if not specified the global coordinate system is used.
WM(n1[,n2])	Magnitude of angular velocity of node n1 relative to node n2. Node n2 is optional and if omitted the angular velocity is computed relative to ground.
WX(n1[,n2,n3])	x-component of the difference between the angular velocity vectors of node n1 and node n2 in the local coordinate system of node n3. If node n2 is omitted if defaults to ground. Node n3 is optional and if not specified the global coordinate system is used.
WY(n1[,n2,n3])	x-component of the difference between the angular velocity vectors of node n1 and node n2 in the local coordinate system of node n3. If node n2 is omitted if defaults to ground. Node n3 is optional and if not specified the global coordinate system is used.
WZ(n1[,n2,n3])	z-component of the difference between the angular velocity vectors of node n1 and node n2 in the local coordinate system of node n3. If node

n2 is omitted if defaults to ground. Node n3 is optional and if not specified the global coordinate system is used.

Acceleration Functions

FUNCTION NAME	DESCRIPTION
ACCM(n1[,n2])	Magnitude of translational acceleration of node n1 relative to node n2. Node n2 is optional and if omitted the acceleration is computed relative to ground.
ACCX(n1[,n2,n3])	x-component of the difference between the translational acceleration vectors of node n1 and node n2 in the local coordinate system of node n3. If node n2 is omitted if defaults to ground. Node n3 is optional and if not specified the global coordinate system is used.
ACCY(n1[,n2,n3])	y-component of the difference between the translational acceleration vectors of node n1 and node n2 in the local coordinate system of node n3. If node n2 is omitted if defaults to ground. Node n3 is optional and if not specified the global coordinate system is used.
ACCZ(n1[,n2,n3])	z-component of the difference between the translational acceleration vectors of node n1 and node n2 in the local coordinate system of node n3. If node n2 is omitted if defaults to ground. Node n3 is optional and if not specified the global coordinate system is used.
WDTM(n1[,n2])	Magnitude of angular acceleration of node n1 relative to node n2. Node n2 is optional and if omitted the angular acceleration is computed relative to ground.
WDTX(n1[,n2,n3])	x-component of the difference between the angular acceleration vectors of node n1 and node n2 in the local coordinate system of node n3. If node n2 is omitted if defaults to ground. Node n3 is optional and if not specified the global coordinate system is used.
WDTY(n1[,n2,n3])	y-component of the difference between the angular acceleration vectors of node n1 and node n2 in the local coordinate system of node n3. If node n2 is omitted if defaults to ground. Node n3 is optional and if not specified the global coordinate system is used.
WDTZ(n1[,n2,n3])	z-component of the difference between the angular acceleration vectors of node n1 and node n2 in the local coordinate system of node n3. If node n2 is omitted if defaults to ground. Node n3 is optional and if not specified the global coordinate system is used.

Generic Force Functions

FUNCTION NAME	DESCRIPTION
FM(n1[,n2])	Magnitude of net translational force acting between node n1 and n2. Node n2 is optional and if omitted the force that acting only on n1.
FX(n1[,n2,n3])	x-component of the net translational force acting at node n1 as computed in the optional local system of node n3. If n2 is specified then all forces acting between n1 and n2 are included in the force computation.
FY(n1[,n2,n3])	y-component of the of the net translational force acting at node n1 as computed in the optional local system of node n3. If n2 is specified then all forces acting between n1 and n2 are included in the force computation.
FZ(n1[,n2,n3])	z-component of the of the net translational force acting at node n1 as computed in the optional local system of node n3. If n2 is specified then all forces acting between n1 and n2 are included in the force computation.
TM(n1[,n2])	Magnitude of net torque acting between node n1 and n2. Node n2 is optional and if omitted the torque that acting only on n1.
TX(n1[,n2,n3])	x-component of the net torque acting at node n1 as computed in the optional local system of node n3. If n2 is specified then all forces acting between n1 and n2 are included in the torque computation.
TY(n1[,n2,n3])	y-component of the net torque acting at node n1 as computed in the optional local system of node n3. If n2 is specified then all forces acting between n1 and n2 are included in the torque computation.
TZ(n1[,n2,n3])	z-component of the net torque acting at node n1 as computed in the optional local system of node n3. If n2 is specified then all forces acting between n1 and n2 are included in the torque computation.

Contact Force Functions

FUNCTION NAME	DESCRIPTION
RCFORC(id,ims,comp,local)	Returns the component *comp* (see description below) of contact interface *id* (see *CONTACT_..._ID) as calculated in the local coordinate system *local* (see *DEFINE_COORDINATE_...). If *local* equals zero then forces are reported in the global coordinate system. Forces are reported for the slave side when *ims*=1 or master side when *ims*=2.

Following are the admissible values of *comp* and their corresponding force component.
1: x force component
2: y force component
3: z force component
4: resultant force

Element Specific Functions

FUNCTION NAME	DESCRIPTION
BEAM(id,jflag,comp,rm)	Returns the force component *comp* (see description below) of beam *id* as calculated in the local coordinate system *rm*. Forces are reported in the global coordinate system if *rm* is zero. If *rm* equals -1 the beam's r, s, and t force/moment is returned. If *jflag* is set to zero then the force/torque acting on n1 end of the beam is returned, else if *jflag* is set to unity the force/torque on the n2 end of the beam is returned. See *ELEMENT_BEAM for the nodal connectivity rule defining n1 and n2. Admissible values of *comp* are 1-8 and correspond to the following components. 1: force magnitude 2: x force (axial r-force, *rm*=-1) 3: y force (s-shear force, *rm*=-1) 4: z force (t-shear force, *rm*=-1) 5: torque magnitude 6: x torque (torsion, *rm*=-1) 7: y torque (s-moment, *rm*=-1) 8: z torque (t-moment, *rm*=-1)
ELHIST(eid,etype,comp,ipt,local)	Returns the elemental quantity *comp* (see description below) of element e*id* as calculated in the local coordinate system *local*. Quantities are reported in the global coordinate system if *local* is zero. The parameter *ipt* specifies whether the quantity is for particular integration point or maximum, minimum, or averaging is applied across the integration points. The following element classes, specified with *etype*, are supported. 0: solid 2: thin shell Following are admissible values of *comp* and the corresponding elemental quantity. 1-6: x, y, z, xy, yz, and zx stress, respectively 7: effective plastic strain 8: hydrostatic pressure 10: effective stress

45-50: lower surface x,y,z,xy,yz,zx strain
51-56: upper surface x,y,z,xy,yz,zx strain
57-62: middle surface x,y,z,xy,yz,zx strain

Integration point options, specified with *ipt*, follow.
ge.1: quantity is reported for integration point number *ipt*
eq.-1: maximum of all integration points (default)
eq.-2: average of all integration points
eq.-3: minimum of all integration points
eq.-4: lower surface integration point
eq.-5: upper surface integration point
eq.-6: middle surface integration point

The local coordinate option *local* currently defaults to the global coordinate system for solid elements and other coordinate system options are unavailable. In the case of thin shell elements the quantity is reported only in the element local coordinate system.
eq.1: global coordinate system (solid elements)
eq.2: element coordinate system (thin shell elements)

JOINT(id,jflag,comp,rm) Returns the force component *comp* (see description below) due to rigid body joint *id* as calculated in the local coordinate system *rm*. If *jflag* is set to zero then the force/torque acting on n1 end of the joint is returned. The force/torque on the n2 end of the joint is returned if jflag is set to 1. See *CONSTRAINED_JOINT for the rule defining n1 and n2.

General Functions

FUNCTION NAME	DESCRIPTION

CHEBY(x,x0,a0,…,a30) Evaluates a Chebyshev polynomial at the user specified value x. The parameters x0, a0, a1, …, a30 are used to define the constants for the polynomial defined by:

$$C(x) = \sum a_j T_j(x - x_0)$$

where the functions T_j is defined recursively as

$$T_j(x - x_0) = 2 \cdot (x - x_0) \cdot T_{j-1}(x - x_0) - T_{j-2} \cdot (x - x_0)$$

where

$$T_0(x - x_0) = 1$$
$$T_1(x - x_0) = x - x_0$$

FORCOS(x,x0,ω,a0,…,a30) Evaluates a Fourier cosine series at the user specified value x. The parameters x0, a0, a1, …, a30 are used to define the constants for the series defined by:

$$F(x) = \sum a_j T_j(x - x_0)$$

where

$$T_j(x - x_0) = \cos\left[j \cdot \omega \cdot (x - x_0)\right]$$

FORSIN(x,x0,ω,a0,…,a30) Evaluates a Fourier sine series at the user specified value x. The parameters x0, a0, a1, …, a30 are used to define the constants for the series defined by:

$$F(x) = \sum a_j T_j(x - x_0)$$

where

$$T_j(x - x_0) = \sin\left[j \cdot \omega \cdot (x - x_0)\right]$$

POLYL(x,x0,a0,…,a30) Evaluates a standard polynomial at the user specified value x. The parameters x0, a0, a1, …, a30 are used to define the constants for the polynomial defined by:

$$P(x) = a_0 + a_1(x - x_0) + a_2(x - x_0)^2 + \ldots a_n(x - x_0)^n$$

SHF(x,x0,a,ω,phi,b) Evaluates a Fourier sine series at the user specified value x. The parameters x0, a0, a1, …, a30 are used to define the constants for the series defined by:

$$SHF = a \cdot \sin\left[\omega \cdot (x - x_0) - phi\right] + b$$

STEP(x,x0,h0,x1,h1) Approximates the Heavyside function with a cubic polynomial using the equation:

$$STEP = \left\langle \begin{array}{l} h_0 \\ h_0 + (h_1 - h_0) \cdot \left[(x - x_0)/(x_1 - x_0)\right]^2 \cdot \left\{3 - 2 \cdot \left[(x - x_0)/(x_1 - x_0)\right]\right\}; x_0 < x < x_1 \\ h_1 \end{array} \middle| \begin{array}{l} x \le x_0 \\ x_0 < x < x_1 \\ x \ge x_1 \end{array} \right\rangle$$

Remarks:

1. A local coordinate system must be attached to nodes if they are referenced by functions involving rotational motion, for example, angular displacement or angular velocity. The local coordinate system is attached to the node using *DEFINE_COORDINATE_NODES where FLAG must be set equal to unity. Similarly, a local coordinate system must also be attached to node n3 if n3 is referenced in

functions: DX, DY, DZ, VX, VY, VZ, WX, WY, WZ, ACCX, ACCY, ACCZ, WDTX, WDTY, WDTZ, FX, FY, FZ, TX, TY, or TZ.

*DEFINE_CURVE_SMOOTH

Purpose: Define a smoothly varying curve using few parameters. This shape is useful for velocity control of tools in metal forming applications.

Card	1	2	3	4	5	6	7	8
Variable	LCID	SIDR	DIST	TSTART	TEND	TRISE	V0	
Type	I	I	F	F	F	F	F	
Default	none	none	none	none	none	none	none	

VARIABLE	DESCRIPTION
LCID	Load curve ID, must be unique.
SIDR	Stress initialization by dynamic relaxation: EQ.0: load curve used in transient analysis only or for other applications, EQ.1: load curve used in stress initialization but not transient analysis, EQ.2: load curve applies to both initialization and transient analysis.
DIST	Total distance tool will travel (area under curve).
TSTART	Time curve starts to rise
TEND	Time curve returns to zero. If TEND is nonzero, VMAX will be computed automatically to satisfy required travel distance DIST. Input either TEND or VMAX.
TRISE	Rise time
VMAX	Maximum velocity (maximum value of curve). If VMAX is nonzero, TEND will be computed automatically to satisfy required travel distance DIST. Input either TEND or VMAX.

Remarks:

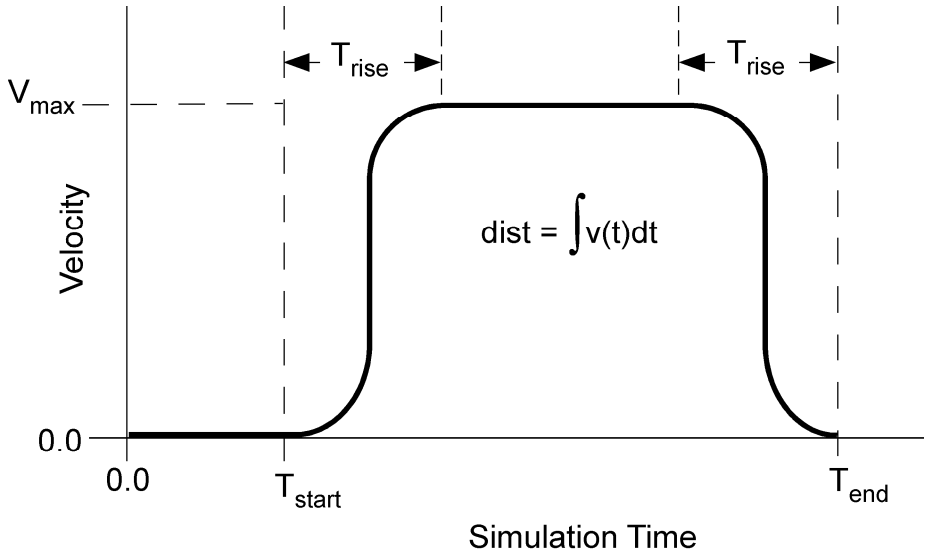


Figure 11.4. Smooth curve created automatically using *DEFINE_CURVE_SMOOTH. This shape is commonly used to control velocity of tools in metal forming applications as shown in the above graph, but can be used for other applications in place of any standard load curve.

***DEFINE_CURVE_TRIM_** {*OPTION*}

Available options include:

<BLANK>

3D

NEW

Purpose: Define a curve for trimming. Also, see *INTERFACE_SPRINGBACK.When option 3D is used, the trimming is processed based on the element normal rather than the vector. The option _NEW is used to activate a new searching algorithm, which enables a much faster trimming operation. For big models, the computation efficiency of the _NEW option is significant compared to the old method. In addition, like _3D requires user to pick a seed node, and the input is the same as *DEFINE_CURVE_TRIM_3D.

Card 1 1 2 3 4 5 6 7 8

Variable	TCID	TCTYPE	TFLG	TDIR	TCTOL	TOLN/IGB	NSEED	
Type	I	I	I	I	F	F	I	
Default	none	none	none	none	0.25	2.0	NONE	
Remarks	1,2,3			Fig. 11.5	4			

Card 2, 3, 4, etc. defined if and only if TCTYPE=1. Put one pair of points per card (2E20.0) Input is terminated when a "*" card is found.

Card 2… 1 2 3 4 5 6 7 8

Variable	CX	CY		
Type	F	F		
Default	0.0	0.0		
Type	C			

Defined if and only if TCTYPE=2.

Card 2	1	2	3	4	5	6	7	8
Variable	FILENAME							
Type	C							

VARIABLE	DESCRIPTION
TCID	ID number for trim curve.
TCTYPE	Trim curve type: EQ.1: digitized curve provided, EQ.2: IGES trim curve.
TFLG	Element removal option: EQ. -1: remove material outside curve, EQ. 1: remove material inside curve.
TDIR	ID of vector (*DEFINE_VECTOR) giving direction of projection for trim curve (see Figure 11.5). EQ. 0: default vector (0,0,1) is used. Curve is defined in global XY plane, and projected onto mesh in global Z-direction to define trim line.
TCTOL	Tolerance limiting size of small elements created during trimming (see Figure 11.6). LT.0: "simple" trimming, producing jagged edge mesh
TOLN	The maximum gap between the trimming curve and the mesh. If the gap is bigger than this value, this section in the curve will not be used. Used only when options 3D or _NEW is chosen. If option 3D is not used, then IGB.EQ.0: trimming curve is defined in local coordinate system IGB.EQ.1: trimming curve is defined in global coordinate system
NSEED	Any node within the mesh that remains after trimming. Used only when options 3D or _NEW is chosen.
CX	x-coordinate of trim curve. Define if and only if TCTYPE=1.
CY	y-coordinate of trim curve. Define if and only if TCTYPE=1.
FILENAME	Name of IGES database containing trim curve(s). Define if and only if TCTYPE=2.

Remarks:

1. This command in combination with *ELEMENT_TRIM trims the requested parts before the job starts.

2. If the command *ELEMENT_TRIM does not exist the parts are trimmed after the job is terminated.

3. Pre-trimming (*ELEMENT_TRIM + *DEFINE_CURVE_TRIM) can handle adaptive mesh and post-trimming. The keyword *DEFINE_CURVE_TRIM by itself cannot deal with an adaptive mesh. See the detailed procedure outlined in the Remarks in the Section *INTERFACE_SPRINGBACK.

4. The trimming tolerance TCTOL limits the size of the smallest element created during trimming. A value of 0.0 places no limit on element size. A value of 0.5 restricts new elements to be at least half of the size of the parent element. A value of 1.0 allows no new elements to be generated, only repositioning of existing nodes to lie on the trim curve. A negative tolerance value activates "simple" trimming, where entire elements are removed, leaving a jagged edge.

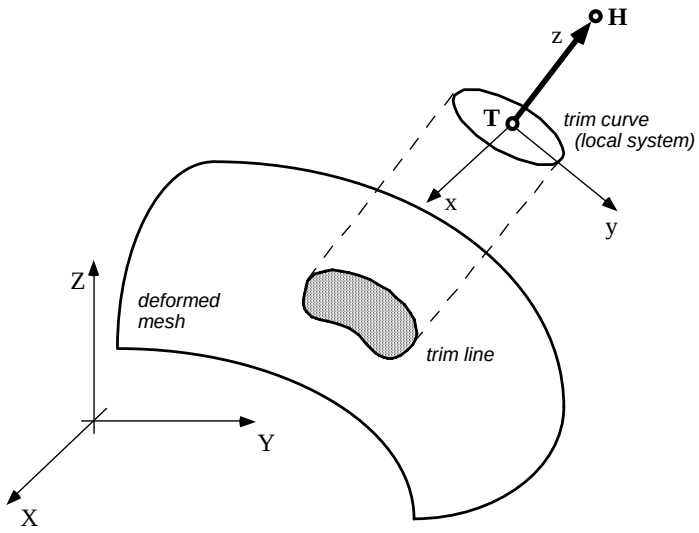
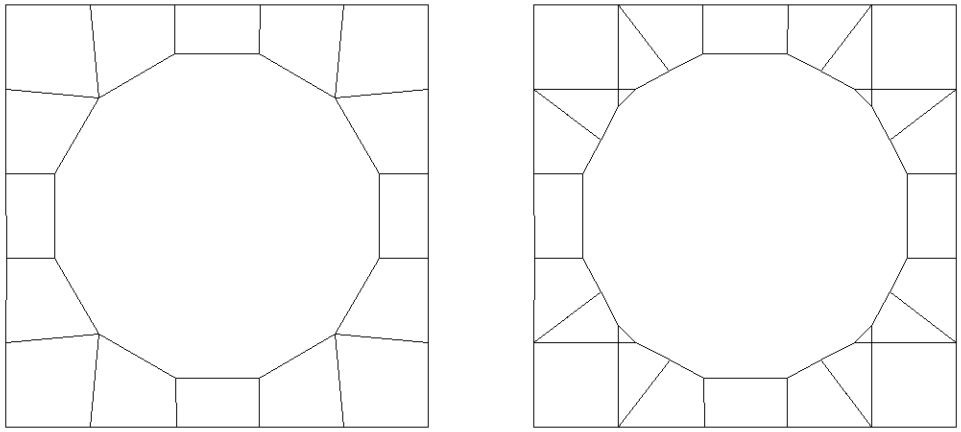


Figure 11.5. Trimming Orientation Vector. The tail (**T**) and head (**H**) points define a local coordinate system (x,y,z). The global coordinate system is named (X,Y,Z). The local x-direction is constructed in the Xz plane. If X and z nearly coincide ($|X \cdot z| > 0.95$), then the local x-direction is instead constructed in the Yz plane. Trim curve data is input in the x-y plane, and projected in the z-direction onto the deformed mesh to obtain the trim line.



tol = 0.25 (default) *tol = 0.01*

Figure 11.6 **Trimming Tolerance.** The tolerance limits the size of the small elements generated during trimming. The default tolerance (left) produces large elements. Using a tolerance of 0.01 (right) allows smaller elements, and more detail in the trim line.

DEFINE_DEATH_TIMES_OPTION*

Available options include:

NODES

SET

RIGID

Purpose: To dynamically define the death times for *BOUNDARY_PRESCRIBED_MOTION based on the locations of nodes and rigid bodies. Once a node or rigid body moves past a plane or a geometric entity, the death time is set to the current time. The input in this section continues until the next '*' card is detected.

Card 1

Variable	GEO	N1	N2	N3				
Type	I	I	I	I				
Default		0	0	0				

Card 2

Variable	X_T	Y_T	Z_T	X_H	Y_H	Z_H	R	FLAG
Type	F	F	F	F	F	F	F	
Default								1

Cards 3, …, The next "*" card terminates the friction definition.

Card 3…

Variable	NSID1	NSID2	NSID3	NSID4	NSID5	NSID6	NSID7	NSID8
Type	I	I	I	I	I	I	I	I
Default								

VARIABLE	DESCRIPTION
GEO	Geometric entity type. =1 plane, =2 infinite cylinder, =3 sphere
N1	Node defining the origin of the geometric entity (optional).
N2	Node defining the tail of the orientation vector (optional).
N3	Node defining the head of the orientation vector (optional).
X_T	X coordinate of the origin of the geometric entity and the tail of the orientation vector.
Y_T	Y coordinate of the origin of the geometric entity and the tail of the orientation vector.
Z_T	Z coordinate of the origin of the geometric entity and the tail of the orientation vector.
X_H	X coordinate of the head of the orientation vector.
Y_H	Y coordinate of the head of the orientation vector.
Z_H	Z coordinate of the head of the orientation vector.
R	Radius of cylinder or sphere.
FLAG	+1 for killing motion when the node is outside of the geometric entity or on the positive side of the plane as defined by the normal direction, or -1 for the inside.
NSIDi	i-th node, node set, or rigid body

Remarks:

1. Either N1 or X_T, Y_T, and Z_T should be specified, but not both.

2. Either N2 and N3 or X_H, Y_H, and Z_H should be specified, but not both. If N2 and N3. Specifying N2 and N3 is equivalent of setting the head of the vector equal to the tail of the vector (X_T, Y_T, and Z_T) plus the vector from N2 to N3.

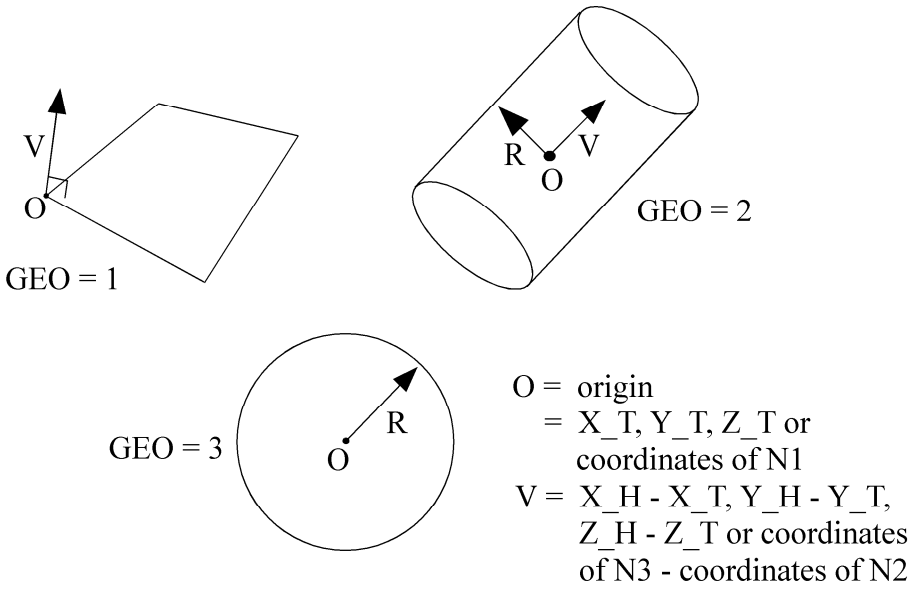

V
O
GEO = 1
R
V
O
GEO = 2
GEO = 3
R
O
O = origin
= X_T, Y_T, Z_T or
coordinates of N1
V = X_H - X_T, Y_H - Y_T,
Z_H - Z_T or coordinates
of N3 - coordinates of N2

Figure 11.7.

DEFINE_ELEMENT_DEATH_*OPTION

Available options include:

SOLID

SOLID_SET

BEAM

BEAM_SET

SHELL

SHELL_SET

THICK_SHELL

THICK_SHELL_SET

Purpose: To define a discrete time or box to delete an element or element set during the simulation.

Card 1 1 2 3 4 5 6 7 8

Variable	EID/SID	TIME	BOXID	INOUT				
Type	I	I	I	I				
Default		0	0	0				

VARIABLE	DESCRIPTION
EID/SID	Element ID or element set ID.
TIME	Deletion time for elimination of the element or element set. If BOXID is nonzero, a TIME value of zero is reset to 1.0E+16.
BOXID	Element inside or outside of defined box are deleted depending on the value of INOUT.
INOUT	Location of deleted element: EQ.0: Elements inside box are deleted EQ.1: Element outside of box are deleted

***DEFINE_FILTER**

Purpose: Define a general purpose filter, currently used by this option:

SENSOR_SWITCH,

The input in this section consists of two cards:

Card 1

1	2	3	4	5	6	7	8
Variable	ID	Title					
Type	I	A70					

Card 2…

1	2	3	4	5	6	7	8
Type	Type	Data1	Data2	Data3	Data4	Data5	Data6	Data7
Type	A10							

VARIABLE	DESCRIPTION
ID	Identification number.
Title	Title for this filter.
Type	One of the 3 currently defined filter types: DISCRETE, CONTINUOUS, or CHAIN
Data1-7	Filter type specific data, which determines what the filter does.

NOTES:There are currently 3 types of filters available. The data and behavior of each is as follows:

DISCRETE: The discrete filter operates on a fixed number of values of the input data. The first data field is an A10 character field, which gives the type of operation the filter performs: MIN, MAX, and AVG are the available options. The second data field is an I10 field, giving the number of input values over which the minimum, maximum, or average is computed.

CONTINUOUS: Similar to the DISCRETE filter, except that it operates over a fixed time interval. The first data field is exactly the same as for the DISCRETE option. The second data field is an F10 field, indicating the duration of the filter. For example, if AVG is given, and the duration is set to 0.1, a running timestep weighted average is computed over the last 0.1 time of the simulation.

CHAIN: Here, data fields 1-7 are all I10 fields, and give the IDs of a list of other filters (including other CHAIN filters, if desired), each of which will be applied in order. So the raw data is fed to the filter indicated by Data1. The output of that is fed to the next filter, and so on, with up to 7 filters in the chain. List only as many filters as you need.

***DEFINE_FRICTION**

Purpose: Define friction coefficients between parts for use in the contact options:

SINGLE_SURFACE,
AUTOMATIC_GENERAL,
AUTOMATIC_SINGLE_SURFACE,
AUTOMATIC_NODES_TO_SURFACE,
AUTOMATIC_SURFACE_TO_SUFACE,
AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE,
ERODING_SINGLE_SURFACE.

The input in this section continues until then next "*" card is encountered. Default friction values are used for any part ID pair that is not defined. Only one table can defined. The table is used if FS=-2.0 on the second card of the *CONTACT input definition. If FS=-2.0, this table will override the coefficients defined in *PART_CONTACT, which is activated by setting FS=-1.0.

Card 1

Variable	ID	FS_D	FD_D	DC_D	VC_D			
Type	I	F	F	F	F			
Default	0	0.0	0.0	0.0	0.0			

Cards 2, 3, …, The next "*" card terminates the friction definition.

Card 2…

Variable	PID_I	PID_J	FS_IJ	FD_IJ	DC_IJ	VC_IJ		
Type	I	I	F	F	F	F		
Default			0.0	0.0	0.0	0.0		

VARIABLE	DESCRIPTION
ID	Identification number. Only one table is allowed.
FS_D	Default value of the static coefficient of friction. The frictional coefficient is assumed to be dependent on the relative velocity v_{rel} of the surfaces in contact $\mu_c = FD + (FS - FD)e^{-DC\cdot\lvert v_{rel}\rvert}$. Default values are used when part pair are undefined.
FD_D	Default value of the dynamic coefficient of friction. The frictional coefficient is assumed to be dependent on the relative velocity v_{rel} of the surfaces in contact $\mu_c = FD + (FS - FD)e^{-DC\cdot\lvert v_{rel}\rvert}$. Default values are used when part pair are undefined.
DC_D	Default value of the exponential decay coefficient. The frictional coefficient is assumed to be dependent on the relative velocity v_{rel} of the surfaces in contact $\mu_c = FD + (FS - FD)e^{-DC\cdot\lvert v_{rel}\rvert}$. Default values are used when part pair are undefined.
VC_D	Default value of the coefficient for viscous friction. This is necessary to limit the friction force to a maximum. A limiting force is computed $F_{lim} = VC \cdot A_{cont}$. A_{cont} being the area of the segment contacted by the node in contact. The suggested value for *VC* is to use the yield stress in shear $VC = \frac{\sigma_o}{\sqrt{3}}$ where σ_o is the yield stress of the contacted material. Default values are used when part pair are undefined.
PID_I	Part ID I.
PID_J	Part ID J.
FS_IJ	Static coefficient of friction between parts I and J.
FD_IJ	Dynamic coefficient of friction between parts I and J.
DC_IJ	Exponential decay coefficient between parts I and J.
VC_IJ	Viscous friction between parts I and J.

***DEFINE_FUNCTION**

Purpose: Define a function that can be referenced by a limited number of keyword options.

The following card is read if and only if the ID option is specified.

Optional 1 2-8

Variable	FID	HEADING
Type	I	A70

Card 2, 3, 4, etc. (not to exceed 10). These cards are combined to form a single line of input. The next "*" terminates the input.

Cards 2,3,..

Variable	FUNCTION
Type	C

VARIABLE	DESCRIPTION
FID	Function ID. Functions, tables (see *DEFINE_TABLE), and load curves may not share common ID's. A unique number has to be defined.
HEADING	An optional descriptive heading.
FUNCTION	Arithmetic expression involving a combination of independent variables and other functions, i.e., f(a,b,c)=a*2+b*c+sqrt(a*c) where a, b, and c are the independent variables. The function name, f(a,b,c), must be unique since other functions can then use and reference this function. For example, g(a,b,c,d)=f(a,b,c)**2+d. In this example, two *DEFINE_ FUNCTION definitions are needed to define functions f and g.

*DEFINE_GROUND_MOTION

Purpose: Define an earthquake ground motion history using ground motion records provided as load curves, for use in conjunction with the *BOUNDARY_SPECIFIED_GROUND_MOTION card for dynamic earthquake analysis including nonlinear soil-structure interaction.

Card 1	1	2	3	4	5	6	7	8
Variable	GMID	ALCID	VLCID					
Type	I	I	I					
Default	none	none	0					

VARIABLE	DESCRIPTION
GMID	Ground motion ID. A unique number has to be defined.
ALCID	Load curve ID of ground acceleration history.
VLCID	Load curve ID of ground velocity history.

Remarks:

1. Earthquake ground motion data is typically available either only as ground accelerations, or as a triple of ground accelerations, velocities and displacements. Usually, the velocities and the displacements are computed from the accelerations using specialized filtering and baseline correction techniques, e.g. see peer.berkeley.edu/smcat/process.html. Either input is accepted, with each quantity specified as a load curve. Only the acceleration and the velocity is required in the latter case; LS-DYNA does not require the ground displacement.

2. If only the ground acceleration data is provided for a particular ground motion, LS-DYNA generates a corresponding load curve for the velocity by integrating the acceleration numerically. The generated load curves are printed out to the D3HSP file. It is up to the user to ensure that these generated load curves are satisfactory for the analysis.

***DEFINE_HEX_SPOTWELD_ASSEMBLY_**{*OPTION*}

Available options include the number of solid hexahedron elements that are used in the spot weld patch:

<BLANK>

N

Purpose: Define a list of hexahedral solid elements clusters that make up a single spot weld for computing the force and moment resultants that are written into the SWFORC output file. A maximum of a 16 element cluster may be used to define a single spot weld. See Fig. 11.8. This table is generated automatically when beam elements are converted to solid elements. See the input parameter, RPBHX, which is described in the control section: *CONTROL_SPOTWELD_BEAM.

Card 1

1	2	3	4	5	6	7	8
ID_SW							
I							
0							

Card 2

Variable	EID1	EID2	EID3	EID4	EID5	EID6	EID7	EID8
Type	I	I	I	I	I	I	I	I
Default	0	0	0	0	0	0	0	0

Define the following card if and only if N>8

Optional	1	2	3	4	5	6	7	8
Variable	EID9	EID10	EID11	EID12	EID13	EID14	EID15	EID16
Type	I	I	I	I	I	I	I	I
Default	0	0	0	0	0	0	0	0

VARIABLE	DESCRIPTION
ID_SW	Spot weld ID. A unique ID number must be used.
EID*n*	Element ID n for up to 16 solid hexahedral elements.

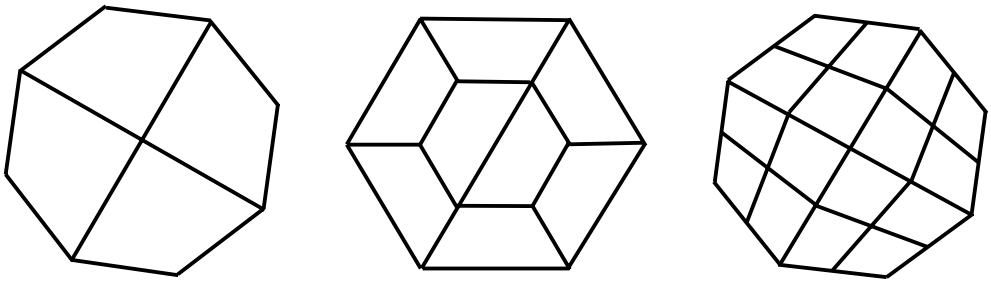


Figure 11.8. Sample four, eight, and sixteen element spot weld clusters comprised of solid hexahedron elements.

*DEFINE_SD_ORIENTATION

Purpose: Define orientation vectors for discrete springs and dampers. These orientation vectors are optional for this element class. Four alternative options are possible. With the first two options, IOP= 0 or 1, the vector is defined by coordinates and is fixed permanently in space. The third and fourth option orients the vector based on the motion of two nodes, so that the direction can change as the line defined by the nodes rotates.

Card	1	2	3	4	5	6	7	8
Variable	VID	IOP	XT	YT	ZT	NID1	NID2	
Type	I	I	F	F	F	I	I	
Default	0	0	0.0	0.0	0.0	0	0	
Remarks	none	1	IOP=0,1	IOP=0,1	IOP=0,1	IOP=2,3	IOP=2,3	

VARIABLE	DESCRIPTION
VID	Orientation vector ID. A unique ID number must be used.
IOP	Option: EQ.0: deflections/rotations are measured and forces/moments applied along the following orientation vector. EQ.1: deflections/rotations are measured and forces/moments applied along the axis between the two spring/damper nodes projected onto the plane normal to the following orientation vector. EQ.2: deflections/rotations are measured and forces/moments applied along a vector defined by the following two nodes. EQ.3: deflections/rotations are measured and forces/moments applied along the axis between the two spring/damper nodes projected onto the plane normal to the a vector defined by the following two nodes.
XT	x-value of orientation vector. Define if IOP=0,1.
YT	y-value of orientation vector. Define if IOP=0,1.
ZT	z-value of orientation vector. Define if IOP=0,1.
NID1	Node 1 ID. Define if IOP=2,3.
NID2	Node 2 ID. Define if IOP=2, 3.

Remarks:

1. The orientation vectors defined by options 0 and 1 are fixed in space for the duration of the simulation. Options 2 and 3 allow the orientation vector to change with the motion of the nodes. Generally, the nodes should be members of rigid bodies, but this is not mandatory. When using nodes of deformable parts to define the orientation vector, care must be taken to ensure that these nodes will not move past each other. If this happens, the direction of the orientation vector will immediately change with the result that initiate severe instabilities can develop.

***DEFINE_SET_ADAPTIVE**

Purpose: To control the adaptive refinement level by element or part set.

Card 1	1	2	3	4	5	6	7	8
Variable	SETID	STYPE	ADPLVL	ADPSIZE				
Type	I	I	I	F				
Default	none	none	none	none				

VARIABLE	DESCRIPTION
SETID	Element set ID or part set ID
STYPE	Set type for SETID: 1-element set 2-part set
ADPLVL	Adaptive refinement level for all elements in SETID set.
ADSIZE	Minimum element size to be adapted based on element edge length for all elements in SETID set.

Remarks:

1. This option is for 3D-shell h-adaptivity only at the present time.
2. The order of defining refinement level for any elements is *CONTROL_ADAPTIVITY and *DEFINE_BOX_ADAPTIVE.
3. If there are multiple definitions of refinement level or element size for any elements, the latter one will be used.

*DEFINE_SPOTWELD_FAILURE_RESULTANTS

Purpose: Define failure criteria between part pairs for predicting spot weld failure. This table is implemented for *solid* element spot welds, which are used with the tied, constraint based, contact option: *CONTACT_TIED_SURFACE_TO_SURFACE. *Note that other tied contact types cannot be used.* The input in this section continues until then next "*" card is encountered. Default values are used for any part ID pair that is not defined. Only one table can defined. See *MAT_SPOTWELD where this option is used whenever *OPT*=7.

Card 1	1	2	3	4	5	6	7	8
Variable	ID	DSN	DSS	DLCIDSN	DLCIDSS			
Type	I	F	F	I	I			
Default	0	0.0	0.0	0	0			

Cards 2, 3, …, The next "*" card terminates the table definition.

Card 2…	1	2	3	4	5	6	7	8
Variable	PID_I	PID_J	SNIJ	SSIJ	LCIDSNIJ	LCIDSSIJ		
Type	I	I	F	F	I	I		
Default	none	none	0.0	0.0	0	0		

VARIABLE	DESCRIPTION
ID	Identification number. Only one table is allowed.
DSN	Default value of the normal static stress at failure.
DSS	Default value of the transverse static stress at failure.
DLCIDSN	Load curve ID defining a scale factor for the normal stress as a function of strain rate. This factor multiplies DSN to obtain the failure value at a given strain rate.

VARIABLE	DESCRIPTION
DLCIDSS	Load curve ID defining a scale factor for static shear stress as a function of strain rate. This factor multiplies DSN to obtain the failure value at a given strain rate.
PID_I	Part ID I.
PID_J	Part ID J.
SNIJ	The maximum axial stress at failure between parts I and J. The axial stress is computed from the solid element stress resultants, which are based on the nodal point forces of the solid element.
DSSIJ	The maximum shear stress at failure between parts I and J. The shear stress is computed from the solid element stress resultants, which are based on the nodal point forces of the solid element.
LCIDSNIJ	Load curve ID defining a scale factor for the normal stress as a function of strain rate. This factor multiplies SNIJ to obtain the failure value at a given strain rate.
LCIDSSIJ	Load curve ID defining a scale factor for static shear stress as a function of strain rate. This factor multiplies SSIJ to obtain the failure value at a given strain rate.

Remarks:

The stress based failure model, which was developed by *Toyota Motor Corporation*, is a function of the peak axial and transverse shear stresses. The entire weld fails if the stresses are outside of the failure surface defined by:

$$\left(\frac{\sigma_{rr}}{\sigma_{rr}^F}\right)^2+\left(\frac{\tau}{\tau^F}\right)^2-1=0$$

where σ_{rr}^F and τ^F are specified in the above table by part ID pairs. LS-DYNA automatically identifies the part ID of the attached shell element for each node of the spot weld solid and checks for failure. If failure is detected the solid element is deleted from the calculation.

If the effects of strain rate are considered, then the failure criteria becomes:

$$\left(\frac{\sigma_{rr}}{f_{dsn}\left(\dot{\varepsilon}^p\right)\sigma_{rr}^F}\right)^2+\left(\frac{\tau}{f_{dss}\left(\dot{\varepsilon}^p\right)\tau^F}\right)^2-1=0$$

*DEFINE_SPOTWELD_RUPTURE_PARAMETER

Purpose: Define a parameter by part ID for shell elements attached to spot weld *beam* elements using the constrained contact option: *CONTACT_SPOTWELD. *This table will not work with other contact types.* Only one table is permitted in the problem definition. Data, which is defined in this table, is used by the stress based spot weld failure model developed by *Toyota Motor Corporation*. See *MAT_SPOTWELD where this option is activated by setting the parameter *OPT* to a value of 9. This spot weld failure model is a development of *Toyota Motor Corporation*.

Card 1

Variable	PID							
Type	I							
Default								

Card 2

Variable	C11	C12	C13	N11	N12	N13		SIG_PF
Type	F	F	F	F	F	F		F
Default								

Card 3

Variable	C21	C22	C23	N2				SIG_NF
Type	F	F	F	F				
Default								

Card 4	1	2	3	4	5	6	7	8
Variable	LCDPA	LCDPM	LCDPS	LCDNA	LCDNM	LCDNS		NSMT
Type	F	F	F	F	F	F		F
Default	0	0	0	0	0	0		0

VARIABLE	DESCRIPTION
PID	Part ID for the attached shell.
C11-N2	Parameters for model, see Remarks below.
SIG_PF	Nugget pull-out stress, σ_P.
SIG_NF	Nugget fracture stress, σ_F.
LCDPA	Curve ID defining dynamic scale factor of spot weld axial load rate for nugget pull-out mode.
LCDPM	Curve ID defining dynamic scale factor of spot weld moment load rate for nugget pull-out mode.
LCDPS	Curve ID defining dynamic scale factor of spot weld shear load rate for nugget pull-out mode.
LCDNA	Curve ID defining dynamic scale factor of spot weld axial load rate for nugget fracture mode.
LCDNM	Curve ID defining dynamic scale factor of spot weld moment load rate for nugget fracture mode.
LCDNS	Curve ID defining dynamic scale factor of spot weld shear load rate for nugget fracture mode.
NSMT	The number of time steps used for averaging the resultant rates for the dynamic scale factors.

Remarks:

This failure model incorporates two failure functions, one for nugget pull-out and the other for nugget fracture. The nugget pull-out failure function is

$$F_p = \frac{C11 \cdot A / D^{N11} + C12 \cdot M / D^{N12} + C13 \cdot S / D^{N13}}{\sigma_P \left[1 + \left(\frac{\dot{\varepsilon}^p}{C} \right)^{1/p} \right]}$$

where *A*, *M*, and *S* are the axial force, moment, and shear resultants respectively, *D* is the spot weld diameter, and the Cowper-Symonds coefficients are from the attached shell material model. If the Cowper-Symonds coefficients aren't specified, the term within the square brackets, [ ], is 1.0. The fracture failure function is

$$F_n = \frac{\sqrt{(C21 \cdot A + C22 \cdot M)^2 + 3(C23 \cdot S)^2}}{D^{N2} \cdot \sigma_F \left[1 + \left(\frac{\dot{\varepsilon}^p}{C} \right)^{1/p} \right]}.$$

When the load curves for the rate effects are specified, the failure criteria are

$$F_p = \frac{C11 \cdot f_{dpa}(\dot{A}) \cdot A / D^{N11} + C12 \cdot f_{dpa}(\dot{M}) \cdot M / D^{N12} + C13 \cdot f_{dpa}(\dot{S}) \cdot S / D^{N13}}{\sigma_P}$$

$$F_n = \frac{\sqrt{(C21 \cdot f_{dna}(\dot{A}) \cdot A + C22 \cdot f_{dnm}(\dot{M}) \cdot M)^2 + 3(C23 \cdot f_{dns}(\dot{S}) \cdot S)^2}}{D^{N2} \cdot \sigma_F}$$

where *f* is the appropriate load curve scale factor. The scale factor for each term is set to 1.0 for when no load curve is specified. No extrapolation is performed if the rates fall outside of the range specified in the load curve to avoid negative scale factors. A negative load curve ID designates that the curve abscissa is the $\log_{10}$ of the resultant rate. This option is recommended when the curve data covers several orders of magnitude in the resultant rate. Note that the load curve dynamic scaling replaces the Cowper-Symonds model for rate effects.

Failure occurs when either of the failure functions is greater than 1.0.

***DEFINE_SPOTWELD_RUPTURE_STRESS**

Purpose: Define a static stress rupture table by part ID for shell elements connected to spot weld *beam* elements using the constrained contact option: *CONTACT_SPOTWELD. *This table will not work with other contact types*. Only one table is permitted in the problem definition. Data, which is defined in this table, is used by the stress based spot weld failure model developed by *Toyota Motor Corporation*. See *MAT_SPOTWELD where this option is activated by setting the parameter *OPT* to a value of 6.

Define rupture stresses part by part. The next "*" card terminates this input.

Card	1	2	3	4	5	6	7	8
Variable	PID	SRSIG	SIGTAU	ALPHA				
Type	I	F	F	F				

VARIABLE	DESCRIPTION
PID	Part ID for the attached shell.
SRSIG	Axial (normal) rupture stress, σ_{rr}^{F}.
SRTAU	Transverse (shear) rupture stress, τ^{F}.
ALPHA	Scaling factor for the axial stress as defined by Toyota. The default value is 1.0.

Remarks:

The stress based failure model, which was developed by *Toyota Motor Corporation*, is a function of the peak axial and transverse shear stresses. The entire weld fails if the stresses are outside of the failure surface defined by:

$$\left(\frac{\sigma_{rr}}{\sigma_{rr}^{F}}\right)^2+\left(\frac{\tau}{\tau^{F}}\right)^2-1=0$$

where σ_{rr}^{F} and τ^{F} are specified in the above table by part ID. LS-DYNA automatically identifies the part ID of the attached shell element for each node of the spot weld beam and independently checks each end for failure. If failure is detected in the end attached to the shell with the greatest plastic strain, the beam element is deleted from the calculation.

If the effects of strain rate are considered, then the failure criteria becomes:

$$\left(\frac{\sigma_{rr}}{\sigma_{rr}^{F}\left(\dot{\varepsilon}^{p}\right)}\right)^{2}+\left(\frac{\tau}{\tau^{F}\left(\dot{\varepsilon}^{p}\right)}\right)^{2}-1=0$$

where $\sigma_{rr}^{F}\left(\dot{\varepsilon}^{p}\right)$ and $\tau^{F}\left(\dot{\varepsilon}^{p}\right)$ are found by using the Cowper and Symonds model which scales the static failure stresses:

$$\sigma_{rr}^{F}\left(\dot{\varepsilon}^{p}\right)=\sigma_{rr}^{F}\cdot\left[1+\left(\frac{\dot{\varepsilon}^{p}}{C}\right)^{1/p}\right]$$

$$\tau^{F}\left(\dot{\varepsilon}^{p}\right)=\tau^{F}\cdot\left[1+\left(\frac{\dot{\varepsilon}^{p}}{C}\right)^{1/p}\right]$$

where $\dot{\varepsilon}^{p}$ is the average plastic strain rate which is integrated over the domain of the attached shell element, and the constants *p* and *C* are uniquely defined at each end of the beam element by the constitutive data of the attached shell. The constitutive model is described in the material section under keyword: *MAT_PIECEWISE_LINEAR_PLASTICITY.

The peak stresses are calculated from the resultants using simple beam theory.

$$\sigma_{rr}=\frac{N_{rr}}{A}+\frac{\sqrt{M_{rs}^{2}+M_{rt}^{2}}}{\alpha Z}\qquad\tau=\frac{M_{rr}}{2Z}+\frac{\sqrt{N_{rs}^{2}+N_{rt}^{2}}}{A}$$

where the area and section modulus are given by:

$$A=\pi\frac{d^{2}}{4}$$

$$Z=\pi\frac{d^{3}}{32}$$

and *d* is the diameter of the spot weld beam.

*DEFINE_STAGED_CONSTRUCTION_PART

Purpose: Staged construction. This keyword offers a simple way to define parts that are removed (e.g., during excavation), added (e.g., new construction) and used temporarily (e.g., props) during the analysis. Available for solid, shell, and beam element parts.

Note: This keyword card will be available starting in release 3 of version 971.

Card	1	2	3	4	5	6	7	8
Variable	PID	STGA	STGR					
Type	I	I	I					
Default	none	See Remarks	See Remarks					

VARIABLE	DESCRIPTION
PID	Part ID
STGA	Construction stage at which part is added
STGR	Construction stage at which part is removed

Remarks:

Used with *DEFINE_CONSTRUCTION_STAGES (defines the meaning of stages STGA and STGR) and *CONTROL_STAGED_CONSTRUCTION. If STGA=0, the part is present at the start of the analysis. If STGR=0, the part is still present at the end of the analysis. Examples:

1. Soil that is excavated would have STGA=0 but STGR>0

2. New construction would have STGA>0 and STGR=0

3. Temporary works would have STGA>0, STGR>STGA.

This is a convenience feature that reduces the amount of input data needed for many typical construction models. Internally, LS-DYNA checks for *LOAD_REMOVE_PART, *LOAD_GRAVITY_PART and *LOAD_STIFFEN_PART referencing the same PID. Generally, these will not be present and LS-DYNA creates the data using STGA and STGR, and default gravity and pre-construction stiffness factor from *CONTROL_STAGED_CONSTRUCTION. If existing cards are found, STGA and STGR are inserted into the existing data. During the analysis, any load curves entered on those existing cards will override STGA and STGR.

*DEFINE_TABLE

Purpose: Define a table. This input section is somewhat unique in that another keyword, ***DEFINE_CURVE**, is used as part of the input in this section. A table consists of a ***DEFINE_TABLE** card followed by n lines of input. Each of the n additional lines define a numerical value in ascending order corresponding to a ***DEFINE_CURVE** input which follows the ***DEFINE_TABLE** keyword and the related input. For example, to define strain rate dependency where it is desired to provide a stress versus strain curve for each strain rate, n strain rates would be defined following the ***DEFINE_TABLE** keyword. The curves then follow which make up the table. Each curve may have unique spacing and an arbitrary number of points in their definition. (Load curve ID's defined for the table may be referenced elsewhere in the input.) *However, the curves must not cross except at the origin and the curves must share the same origin and end point.* This rather awkward input is done for efficiency reasons related to the desire to avoid indirect addressing in the inner loops used in the constitutive model stress evaluation.

Card 1

Variable	TBID	SFA	OFFA					
Type	I	F	F					
Default	none	1.	0.					

Card 2, 3, 4, etc. Put one point per card (E20.0). Input is terminated when a "*DEFINE_CURVE" card is found.

Card 2…

Variable	VALUE			
Type	F			
Default	0.0			

Insert one ***DEFINE_CURVE** input section here for each point defined above.

VARIABLE	DESCRIPTION
TBID	Table ID. Tables and Load curves may not share common ID's. LS-DYNA allows load curve ID's and table ID's to be used interchangeably.
SFA	Scale factor for VALUE.
OFFA	Offset for VALUE, see explanation below.
VALUE	Load curve will be defined corresponding to this value, e.g., this value could be a strain rate, see purpose above.

Remarks:

1. If for example, 10 stress-strain curves for 10 different strain rates are given, 10 cards with the ascending values of strain rate then follow the first card. Afterwards, 10 corresponding *DEFINE_CURVE specifications have to follow.

2. VALUE is scaled in the same manner as in *DEFINE_CURVE, i.e.,

$$Scaled\ value = SFA \cdot (Defined\ value + OFFA).$$

*DEFINE_TABLE_2D

Purpose: Define a table. Unlike the *DEFINE_TABLE keyword above, a curve ID is specified for each value defined in the table. This allows the same curve ID to be referenced by multiple tables, and the curves may be defined anywhere in the input file. As in *DEFINE_TABLE, each curve may have unique spacing and an arbitrary number of points in their definition. *However, the curves must not cross except at the origin and the curves must share the same origin and end point*

Card 1	1	2	3	4	5	6	7	8
Variable	TBID	SFA	OFFA					
Type	I	F	F					
Default	none	1.	0.					

Card 2, 3, 4, etc. Define one point and curve ID per card (E20.0,I20). Input is terminated when a "*" card is found.

Card 2…	1	2	3	4	5	6	7	8
Variable	VALUE		CURVE ID					
Type	F		I					
Default	0.0		none					

VARIABLE	DESCRIPTION
TBID	Table ID. Tables and Load curves may not share common ID's. LS-DYNA allows load curve ID's and table ID's to be used interchangeably.
SFA	Scale factor for VALUE.
OFFA	Offset for VALUE, see explanation below.

VARIABLE	DESCRIPTION
VALUE	Load curve will be defined corresponding to this value, e.g., this value could be a strain rate for example.
CURVEID	Load curve ID.

Remarks:

1. VALUE is scaled in the same manner as in *DEFINE_CURVE, i.e.,

$$Scaled\ value = SFA \cdot (Defined\ value + OFFA).$$

*DEFINE_TABLE_3D

Purpose: Define a three dimensional table. For each value defined below, a table ID is specified. For example, in a thermally dependent material model, the value given below could correspond to temperture for a table ID defining effective stress versus strain curves for a set of strain rate values. Each table ID can be referenced by multiple three dimensional tables, and the tables may be defined anywhere in the input.

Card 1

1	2	3	4	5	6	7	8
Variable	TBID	SFA	OFFA					
Type	I	F	F					
Default	none	1.	0.					

Card 2, 3, 4, etc. Define one point and table ID per card (E20.0,I20). Input is terminated when a "*" card is found.

Card 2…

1	2	3	4	5	6	7	8
Variable	VALUE	TABLE ID					
Type	F	I					
Default	0.0	none					

VARIABLE	DESCRIPTION
TBID	Table ID. Tables and Load curves may not share common ID's. LS-DYNA allows load curve ID's and table ID's to be used interchangeably.
SFA	Scale factor for VALUE.
OFFA	Offset for VALUE, see explanation below.
VALUE	Load curve will be defined corresponding to this value, e.g., this value could be a strain rate for example.

VARIABLE	DESCRIPTION
TABLEID	Table ID.

Remarks:

1. VALUE is scaled in the same manner as in *DEFINE_CURVE, i.e.,

$$Scaled\ value = SFA \cdot (Defined\ value + OFFA).$$

*DEFINE_TRACER_PARTICLES_2D

Purpose: Define tracer particles that follow the deformation of a material. This is useful for visualizing the deformation of a part that is being adapted in a metal forming operation. Nodes used as tracer particles should only be used for visualization and not associated with anything in the model that may alter the response of the model, e.g., they should not be used in any elements except those with null materials.

Card 1 1 2 3 4 5 6 7 8

Variable	NSET	PSET						
Type	I	I						
Default	none	0						

VARIABLE	DESCRIPTION
NSET	The node set ID for the nodes used as tracer particles.
PSET	Optional part set ID. If this part set is specified, only tracer particles in these parts are updated and the others are stationary. If this part set is not specified, all tracer particles are updated.

***DEFINE_TRANSFORMATION**

Purpose: Define a transformation for the INCLUDE_TRANSFORM keyword option. The *DEFINE_TRANSFORMATION command must be defined before the *INCLUDE_ TRANSFORM command can be used.

Cards 1, 2, 3, 4, ... (The next "*" card terminates the input.) This set is a combination of a series of options listed in the table defined below.

Card 1

	1	2	3	4	5	6	7	8
Variable	TRANID							
Type	I							
Default	none							

Card 2

Variable	OPTION	A1	A2	A3	A4	A5	A6	A7
Type	A	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
TRANID	Transform ID.
OPTION	For the available options see the table below.
A1-A7	Specified entity. Each card must have an option specified. See Table 11.1 below for the available options.

Table 11.1

FORMAT (A10,7F10.0)		
OPTION	ENTITIES + ATTRIBUTES	FUNCTION
SCALE	a1, a2, a3	Scale the global x, y, and z coordinates of a point by a1, a2, and a3, respectively. If zero, a default of unity is set.
ROTATE	a1, a2, a3, a4, a5, a6, a7	Rotate through an angle, a7, about a line with direction cosines a1, a2, and a3 passing through the point with coordinates a4, a5, and a6. If a4 through a7 are zero, then a1 and a2 are the ID's of two POINTs and a3 defines the rotation angle. The axis of rotation is defined by a vector going from point with ID a1 to point with ID a2.
TRANSL	a1, a2, a3	Translate the x, y, and z coordinates of a point by a1, a2, and a3, respectively.
POINT	a1,a2,a3,a4	Define a point with ID, a1, with the initial coordinates a2, a3, and a4.
POS6P	a1, a2, a3, a4, a5, a6	Positioning by 6 points. Affine transformation (rotation and translation, no scaling) given by three start points a1, a2, and a3 and three target points a4, a5, and a6. The six POINTs must be defined before they are referenced.

The ordering of the SCALE, ROTATE, and TRANSL commands is important. It is generally recommend to first scale, then rotate, and finally translate the model.

The POINT option in ROTATE provides a means of defining rotations about axes defined by the previous transformations. The coordinates of the two POINTs are transformed by all the transformations up to the transformation where they are referenced. The POINTs must be defined before they are referenced, and their identification numbers are local to each *DEFINE_TRANSFORMATION. The coordinates of a POINT are transformed using all the transformations before it is referenced, not just the transformations between its definition and its reference. To put it another way, while the ordering of the transformations is important, the ordering between the POINTs and the transformations is not important.

In the following example, the *DEFINE_TRANSFORMATION command is used 3 times to input the same dummy model and position it as follows:

1. Transformation id 1000 imports the dummy model (dummy.k) and rotates it 45 degrees about z-axis at the point (0.0,0.0,0.0). Transformation id 1001 performs the same transformation using the POINT option.

2. Transformation id 2000 imports the same dummy model (dummy.k) and translates 1000 units in the x direction.

3. Transformation id 3000 imports the same dummy model (dummy.k) and translates 2000 units in the x direction. For each *DEFINE_TRANSFORMATION, the commands TRANSL, SCALE, and ROTATE are available. The transformations are applied in the order in which they are defined in the file, e.g., transformation id 1000 in this example would translate, scale and then rotate the model. *INCLUDE_TRANSFORM uses a transformation id defined by a *DEFINE_TRANSFORMATION command to import a model and perform the associated transformations. It also allows the user upon importing the model to apply offsets to the various entity ids and perform unit conversion of the imported model.

```
*KEYWORD
*DEFINE_TRANSFORMATION
      1000
$ option &       dx&       dy&       dz&
TRANSL        0000.0       0.0       0.0
$ option &       dx&       dy&       dz&
SCALE           1.00       1.0       1.0
$ option &       dx&       dy&       dz&       px&       py&       pz&
angle&
ROTATE          0.00       0.0       1.0      0.00      0.00       0.0
45.00
*DEFINE_TRANSFORMATION
      1001
POINT              1       0.0       0.0       0.0
POINT              2       0.0       0.0       1.0
ROTATE             1         2      45.0
*DEFINE_TRANSFORMATION
      2000
$ option &       dx&       dy&       dz&
TRANSL        1000.0       0.0       0.0
*DEFINE_TRANSFORMATION
$ tranid &
     3000
$ option &       dx&       dy&       dz&

TRANSL        2000.0       0.0       0.0
*INCLUDE_TRANSFORM
dummy.k
$idnoff  &   ideoff&   idpoff& idmoff  &  idsoff &  iddoff&  iddoff &
         0         0         0         0         0        0         0
$  idroff&   ilctmf&
         0         0
$  fctmas&   fcttim&   fctlen&  fcttem &   incout&
    1.0000    1.0000      1.00       1.0         1
$ tranid &
      1000
*INCLUDE_TRANSFORM
dummy.k
$idnoff  &   ideoff&   idpoff& idmoff  &  idsoff &  iddoff&  iddoff &
   1000000   1000000   1000000   1000000   1000000  1000000   1000000
$  idroff&   ilctmf&
   1000000   1000000
$  fctmas&   fcttim&   fctlen&  fcttem &   incout&
    1.0000    1.0000      1.00       1.0         1
$ tranid &
      2000
*INCLUDE_TRANSFORM
dummy.k
$idnoff  &   ideoff&   idpoff& idmoff  &  idsoff &  iddoff&  iddoff &
```

```
   2000000   2000000   2000000   2000000   2000000  2000000   2000000
$  idroff&   ilctmf&
   2000000   2000000
$  fctmas&   fcttim&   fctlen&  fcttem &   incout&
    1.0000    1.0000      1.00       1.0         1
$ tranid &
      3000
*END
```

*DEFINE_VECTOR

Purpose: Define a vector by defining the coordinates of two points.

Card	1	2	3	4	5	6	7	8
Variable	VID	XT	YT	ZT	XH	YH	ZH	CID
Type	I	F	F	F	F	F	F	I
Default	0	0.0	0.0	0.0	0.0	0.0	0.0	0
Remarks								

VARIABLE	DESCRIPTION
VID	Vector ID
XT	X-coordinate of tail of vector
YT	Y-coordinate of tail of vector
ZT	Z-coordinate of tail of vector
XH	X-coordinate of head of vector
YH	Y-coordinate of head of vector
ZH	Z-coordinate of head of vector
CID	Coordinate system ID to define vector in local coordinate system. All coordinates, XT, YT, ZT, XH, YH, and ZH are in respect to CID. EQ.0: global (default).

Remarks:

1. The coordinates should differ by a certain margin to avoid numerical inaccuracies.

EXAMPLES

The following examples demonstrate the input for these options:

*DEFINE_BOX
*DEFINE_COORDINATE_NODES,
*DEFINE_COORDINATE_SYSTEM,
*DEFINE_COORDINATE_VECTOR
*DEFINE_CURVE
*DEFINE_SD_ORIENTATION
*DEFINE_VECTOR commands.

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$   *DEFINE_BOX
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Define box number eight which encloses a volume defined by two corner
$  points: (-20.0, -39.0, 0.0) and (20.0, 39.0, 51.0).  As an example, this
$  box can be used as an input for the *INITIAL_VELOCITY keyword in which
$  all nodes within this box are given a specific initial velocity.
$
*DEFINE_BOX
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$    boxid       xmm       xmx       ymn       ymx       zmn       zmx
         8     -20.0      20.0     -39.0      39.0       0.0      51.0
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$



$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$   *DEFINE_COORDINATE_NODES
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Define local coordinate system number 5 using three nodes: 10, 11 and 20.
$  Nodes 10 and 11 define the local x-direction.  Nodes 10 and 20 define
$  the local x-y plane.
$
$  For example, this coordinate system (or any coordinate system defined using
$  a *DEFINE_COORDINATE_option keyword) can be used to define the local
$  coordinate system of a joint, which is required in order to define joint
$  stiffness using the *CONSTRAINED_JOINT_STIFFNESS_GENERALIZED keyword.
$
*DEFINE_COORDINATE_NODES
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$      cid        n1        n2        n3
         5        10        11        20
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$   *DEFINE_COORDINATE_SYSTEM
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Define local coordinate system number 3 using three points.  The origin of
$  local coordinate system is at (35.0, 0.0, 0.0).  The x-direction is defined
$  from the local origin to (35.0, 5.0, 0.0).  The x-y plane is defined using
$  the vector from the local origin to (20.0, 0.0, 20.0) along with the local
$  x-direction definition.
$
*DEFINE_COORDINATE_SYSTEM
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$      cid        Xo        Yo        Zo        Xl        Yl        Zl
         3      35.0       0.0       0.0      35.0       5.0       0.0
$
$       Xp        Yp        Zp
      20.0       0.0      20.0
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$   *DEFINE_COORDINATE_VECTOR
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Define local coordinate system number 4 using two vectors.
$    Vector 1 is defined from (0.0, 0.0, 0.0) to (1.0, 1.0, 0.0)
$    Vector 2 is defined from (0.0, 0.0, 0.0) to (1.0, 1.0, 1.0)
$  See the corresponding keyword command for a description.
$
*DEFINE_COORDINATE_VECTOR
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$      cid        Xx        Yx        Zx        Xv        Yv        Zv
         4       1.0       1.0       0.0       1.0       1.0       1.0
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$   *DEFINE_CURVE
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Define curve number 517.  This particular curve is used to define the
$  force-deflection properties of a spring defined by a *MAT_SPRING_INELASTIC
$  keyword.  The abscissa value is offset 25.0 as a means of modeling a gap
$  at the front of the spring.  This type of spring would be a compression
$  only spring.
$
*DEFINE_CURVE
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$     lcid      sidr      scla      sclo      offa      offo
       517                                    25.0
$
$          abscissa            ordinate
                0.0                 0.0
               80.0                58.0
               95.0                35.0
              150.0                44.5
              350.0                45.5
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$   *DEFINE_SD_ORIENTATION
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  A discrete spring is defined with two nodes in 3-D space.  However, it is
$  desired to have the force of that spring to act only in the z-direction.
$  The following definition makes this happen.  Additionally, vid = 7
$  must be specified in the *ELEMENT_DISCRETE keyword for this spring.
$
*DEFINE_SD_ORIENTATION
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$      vid       iop        xt        yt        zt      nid1      nid2
         7         0       0.0       0.0       1.0
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$   *DEFINE_VECTOR
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Define vector number 5 from (0,0,0) to (0,1,1).  As an example, this vector
$  can be used to define the direction of the prescribed velocity of a node
$  using the *BOUNDARY_PRESCRIBED_MOTION_NODE keyword.
$
*DEFINE_VECTOR
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$     vid        xt        yt        zt        xh        yh        zh
        3       0.0       0.0       0.0       0.0       1.0       1.0
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

*DEFORMABLE_TO_RIGID

The cards in this section are defined in alphabetical order and are as follows:

***DEFORMABLE_TO_RIGID**

***DEFORMABLE_TO_RIGID_AUTOMATIC**

***DEFORMABLE_TO_RIGID_INERTIA**

If one of these cards is defined, then any deformable part defined in the model may be switched to rigid during the calculation. Parts that are defined as rigid (*MAT_RIGID) in the input are permanently rigid and cannot be changed to deformable.

Deformable parts may be switched to rigid at the start of the calculation by specifying them on the *DEFORMABLE_TO_RIGID card.

Part switching may be specified on a restart (see RESTART section of this manual) or it may be performed automatically by use of the *DEFORMABLE_TO_RIGID_AUTOMATIC cards.

The *DEFORMABLE_TO_RIGID_INERTIA cards allow inertial properties to be defined for deformable parts that are to be swapped to rigid at a later stage.

It is not possible to perform part material switching on a restart if it was not flagged in the initial analysis. The reason for this is that extra memory needs to be set up internally to allow the switching to take place. If part switching is to take place on a restart, but no parts are to be switched at the start of the calculation, no inertia properties for switching and no automatic switching sets are to be defined, then just define one *DEFORMABLE_TO_RIGID card without further input.

***DEFORMABLE_TO_RIGID**

Purpose: Define materials to be switched to rigid at the start of the calculation.

Card	1	2	3	4	5	6	7	8
Variable	PID	MRB						
Type	I	I						
Default	none	0						

VARIABLE	DESCRIPTION
PID	Part ID of the part which is switched to a rigid material, also see *PART.
MRB	Part ID of the master rigid body to which the part is merged. If zero, the part becomes either an independent or master rigid body.

*DEFORMABLE_TO_RIGID_AUTOMATIC

Purpose: Define a set of parts to be switched to rigid or to deformable at some stage in the calculation.

Card 1

Variable	SWSET	CODE	TIME 1	TIME 2	TIME 3	ENTNO	RELSW	PAIRED
Type	I	I	F	F	F	I	I	I
Default	none	0	0.	1.0E20	0.	0.	0	0
Remark		1				1,2		3

Card 2

Variable	NRBF	NCSF	RWF	DTMAX	D2R	R2D	OFFSET	
Type	I	I	I	F	I	I	F	
Default	0	0	0	0.	0	0	0	
Remark	4	4	4					

VARIABLE **DESCRIPTION**

SWSET — Set number for this automatic switch set. Must be unique.

CODE — Activation switch code. Defines the test to activate the automatic material switch of the part:

EQ.0: switch takes place at time 1,

EQ.1: switch takes place between time 1 and time 2 if rigid wall force (specified below) is zero,

EQ.2: switch takes place between time 1 and time 2 if contact surface force (specified below) is zero,

EQ.3: switch takes place between time 1 and time 2 if rigid wall force (specified below) is non-zero,

EQ.4: switch takes place between time 1 and time 2 if contact surface force (specified below) is non-zero.

VARIABLE	DESCRIPTION
TIME 1	Switch will not take place before this time.
TIME 2	Switch will not take place after this time: EQ.0 Time 2 set to 1.0e20.
TIME 3	Delay period. After this part switch has taken place, another automatic switch will not take place for the duration of the delay period. If set to zero a part switch may take place immediately after this switch.
ENTNO	Rigid wall/contact surface number for switch codes 1, 2, 3, 4.
RELSW	Related switch set. The related switch set is another automatic switch set that must be activated before this part switch can take place: EQ.0: no related switch set.
PAIRED	Define a pair of related switches. EQ. 0: not paired EQ. 1: paired with switch set RELSW and is the Master switch. EQ.-1: paired with switch set RELSW and is the Slave switch.
NRBF	Flag to delete or activate nodal rigid bodies. If nodal rigid bodies or generalized, weld definitions are active in the deformable bodies that are switched to rigid, then the definitions should be deleted to avoid instabilities: EQ.0: no change, EQ.1: delete, EQ.2: activate.
NCSF	Flag to delete or activate nodal constraint set. If nodal constraint/spot weld definitions are active in the deformable bodies that are switched to rigid, then the definitions should be deleted to avoid instabilities: EQ.0: no change, EQ.1: delete, EQ.2: activate.
RWF	Flag to delete or activate rigid walls: EQ.0: no change, EQ.1: delete, EQ.2: activate.
DTMAX	Maximum permitted time step size after switch.
D2R	Number of deformable parts to be switched to rigid plus number of rigid parts for which new master/slave rigid body combinations will be defined: EQ.0: no parts defined.
R2D	Number of rigid parts to be switched to deformable: EQ.0: no parts defined.

VARIABLE	DESCRIPTION
OFFSET	Optional contact thickness for switch to deformable. For contact, its value should be set to a value greater than the contact thickness offsets to ensure the switching occurs prior to impact. This option applies if and only if CODE is set to 3 or 4. For CODE=3 all rigid wall options are implemented. For CODE=4, the implementation works for the contact type CONTACT_AUTOMATIC_ when the options: ONE_WAY_ SURFACE_TO_SURFACE, NODES_TO_SURFACE, and SURFACE_ TO_ SURFACE are invoked.

Remarks:

1. Only surface to surface and node to surface contacts can be used to activate an automatic part switch.

2. Rigid wall numbers are the order in which they are defined in the deck. The first rigid wall and the first contact surface encountered in the input deck will have an entity number of 1. The contact surface id is that as defined on the *CONTACT_....._ID card.

3. Switch sets may be paired together to allow a pair of switches to be activated more than once. Each pair of switches should use consistent values for CODE, i.e. 1&3 or 2&4. Within each pair of switches the related switch, RELSW, should be set to the ID of the other switch in the pair. The Master switch (PAIRED = 1) will be activated before the Slave switch (PAIRED = -1). Pairing allows the multiple switches to take place as for example when contact is made and lost several times during an analysis.

4. If the delete switch is activated, ALL corresponding constraints are deactivated regardless of their relationship to a switched part. By default, constraints which are directly associated with a switched part are deactivated/activated as necessary.

```
$  Define a pair or related switches that will be activated by (no)force on
$  Contact 3.  To start with switch set 20 will be activated (PAIRED=1) swapping
$  the PARTS to RIGID. When the contact force is none zero switch set 10 will be
$  activated swapping the PARTS to DEFORMABLE.  If the contact force returns to
$  zero switch set 20 will be activated again making the PARTS RIGID.
$
*DEFORMABLE_TO_RIGID_AUTOMATIC
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$    swset      code    time 1    time 2    time 3     entno     relsw    paired
        20         2                                       3        10         1
$     nrbf      ncsf       rwf     dtmax       D2R       R2D
                                                1
*DEFORMABLE_TO_RIGID_AUTOMATIC
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$    swset      code    time 1    time 2    time 3     entno     relsw    paired
        10         4                                       3        20        -1
$     nrbf      ncsf       rwf     dtmax       D2R       R2D
                                                    1
```

*DEFORMABLE_TO_RIGID *DEFORMABLE_TO_RIGID_AUTOMATIC

Define D2R cards below:

Card	1	2	3	4	5	6	7	8
Variable	PID	MRB						
Type	I	I						
Default	none	0						

VARIABLE	DESCRIPTION
PID	Part ID of the part which is switched to a rigid material.
MRB	Part ID of the master rigid body to which the part is merged. If zero, the part becomes either an independent or master rigid body.

Define R2D cards below:

Card	1	2	3	4	5	6	7	8
Variable	PID							
Type	I							
Default	none							

VARIABLE	DESCRIPTION
PID	Part ID of the part which is switched to a deformable material.

*DEFORMABLE_TO_RIGID_INERTIA

Purpose: Inertial properties can be defined for the new rigid bodies that are created when the deformable parts are switched. These can only be defined in the initial input if they are needed in a later restart. Unless these properties are defined, LS-DYNA will recompute the new rigid body properties from the finite element mesh. The latter requires an accurate mesh description. **When rigid bodies are merged to a master rigid body, the inertial properties defined for the master rigid body apply to all members of the merged set.**

Card 1

Variable	PID							
Type	I							
Default	none							

Card 2

Variable	XC	YC	ZC	TM				
Type	F	F	F	F				

Card 3

Variable	IXX	IXY	IXZ	IYY	IYZ	IZZ		
Type	F	F	F	F	F	F		
Default	none	0.0	0.0	none	0.0	none		

VARIABLE	DESCRIPTION
PID	Part ID, see *PART.
XC	x-coordinate of center of mass
YC	y-coordinate of center of mass
ZC	z-coordinate of center of mass
TM	Translational mass
IXX	I_{xx}, xx component of inertia tensor
IXY	I_{xy}
IXZ	I_{xz}
IYY	I_{yy}
IYZ	I_{yz}
IZZ	I_{zz}

*EF

Exchange factors characterize radiative heat transfer between collections of flat surfaces, the union of which is a closed surface (an enclosure). LS-DYNA can calculate exchange factors and then use them as boundary conditions for thermal runs. The $(i,j)^{th}$ element of an exchange factor matrix, E_{ij}, is the fraction of the Stefan-Boltzman surface energy radiated from surface `i` that is absorbed by surface `j`. LS-DYNA employs a Monte Carlo algorithm to calculate these exchange factors. For each surface, LS-DYNA simulates photon emission one photon at a time. For each photon, LS-DYNA generates a random initial position on the emitting surfaces as well as a random initial direction that points into the enclosure. LS-DYNA ray traces each photon until it is absorbed. The path of a simulated photon can be complex involving multiple diffuse and specular reflections as well as multiple diffuse and specular transmissions. The results of this Monte Carlo algorithm are used to assemble a matrix that is related to the exchange factor matrix, for which, the $(i,j)^{th}$ entry contains the number of photons emitted from surface `i` that are absorbed by surface `j`. From this matrix LS-DYNA then assembles the exchange factor matrix.

Limitations

The exchange factor algorithm is used to model heat transfer across an enclosure containing a non-participating media. The media within the enclosure is assumed to be transparent to the electromagnetic radiation. For modeling heat transfer across enclosures that are made entirely of diffusively reflecting grey-body surfaces, LS-DYNA features a simpler and faster-running algorithm than the Monte Carlo algorithm, that is, the view factor method (see `*BOUNDARY_RADATION_VF, type=2`). The exchange factor calculation cannot be used concurrently with view factors. A further limitation is that LS-DYNA is capable of including only one exchange factor enclosure per simulation.

Output

The file “exchfl” is a text file containing the exchange factors. This file is written when using the *BOUNDARY_RADIATION_SET_VF_CALCULATE keyword and read when using the *BOUNDARY_RADIATION_SET_VF_READ keyword.

EF Cards

LS-DYNA requires that the user supply all of the cards listed below unless noted as optional.

- ***EF_CONTROL**
- ***EF_GRID** (optional)
- ***EF_MATERIAL**
- ***EF_TOGGLES** (optional)

***EF_CONTROL**

Purpose: This card allows the user to set the parameters for the Monte Carlo algorithm.

This keyword should be used only once.

Card (1 of 2)

Card 1	1	2	3	4	5	6	7	8
Variable	NPHTON	NREFS	NWARNS	NLOST	NLOOPS	ERRDEF	INSEED	
Type	I	I	I	I	I	F	I	
Default	None	100	100	100	1	.01	0	

Card (2 of 2)

Card 2	1	2	3	4	5	6	7	8
Variable	DELT	SPLTOL	AREATOL	NINCR				
Type	F	F	F	I				
Default	0.01	0.0001	0.0001	0				

VARIABLE	DESCRIPTION
NPHTON	The base number of photons emitted per band per surface per convergence loop. Note that NPHT from *BOUNDARY_ RADIATION_SET_EF_CALCULATE also effects the number of photons emitted per surface per band per convergence loop.
NREFS	The maximum number of reflections allowed per photon before LS-DYNA issues a warning.
NWARNS	The maximum number of warnings allowed per surface before the run is aborted
NLOST	The maximum number of lost photons allowed per surface. Round off error often causes the loss of photons, so this number ought not to be set too small (usually the default is reasonable).

VARIABLE	DESCRIPTION
NLOOPS	This specifies the maximum number of convergence loops. If the relative error obtained upon the completion of a run is not within the specified tolerances, LS-DYNA will rerun the model combining the results of all previous runs together with the results of the present run to obtain a more accurate result. LS-DYNA will rerun the problem NLOOPS times to achieve error margins within the specified tolerances. If the desired level of convergence is not obtained within NLOOPS iterations LS-DYNA error terminates.
ERRDEF	Specifies that tolerance for convergence of the surface exchange fractions. This may be overridden on a surface by surface basis with the ERRMAX setting. (see *BOUNDARY_RADIATION_ SET_EF_CALCULATE)

INSEED	Tells LS-DYNA how to obtain an initial seed for the Monte Carlo random number generator.
	`if(INSEED.eq.0) then [use date and time]`
	`if(INSEED.gt.0) then [use INSEED as seed]`
	`if(INSEED.lt.0) then [use a default seed]`

DELT	The cone angle interval used to numerically integrate material properties.
SPLTOL	To calculate exchange factors, LS-DYNA splits all of the enclosure's quadrilateral surfaces into two triangular surfaces. SPLTOL specifies the amount by which the dot product of the unit normal vectors of the two triangular surfaces can differ from unity.
AREATOL	LS-DYNA splits quadrilateral surfaces in the enclosures along the line connecting the first and third nodes. Quadrilaterals could, just as well, be split along the line connecting the second and fourth nodes. For numerical stability it is important the areas of the triangles created by either splitting be almost identical. AREATOL specifies the largest allowable difference in area.

NINCR	Controls restart-related behavior of LS-DYNA's exchange factor solver.
	`if(NINCR.eq.0) then [Run normal, no restart files output]`
	`if(NINCR.gt.0) then [Write restart file after every NINC surfaces]`

***EF_GRID**

Purpose: This card allows the user to specify grid parameters.

This keyword should appear only once.

Card (1 of 1)

Card 1	1	2	3	4	5	6	7	8
Variable	NGX	NGY	NGZ					
Type	I	I	I					
Default	None	None	None					

VARIABLE	DESCRIPTION
NGX	The mathematical algorithm underlying the ray tracer, involves gridding the enclosure. NGX specify the number of grid divisions along the x axis. This parameter does not affect LS-DYNA's ability to obtain a solution, but it does effect the amount of CPU time consumed to process each photon. There is no fixed rule for picking NGX, NGY, and NGZ, however for large geometries involving 1,000 to 15,000 surfaces NGX = NGY = NGZ = 25 is often optimal. For smaller geometries smaller values are recommended.
NGY	Specifies the number of grid divisions along the y-axis.
NGZ	Specifies the number of grid divisions along the z-axis.

*EF_MATERIAL

Purpose: This keyword defines exchange factor material IDs. To define multiple materials use this keyword more than once.

(Material Specifier)

Card 1	1	2	3	4	5	6	7	8
Variable	NMAT	NAME						
Type	F	A70						
Default	none	none						

These next two cards specify the properties of the material.

Card 2	1	2	3	4	5	6	7	8
Variable	MTYP	EXE	EYE	EZE				
Type	I	F	F	F				
Default	0	0	0	0				

Card 3

Variable	RHOS	RHOD	TAUS	TAUD	RDIFFR	RDIFFT		
Type	F	F	F	F	F	F		
Default	0	0	0	0	1	1		
Remark	1	1	1	1	2	2		

VARIABLE	DESCRIPTION
NMAT	Specifies the material ID, of the exchange factor material.
NAME	Specifies the material's name. This parameter is used only to make the output file easier to read.
MTYP	Specifies if and how emission occurs:

MTYP	Material Emission
-2	There is to be no emission and $F_{ij} = 1$ is written to the output file for this surface.
-1	There is to be no emission and $F_{ij} = 0$ is written to the output file for this surface.
0	Emission is to be distributed in θ according to: $\varepsilon(\theta) = \cos^{r}(\theta)$
1	Beam emission is to occur in the direction $\{E_X, E_Y, E_Z\}$
2	This specifies that emission according to user specified function.

EXE	Specifies the x component of emission for a type 1 material.
EYE	Specifies the y component of emission for a type 1 material.
EZE	Specifies the z component of emission for a type 1 material.
RHOS	Specifies the specular reflectance.
RHOD	Specifies the diffuse reflectance.
TAUS	Specifies the specular transmittance.
TAUD	Specifies the diffuse transmittance.
RDIFFR	LS-DYNA simulates diffuse reflection according to the equation: $\varepsilon(\theta) = \cos^{r}(\theta)$. The user specifies the value for r with RDIFFR.
RDIFFT	LS-DYNA simulates diffuse transmittance according to the equation: $\varepsilon(\theta) = \cos^{r}(\theta)$. The user specifies the value for r with RDIFFT.

Remarks:

1. The standard cosine dependent probability function can be replaced with user-defined probability functions. Negative values of this parameter are taken to be material curve Ids that identify such user-defined probability functions. The range of the defined curve is 0 to 90 degrees.

2. Values different from 1 have been observed to result in errors in reciprocity, so the user is strongly encouraged to consider this when selecting values for RDIFFR and RDIFFT different from 1. Lambertian behavior is achieved by using a value of 1. Values greater than 1 result in biasing the distribution toward the normal, whereas values less than one result in biasing the distribution toward the grazing angle.

*EF_TOGGLES

Purpose: This card allows the user to set output options.

This keyword should be used only once.

Card 1	1	2	3	4	5	6	7	8
Variable	IPRINT1	IPRINT2	IPRINT3	IPRINT4	IDATA	ITRACES	IRSTRT	
Type	I	I	I	I	I	I	I	
Default	0	0	0	0	0	0	0	

VARIABLE	DESCRIPTION
IPRINT1	Controls output of exchange fractions to the d3hsp file. In almost all situations this should be set to 0 because the “exchange factors” are written to the file exchfl. EQ.0: do not write exchange fractions EQ.1: write exchange fraction.
IPRINT2	Controls output of a list of lost photons to the d3hsp file. This is useful for debugging. EQ.0: do not write lost photon list EQ.1: write lost photon list
IPRINT3	Controls output about the grid algorithm to the d3hsp file. EQ.0: do not write grid algorithm information EQ.1: write grid algorithm information
IPRINT4	Controls output about material information pertaining to exchange factors to the d3hsp file. EQ.0: do not write material information EQ.1: write material information
IDATA	Controls execution EQ.0: run proceeds EQ.1: terminate after input parameter check
ITRACES	ITRACES Controls output of photon trajectories. EQ.0: do not write trajectory information

VARIABLE	DESCRIPTION
	EQ.1: write trajectory information. This file becomes large quickly and is only useful for debugging.
IRESTART	IRESTART should be set either to 1 or 0. If IRESTART is set to 1 then LS-DYNA restarts the exchange factor solver. If IRESTART is set to 1 and a .crh file exists, the Monte Carlo solver will pick up where it left off prior to a crash. If there is a .nij file but no .crh file, then LS-DYNA will recycle the results of the previous exchange factor running emitting more photons to increase accuracy.

*ELEMENT

The element cards in this section are defined in alphabetical order:

***ELEMENT_BEAM_**{*OPTION*}_{*OPTION*}

***ELEMENT_DIRECT_MATRIX_INPUT**

***ELEMENT_DISCRETE_**{*OPTION*}

***ELEMENT_INERTIA_**{*OPTION*}

***ELEMENT_MASS**

***ELEMENT_MASS_PART**

***ELEMENT_PLOTEL**

***ELEMENT_SEATBELT**

***ELEMENT_SEATBELT_ACCELEROMETER**

***ELEMENT_SEATBELT_PRETENSIONER**

***ELEMENT_SEATBELT_RETRACTOR**

***ELEMENT_SEATBELT_SENSOR**

***ELEMENT_SEATBELT_SLIPRING**

***ELEMENT_SHELL_**{*OPTION*}

***ELEMENT_SHELL_SOURCE_SINK**

***ELEMENT_SOLID_**{*OPTION*}

***ELEMENT_SPH**

***ELEMENT_TRIM**

***ELEMENT_TSHELL**

The ordering of the element cards in the input file is completely arbitrary. An arbitrary number of element blocks can be defined preceded by a keyword control card.

***ELEMENT_BEAM_**{*OPTION*}**_**{*OPTION*}

Available options include:

<BLANK>

THICKNESS, SCALAR, SCALR or SECTION

PID

OFFSET

ORIENTATION

WARPAGE

Purpose: Define two node elements including 3D beams, trusses, 2D axisymmetric shells, and 2D plane strain beam elements. The type of the element and its formulation is specified through the part ID (see *PART) and the section ID (see *SECTION_BEAM).

Two alternative methods are available for defining the cross sectional property data. The THICKNESS and SECTION options are provided for the user to override the *SECTION_ BEAM data which is taken as the default if the THICKNESS or SECTION option is not used. . The SECTION option applies only to resultant beams (ELFORM.eq.2 on *SECTION_BEAM). End release conditions are imposed using constraint equations, and caution must be used with this option as discussed in remark 2 below. The SCALAR/SCALR options applies only to material model type 146, *MAT_1DOF_GENERALIZED_SPRING.

The PID option is used by the type 9 spot weld element only and is ignored for all other beam types. When the PID option is active an additional card is read that gives two part ID's that are tied by the spot weld element. If the PID option is inactive for the type 9 element the nodal points of the spot weld are located to the two nearest master segments. In either case, *CONTACT_SPOTWELD must be defined with the spot weld beam part as slave and the shell parts (including parts PID1 and PID2) as master. The surface of each segment should project to the other and in the most typical case the node defining the weld, assuming only one node is used, should lie in the middle; however, this is not a requirement. Note that with the spot weld elements only one node is needed to define the weld, and two nodes are optional.

The options ORIENTATION and OFFSET are not available for discrete beam elements.

Card Format (10I8)

Card	1	2	3	4	5	6	7	8	9	10
Variable	EID	PID	N_1	N_2	N_3	RT1	RR1	RT2	RR2	LOCAL
Type	I	I	I	I	I	I	I	I	I	I
Default	none	None	none	none	none	0	0	0	0	2
Remarks					1	2,3	2,3	2,3	2,3	2,3

Optional Card (Required if THICKNESS is specified after the keyword)

Card	1	2	3	4	5
Variable	PARM1	PARM2	PARM3	PARM4	PARM5
Type	F	F	F	F	F
Remarks	4	5	5	5	6

Optional Card (Required if SECTION is specified after the keyword)

Card	1	2	3	4	5	6	7	8
Variable	STYPE	D1	D2	D3	D4	D5	D6	
Type	A	F	F	F	F	F	F	
Remarks								

Optional Card (Required if SCALAR is specified after the keyword)

Card	1	2	3	4	5	6	7	8	9	10
Variable	VOL		INER		CID		DOFN1		DOFN2	
Type	F		F		F		F		F	

Optional Card (Required if SCALR is specified after the keyword)

Card	1	2	3	4	5	6	7	8	9	10
Variable	VOL		INER		CID1		CID2		DOFNS	
Type	F		F		F		F		F	

Optional Card (Required if PID is specified after the keyword)

Card	1	2	3	4	5	6	7	8	9	10
Variable	PID1	PID2								
Type	I	I								
Default	none	none								
Remarks										

Optional Card (Required if OFFSET is specified after the keyword)

Card	1	2	3	4	5	6	7	8
Variable	WX1	WY1	WZ1	WX2	WY2	WZ2		
Type	F	F	F	F	F	F		
Default	0.0	0.0	0.0	0.0	0.0	0.0		
Remarks	8	8	8	8	8	8		

Optional Card (Required if ORIENTATION is specified after the keyword)

Card	1	2	3	4	5	6	7	8
Variable	VX	VY	VZ					
Type	F	F	F					
Default	0.0	0.0	0.0					
Remarks								

Optional Card (Required if WARPAGE is specified after the keyword)

Card	1	2	3	4	5	6	7	8
Variable	SN_1	SN_2						
Type	I	I						
Default	none	none						
Remarks								

VARIABLE	DESCRIPTION
EID	Element ID. A unique ID is generally required, i.e., EID must be different from the element ID's also defined under *ELEMENT_ DISCRETE and *ELEMENT_SEATBELT. If the parameter, BEAM, is set to 1 on the keyword input for *DATABASE_BINARY_D3PLOT, the null beams used for visualization are not created for the latter two types, and the ID's used for the discrete elements and the seatbelt elements can be identical to those defined here.
PID	Part ID, see *PART.
N1	Nodal point (end) 1.
N2	Nodal point (end) 2. This node is optional for the spot weld, beam type 9, since if it not defined it will be created automatically and given a non-conflicting nodal point ID. Nodes N1 and N2 are automatically positioned for the spot weld beam element. For the zero length discrete beam elements where one end is attached to ground, set N2=-N1. In this case, a fully constrained nodal point will be created with a unique ID for node N2.
N3	Nodal point 3 for orientation. The third node, N3, is optional for beam types 3, 6, 7, 8, and 9 if the latter, type 9, has a circular cross section. The third node is used for the discrete beam, type 6, if and only if SCOOR is set to 2.0 in the *SECTION_BEAM input, but even in this case it is optional. An orientation vector can be defined directly by using the option, ORIENTATION. In this case N3 can be defined as zero.
RT1, RT2	Release conditions for translations at nodes N1 and N2, respectively: EQ.0: no translational degrees-of-freedom are released EQ.1: x-translational degree-of-freedom EQ.2: y-translational degree-of-freedom EQ.3: z-translational degree-of-freedom EQ.4: x and y-translational degrees-of-freedom EQ.5: y and z-translational degrees-of-freedom EQ.6: z and x-translational degrees-of-freedom EQ.7: x, y, and z-translational degrees-of-freedom (3DOF) This option does not apply to the spot weld, beam type 9.
RR1, RR2	Release conditions for rotations at nodes N1 and N2, respectively: EQ.0: no rotational degrees-of-freedom are released EQ.1: x-rotational degree-of-freedom EQ.2: y-rotational degree-of-freedom EQ.3: z-rotational degree-of-freedom EQ.4: x and y-rotational degrees-of-freedom EQ.5: y and z-rotational degrees-of-freedom

VARIABLE	DESCRIPTION
	EQ.6: z and x-rotational degrees-of-freedom EQ.7: x, y, and z-rotational degrees-of-freedom (3DOF) This option does not apply to the spot weld, beam type 9.
LOCAL	Coordinate system option: EQ.1: global coordinate system EQ.2: local coordinate system (default)
PARM1	Based on beam type: Type.EQ.1: beam thickness, s direction at node 1 Type.EQ.2: area Type.EQ.3: area Type.EQ.4: beam thickness, s direction at node 1 Type.EQ.5: beam thickness, s direction at node 1 Type.EQ.6: volume, see description for VOL below. Type.EQ.7: beam thickness, s direction at node 1 Type.EQ.8: beam thickness, s direction at node 1 Type.EQ.9: beam thickness, s direction at node 1
PARM2	Based on beam type: Type.EQ.1: beam thickness, s direction at node 2 Type.EQ.2: I_{ss} Type.EQ.3: ramp-up time for dynamic relaxation Type.EQ.4: beam thickness, s direction at node 2 Type.EQ.5: beam thickness, s direction at node 2 Type.EQ.6: geometric inertia Type.EQ.6: Inertia, see description for INER below. Type.EQ.7: beam thickness, s direction at node 2 Type.EQ.8: beam thickness, s direction at node 2 Type.EQ.9: beam thickness, s direction at node 2
PARM3	Based on beam type: Type.EQ.1: beam thickness, t direction at node 1 Type.EQ.2: I_{tt} Type.EQ.3: initial stress for dynamic relaxation Type.EQ.4: beam thickness, t direction at node 1 Type.EQ.5: beam thickness, t direction at node 1 Type.EQ.6: local coordinate ID Type.EQ.7: not used. Type.EQ.8: not used. Type.EQ.9: beam thickness, t direction at node 1
PARM4	Based on beam type: Type.EQ.1: beam thickness, t direction at node 2 Type.EQ.2: I_{rr} Type.EQ.3: not used

VARIABLE	DESCRIPTION
	Type.EQ.4: beam thickness, t direction at node 2 Type.EQ.5: beam thickness, t direction at node 2 Type.EQ.6: area Type.EQ.7: not used. Type.EQ.8: not used. Type.EQ.9: beam thickness, t direction at node 2
PARM5	Based on beam type: Type.EQ.1: not used Type.EQ.2: shear area Type.EQ.3: not used Type.EQ.4: not used Type.EQ.5: not used Type.EQ.6: offset Type.EQ.7: not used. Type.EQ.8: not used. Type.EQ.9: print flag to SWFORC file. The default is taken from the SECTION_BEAM input. To override set PARM5 to 1.0 to suppress printing, and to 2.0 to print.
STYPE	Section type (A format) of resultant beam, see Figure 29.1: EQ.SECTION_01: I-shape EQ.SECTION_02: Channel EQ.SECTION_03: L-shape EQ.SECTION_04: T-shape EQ.SECTION_05: Tubular box EQ.SECTION_06: Z-shape EQ.SECTION_07: Trapezoidal EQ.SECTION_08: Circular EQ.SECTION_09: Tubular EQ.SECTION_10: I-shape1 EQ.SECTION_11: Solid box EQ.SECTION_12: Cross EQ.SECTION_13: H-shape EQ.SECTION_14: T-shape1 EQ.SECTION_15: I-shape2 EQ.SECTION_16: Channel1 EQ.SECTION_17: Channel2 EQ.SECTION_18: T-shape2 EQ.SECTION_19: Box-shape1 EQ.SECTION_20: Hexagon EQ.SECTION_21: Hat-shape EQ.SECTION_22: Hat-shape1
D1-D6	Input parameters for section option using STYPE above.
PID1	Optional part ID for spot weld element type 9.
PID2	Optional part ID for spot weld element type 9.
VOL	Volume of discrete beam and scalar beam. If the mass density of the material model for the discrete beam is set to unity, the magnitude of the lumped mass can be defined here instead. This lumped mass is partitioned to the two nodes of the beam element. The translational time step size for the type 6 beam is dependent on the volume, mass density, and the translational stiffness values, so it is important to define this parameter. Defining the volume is also essential for mass scaling if the type 6 beam controls the time step size.

VARIABLE	DESCRIPTION
INER	Mass moment of inertia for the six degree of freedom discrete beam and scalar beam. This lumped inertia is partitioned to the two nodes of the beam element. The rotational time step size for the type 6 beam is dependent on the lumped inertia and the rotational stiffness values, so it is important to define this parameter if the rotational springs are active. Defining the rotational inertia is also essential for mass scaling if the type 6 beam rotational stiffness controls the time step size.
CID	Coordinate system ID for orientation, material type 146, see *DEFINE_ COORDINATE_SYSTEM. If CID=0, a default coordinate system is defined in the global system.
DOFN1	Active degree-of-freedom at node 1, a number between 1 to 6 where 1, 2, and 3 are the x, y, and z-translations and 4, 5, and 6 are the x, y, and z-rotations. This degree-of-freedom acts in the local system given by CID above. This input applies to material model type 146.
DOFN2	Active degree-of-freedom at node 2, a number between 1 to 6. This degree-of-freedom acts in the local system given by CID above. This input applies to material model type 146.
CID1	Coordinate system ID at node 1 for orientation, material type 146, see *DEFINE_ COORDINATE_SYSTEM. If CID1=0, a default coordinate system is defined in the global system.
CID2	Coordinate system ID at node 2 for orientation, material type 146, see *DEFINE_ COORDINATE_SYSTEM. If CID2=0, a default coordinate system is defined in the global system.
DOFNS	Active degrees-of-freedom at node 1 and node 2. A two-digit number, the first for node 1 and the second for node 2, between 11 to 66 is expected where 1, 2, and 3 are the x, y, and z-translations and 4, 5, and 6 are the x, y, and z-rotations. These degrees-of-freedom acts in the local system given by CID1 and CID2 above. This input applies to material model type 146. If DOFNS=12 the node one has an x-translation and node 2 has a y translation.
WX1-WZ1	Offset vector at nodal point N1. See Remark 8.
WX2-WZ2	Offset vector at nodal point N2. Set Remark 8.
VX,VY, VZ	Orientation vector at node N1. In this case the orientation nodal point N3, is defined as zero.
SN1	Scalar nodal point (end) 1. This node is required for the WARPAGE option.
SN2	Scalar nodal point (end) 2. This node is required for the WARPAGE option.

Remarks:

1. A plane through N_1, N_2, and N_3 defines the orientation of the principal r-s plane of the beam, see Figure 14.1.

2. This option applies to all three-dimensional beam elements. The released degrees-of-freedom can be either global, or local relative to the local beam coordinate system, see Figure 14.1. A local coordinate system is stored for each node of the beam element and the orientation of the local coordinate systems rotates with the node. To properly track the response, the nodal points with a released resultant are automatically replaced with new nodes to accommodate the added degrees-of-freedom. Then constraint equations are used to join the nodal points together with the proper release conditions imposed. **Consequently, nodal points which belong to beam elements which have release conditions applied cannot be subjected to other constraints such as applied displacement /velocity/acceleration boundary conditions, nodal rigid bodies, nodal constraint sets, or any of the constraint type contact definitions.** Force type loading conditions and penalty based contact algorithms may be used with this option.

3. Please note that this option may lead to nonphysical constraints if the translational degrees-of-freedom are released, but this should not be a problem if the displacements are infinitesimal.

4. If the second card is not defined for the resultant beam or if the area, A, is not defined the properties are taken from the cross section cards, see *SECTION_BEAM.

5. Do not define for discrete beams (beam type 6), see *SECTION_BEAM.

6. Define for resultant beam elements only, see *SECTION_BEAM.

7. The stress resultants are output in local coordinate system for the beam. Stress information is optional and is also output in the local system for the beam.

8. Beam offsets are sometimes necessary for correctly modeling beams that act compositely with other elements such as shells or other beams. When the OFFSET option is specified, global X, Y, and Z components of two offset vectors are given, one vector for each of the two beam nodes. The offset vector extends from the beam node (N1 or N2) to the reference axis of the beam. The beam reference axis lies at the origin of the local s and t axes, i.e., halfway between the outermost surfaces of the beam cross-section. Note that for cross-sections that are not doubly symmetric, e.g, a T-section, the reference axis does not pass through the centroid of the cross-section. Beam offsets can only be defined for Hughes-Liu beams (ELFORM=1).

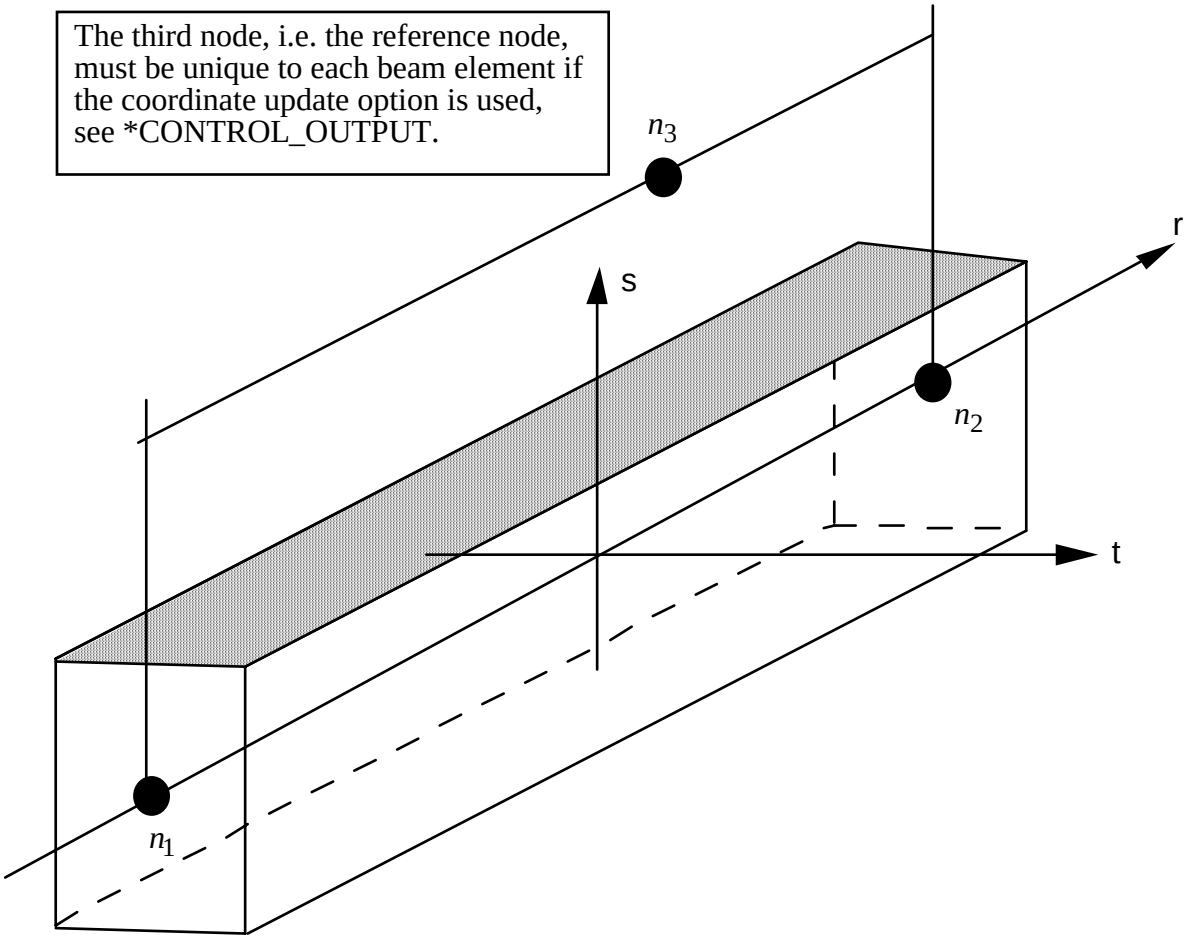


Figure 14.1. LS-DYNA beam elements. Node n_3 determines the initial orientation of the cross section.

***ELEMENT_DIRECT_MATRIX_INPUT_**{*OPTION*}

Available options include:

<BLANK>

BINARY

Purpose: Define an element consisting of mass, damping, stiffness, and inertia matrices in a specified file which follows the format used in the direct matrix input, DMIG, of NASTRAN. The supported format is the type 6 symmetric matrix in real double precision. LS-DYNA supports both the standard and the extended precision formats. The binary format from *CONTROL_IMPLICIT_MODES or *CONTROL_IMPLICIT_STATIC_CONDENSATION is another input option. The mass and stiffness matrices are required. The inertia matrix is required when using *LOAD_BODY_OPTION to correctly compute the action of a prescribed base acceleration on the superelement, otherwise the inertia matrix is unused. The damping matrix is optional. The combination of these matrices is referred to as a superelement. Three input cards are required for each superelement.

The degrees-of-freedom for this superelement may consist of generalized coordinates as well was nodal point quantities. Degrees-of-freedom, defined using *NODE input, are called attachment nodes. Only attachment nodes are included in the output to the ASCII and binary databases.

The matrices for a given superelement can be of different order. However, the explicit integration scheme requires the inversion of the union of the element mass matrix and nodal masses associated with attachment nodes. Any degree of freedom included in the other (stiffness, damping, inertia) matrices but without nonzero columns in the combined mass matrix will be viewed as massless and constrained not to move. After deleting zero rows and columns the combined mass matrix is required to be positive definite.

The inertia matrix is required to have 3 columns which corresponds to the 3 global coordinates. It is used to compute the forces acting on the superelement by multiplying the inertia matrix times the gravitational acceleration specified via *LOAD_BODY_OPTION.

There is no assumption made on the order of the matrices nor the sparse matrix structure of the element matrices except that they are symmetric and the combined mass matrix is invertible as described above.

Multiple elements may be input using *ELEMENT_DIRECT_MATRIX_INPUT. They may share attachment nodes with other direct matrix input elements. Only *BOUNDARY_ PRESCRIBED_MOTION and global constraints imposed via*NODE or *BOUNDARY_SPC on attachment nodes can be applied in explicit applications. Implicit applications can have additional constraints on attachment nodes.

Card Format (I10)

Card 1	1	2	3	4	5	6	7	8
Variable	EID	IFRMT						
Type	I	0						

Card Format (A80)

Card 2

Variable	FILENAME
Type	C

Card Format (4A10)

Card 3	1	2	3	4	5	6	7	8
Variable	MASS	DAMP	STIF	INERT				
Type	C	C	C	C				

VARIABLE	DESCRIPTION
EID	Super element ID.
IFRMT	Format: EQ.0: standard format NE.0: extended precision format
MASS	Name of mass matrix in the file defined by FILENAME. This filename should be no more than eight characters to be compatible with NASTRAN.
DAMP	Name of damping matrix in the file defined by FILENAME. This filename should be no more than eight characters to be compatible with NASTRAN.

VARIABLE	DESCRIPTION
STIF	Name of stiffness matrix in the file defined by FILENAME. This filename should be no more than eight characters to be compatible with NASTRAN.
INERT	Name of inertia matrix in the file defined by FILENAME. This filename should be no more than eight characters to be compatible with NASTRAN. This file must be present when *LOAD_BODY is used to put gravitational forces on the model.

***ELEMENT_DISCRETE_**{*OPTION*}

Available options include:

<BLANK>

LCO

Purpose: Define a discrete (spring or damper) element between two nodes or a node and ground. An option, LCO, is available for using a load curve(s) to initialize the offset to avoid the excitation of numerical noise that can sometimes result with an instantaneous imposition of the offset. This can be done using a single curve at the start of the calculation or two curves where the second is used during dynamic relaxation prior to beginning the transient part. In the latter case, the first curve would simply specify the offset as constant during time. If the LCO option is active, a second card is read. It is recommended that beam type 6, see *ELEMENT_BEAM and SECTION_BEAM, be used whenever possible, especially if orientation is specified. The latter option tends to be more accurate and cost effective. The *ELEMENT_DISCRETE option is no longer being developed and extended

Note: The discrete elements enter into the time step calculations. Care must be taken to ensure that the nodal masses connected by the springs and dampers are defined and unrealistically high stiffness and damping values must be avoided. **All rotations are in radians**.

Card Format (5I8,E16.0,I8,E16.0)

Card	1	2	3	4	5	6	7	8	9	10
Variable	EID	PID	N1	N2	VID	S		PF	OFFSET	
Type	I	I	I	I	I	F		I	F	
Default	none	none	none	none	0	1.		0	0	

Card Format (2I10) Define this card if and only if the option LCO is active

Card 2	1	2	3	4	5	6	7	8
Variable	LCID	LCIDDR						
Type	I	I						
Default	none	none						

VARIABLE	DESCRIPTION
EID	Element ID. A unique number is required. Since null beams are created for visualization, this element ID should not be identical to element ID's defined for ELEMENT_BEAM and ELEMENT_ SEATBELT.
PID	Part ID, see *PART.
N1	Nodal point 1.
N2	Nodal point 2. If zero, the spring/damper connects node N1 to ground.
VID	Orientation option. The orientation option should be used cautiously since forces, which are generated as the nodal points displace, are not orthogonal to rigid body rotation unless the nodes are coincident.. The type 6, 3D beam element, is recommended when orientation is required with the absolute value of the parameter SCOOR set to 2 or 3, since this option avoids rotational constraints. EQ.0: the spring/damper acts along the axis from node N1 to N2, NE.0: the spring/damper acts along the axis defined by the orientation vector, VID defined in the *DEFINE_SD_ORIENTATION section.

VARIABLE	DESCRIPTION
S	Scale factor on forces.
PF	Print flag: EQ.0: forces are printed in DEFORC file, (see *DATABASE_*OPTION*), EQ.1: forces are not printed DEFORC file.
OFFSET	Initial offset. The initial offset is a displacement or rotation at time zero. For example, a positive offset on a translational spring will lead to a tensile force being developed at time zero. Ignore this input if LCID is defined below.
LCID	Load curve ID defining the initial OFFSET as a function of time. Positive offsets correspond to tensile forces, and, likewise negative offset result incompressive forces.
LCIDDR	Load curve ID defining OFFSET as a function of time during the dynamic relaxation phase.

***ELEMENT_INERTIA_**{*OPTION*}

Available options include:

<BLANK>

OFFSET

to allow the lumped mass and inertia tensor to be offset from the nodal point. The nodal point can belong to either a deformable or rigid node.

Purpose: Define a lumped inertia element assigned to a nodal point.

Card Format (3I8)

Card	1	2	3	4	5	6	7	8	9	10
Variable	EID	NID	CSID							
Type	I	I	I							
Default	none	none	none							
Remarks			1							

Card Format (7E10.0)

Card	1	2	3	4	5	6	7	8
Variable	IXX	IXY	IXZ	IYY	IYZ	IZZ	MASS	
Type	F	F	F	F	F	F	F	
Default	0.0	0.0	0.0	0.0	0.0	0.0	0.0	
Remarks		2	2		2			

Define if and only if the OFFSET option is active. Card Format (3E10.0)

Card	1	2	3	4	5	6	7	8
Variable	X-OFF	Y-OFF	Z-OFF					
Type	F	F	F					
Default	0.	0.	0.					
Remarks		2	2					

VARIABLE	DESCRIPTION
EID	Element ID. A unique number must be used.
NID	Node ID. Node to which the mass is assigned.
CSID	Coordinate set ID EQ.0: global inertia tensor GE.1: principal moments of inertias with orientation vectors defined by Coordinate set CSID. See *DEFINE_COORDINATE_SYSTEM and *DEFINE_COORDINATE_VECTOR.
IXX	XX component of inertia tensor.
IXY	XY component of inertia tensor.
IXZ	XZ component of inertia tensor.
IYY	YY component of inertia tensor.
IYZ	YZ component of inertia tensor.
IZZ	ZZ component of inertia tensor.
MASS	Lumped mass
X-OFF	x-offset from nodal point.
Y-OFF	y-offset from nodal point.
Z-OFF	z-offset from nodal point.

Remarks:

1. The coordinate system cannot be defined for this element using the option *DEFINE_COORDINATE_ NODE.

2. If CSID is defined then IXY, IXZ and IYZ are set to zero. The nodal inertia tensor must be positive definite, i.e., its determinant must be greater than zero, since its inverse is required. This check is done after the nodal inertia is added to the defined inertia tensor.

***ELEMENT_MASS**_{*OPTION*}

Available options include:

<BLANK>

NODE_SET

Purpose: Define a lumped mass element assigned to a nodal point or equally distributed to the nodes of a node set.

(Note: NODE_SET option is available starting with the R3 release of Version 971.)

Card Format (2I8,E16.0)

Card	1	2	3	4	5	6	7	8	9	10
Variable	EID	ID	MASS		PID					
Type	I	I	F		I					
Default	none	None	0.		none					

VARIABLE	DESCRIPTION
EID	Element ID. A unique number is recommended. The nodes in a node set share the same element ID.
ID	Node ID or node set ID if the NODE_SET option is active. This is the node or node set to which the mass is assigned.
MASS	Mass value. When the NODE_SET option is active, the mass is equally distributed to all nodes in a node set.
PID	Part ID. This input is optional.

***ELEMENT_MASS_MATRIX_**{*OPTION*}

Available options include:

<BLANK>

NODE_SET

Purpose: Define a 6x6 symmetric nodal mass matrix assigned to a nodal point or each node within a node set

Card Format (3I10)

Card 1	1	2	3	4	5	6	7	8
Variable	EID	ID	CID					
Type	I	I	I					
Default	none	none	0					

Card 2

Variable	M11	M21	M22	M31	M32	M33	M41	
Type	F	F	F	F	F	F	F	
Default	0.0	0.0	0.0	0.0	0.0	0.0	0.0	

Card 3

Variable	M42	M43	M44	M51	M52	M53	M54	
Type	F	F	F	F	F	F	F	
Default	0.0	0.0	0.0	0.0	0.0	0.0	0.0	

Card 4	1	2	3	4	5	6	7	8
Variable	M55	M61	M62	M63	M64	M65	M66	
Type	F	F	F	F	F	F	F	
Default	0.0	0.0	0.0	0.0	0.0	0.0	0.0	

VARIABLE	DESCRIPTION
EID	Element ID. A unique number is recommended. The nodes in a node set share the same element ID.
ID	Node ID or node set ID if the NODE_SET option is active. This is the node or node set to which the mass is assigned.
CID	Local coordinate ID which defines the orientation of the mass matrix
MIJ	The IJ^{th} term of the symmetric mass matrix. The lower triangular part of the matrix is defined.

***ELEMENT_MASS_PART_**{*OPTION*}

Available options include:

<BLANK>

SET

Purpose: Define additional non-structural mass to be distributed by an area weighted distribution to all nodes of a given part ID. As an option, the total mass can be defined and the additional non-structural mass is computed. This option applies to all part ID's defined by shell and solid elements.

Card Format (I8,2E16.0,I16)

Card	1	2	3	4	5	6	7	8	9	10
Variable	ID	ADDMASS		FINMASS		LCID				
Type	I	F		F		I				
Default	none	0.		0.		0				

VARIABLE	DESCRIPTION
ID	Part or part set ID if the SET option is active. A unique number must be used.
ADDMASS	Added translational mass to be distributed to the nodes of the part ID or part set ID. Set to zero if TOTMASS is nonzero. Since the additional mass is not included in the time step calculation of the elements in the PID or SID, ADDMASS must be greater than zero.
FINMASS	Final translational mass of the part ID or part set ID. The total mass of the PID or SID is computed and subtracted from the final mass of the part or part set to obtain the added translational mass, which must exceed zero. Set FINMASS to zero if ADDMASS is nonzero. FINMASS is available in the R3 release of version 971.
LCID	Optional load curve ID to scale the added mass at time=0. This curve defines the scale factor as a function versus time. The curve must start at unity at t=0. This option applies to deformable bodies only.

*ELEMENT_PLOTEL

Purpose: Define a null beam element for visualization.

Card Format (3I8)

Card	1	2	3	4	5	6	7	8	9	10
Variable	EID	N_1	N_2							
Type	I	I	I							
Default	none	none	None							
Remarks	1									

VARIABLE	DESCRIPTION
EID	Element ID. A unique number must be used.
N1	Nodal point (end) 1.
N2	Nodal point (end) 2.

Remarks:

1. Part ID, 10000000, is assigned to PLOTEL elements.
2. PLOTEL element ID's must be unique with respect to other beam elements.

***ELEMENT_SEATBELT**

Purpose: Define a seat belt element.

Card Format (5I8,E16.0,2I8)

Card	1	2	3	4	5	6	7	8	9	10
Variable	EID	PID	N_1	N_2	SBRID	SLEN		N_3	N_4	
Type	I	I	I	I	I	F		I	I	
Default	none	none	none	none	none	0.0		0	0	
Remarks					1	2		3		

VARIABLE	DESCRIPTION
EID	Element ID. A unique number is required. Since null beams are created for visualization, this element ID should not be identical to element ID's defined for ELEMENT_BEAM and ELEMENT_ DISCRETE.
PID	Part ID
N1	Node 1 ID
N2	Node 2 ID
SBRID	Retractor ID, see *ELEMENT_SEATBELT_RETRACTOR.
SLEN	Initial slack length
N3	Optional node 3 ID. When N3>0 and N4>0, the elements becomes a shell seat belt element. The thickness of the shell seatbelt is defined in *SECTION_SHELL, not in *SECTION_SEATBELT. The shell-type seatbelt must be of a rectangular shape as shown in Figure 14.2 and contained in a logically regular mesh.
N4	Node 4 ID, which is required if and only if N3 is defined.

Remarks:

1. The retractor ID should be defined only if the element is initially **inside** a retractor, see *ELEMENT_SEATBELT_RETRACTOR.

2. Belt elements are single degree of freedom elements connecting two nodes. When the strain in an element is positive (i.e. the current length is greater then the unstretched length), a tension force is calculated from the material characteristics and is applied along the current axis of the element to oppose further stretching. The unstretched length of the belt is taken as the initial distance between the two nodes defining the position of the element plus the initial slack length.

3. Seatbelt shell elements are a new feature in version 971 and must be used with caution. The seatbelt shells distribute the loading on the surface of the dummy more realistically than the two node belt elements. For the seatbelt shells to work with sliprings and retractors it is necessary to use a logically regular mesh of quadrilateral elements. *A seatbelt defined by a part ID must not be disjoint.*

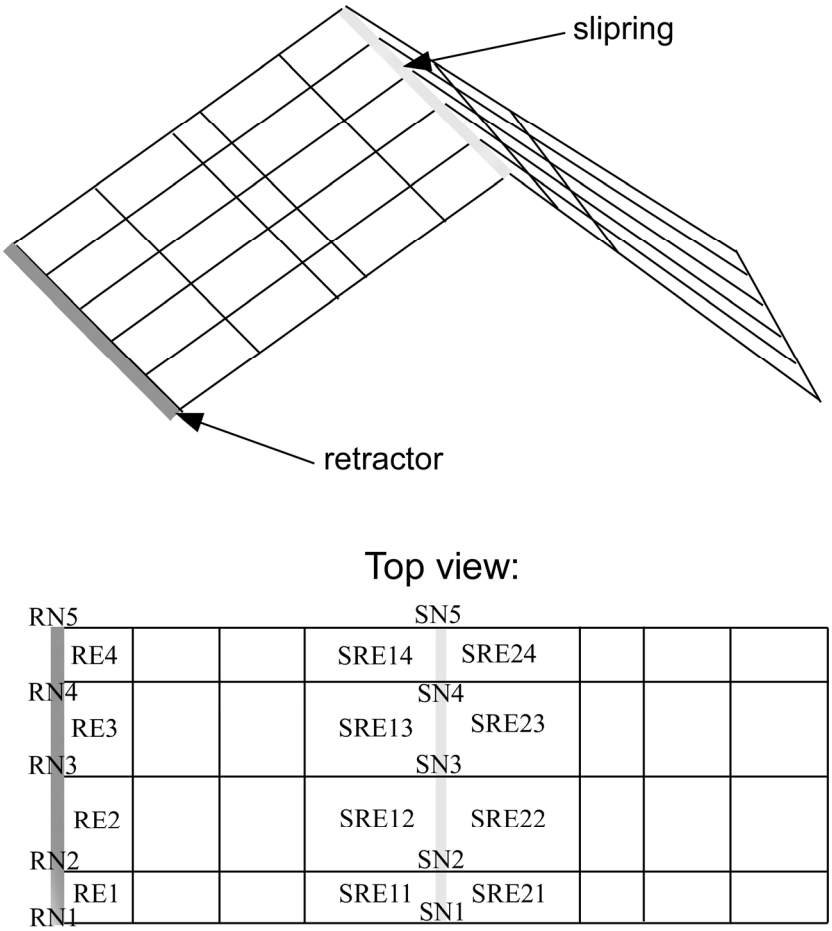


Figure 14.2. Definition of seatbelt shell elements. The ordering of the nodes and elements are important for seatbelt shells. See the input descriptions for SECTION_SHELL, ELEMENT_SEATBELT_RETRACTOR and ELEMENT_SEATBELT_SLIPRING.

*ELEMENT_SEATBELT_ACCELEROMETER

Purpose: Define seat belt accelerometer. The accelerometer is fixed to a rigid body containing the three nodes defined below. Whenever computed accelerations are compared to experimental results or whenever computed accelerations are compared between different runs, accelerometers are essential. Raw nodal accelerations contain considerable numerical noise and their comparisons are generally meaningless and, therefore, misleading.

Card	1	2	3	4	5	6	7	8
Variable	SBACID	NID1	NID2	NID3	IGRAV	INTOPT	MASS	
Type	I	I	I	I	I	I	F	
Default	0	0	0	0	0	0	0.	
Remarks								

VARIABLE	DESCRIPTION
SBACID	Accelerometer ID. A unique number must be used.
NID1	Node 1 ID
NID2	Node 2 ID
NID3	Node 3 ID
IGRAV	Gravitational accelerations due to body force loads. EQ.-6: Z and X components removed from acceleration output EQ.-5 Y and Z components removed from acceleration output EQ.-4: X and Y components removed from acceleration output EQ.-3: Z component removed from acceleration output EQ.-2: Y component removed from acceleration output EQ.-1: X component removed from acceleration output EQ. 0: all components included in acceleration output EQ. 1: all components removed from acceleration output
INTOPT	Integration option. If the accelerometer undergoes rigid body translation without rotation this option has no effect; however, if rotation occurs, option 1 may provide better agreement with the output of the accelerometer. EQ.0: velocities are integrated from the global accelerations and transformed into the local system of the accelerometer

VARIABLE	DESCRIPTION
	EQ.1: velocities are integrated directly from the local accelerations of the accelerometer.
MASS	Optional added mass for accelerometer. This mass is equally distributed to nodal points NID1, NID2, and NID3. This option avoids the need to use the *ELEMENT_MASS keyword input if additional mass is required.

Remarks:

The presence of the accelerometer means that the accelerations and velocities of node 1 will be output to **all** output files in local instead of global coordinates.

The local coordinate system is defined by the three nodes as follows:

- local **x** from node 1 to node 2,
- local **z** perpendicular to the plane containing nodes, 1, 2, and 3 (**z** = **x** × **a)**, where **a** is from node 1 to node 3),
- local **y** = **z** × **x.**

The three nodes should all be part of the same rigid body. The local axis then rotates with the body.

*ELEMENT_SEATBELT_PRETENSIONER

Purpose: Define seat belt pretensioner. A combination with sensors and retractors is also possible.

Card 1 2 3 4 5 6 7 8

Variable	SBPRID	SBPRTY	SBSID1	SBSID2	SBSID3	SBSID4		
Type	I	I	I	I	I	I		
Default	0	0	0	0	0	0		
Remarks			1					

Card

Variable	SBRID	TIME	PTLCID	LMTFRC				
Type	I	F	I	F				
Default	0	0.0	0	0				
Remarks								

VARIABLE **DESCRIPTION**

SBPRID Pretensioner ID. A unique number has to be used.

SBPRTY Pretensioner type (see Remark 2 below):
EQ.1: pyrotechnic retractor with force limits,
EQ.2: pre-loaded spring becomes active,
EQ.3: lock spring removed,
EQ.4: force versus time retractor.
EQ.5: pyrotechnic retractor (old type in version 950) but with optional force limiter, LMTFRC.
EQ.6: combination of types 4 and 5 as described in the notes below.
EQ.7: independent pretensioner/retractor.

VARIABLE	DESCRIPTION
SBSID1	Sensor 1, see *ELEMENT_SEATBELT_SENSOR.
SBSID2	Sensor 2, see *ELEMENT_SEATBELT_SENSOR.
SBSID3	Sensor 3, see *ELEMENT_SEATBELT_SENSOR.
SBSID4	Sensor 4, see *ELEMENT_SEATBELT_SENSOR.
SBRID	Retractor number (SBPRTY = 1, 4, 5, or 6) or spring element number (SBPRTY = 2 or 3).
TIME	Time between sensor triggering and pretensioner acting.
PTLCID	Load curve for pretensioner (Time after activation, Pull-in) (SBPRTY = 1, 4, 5 or 6).
LMTFRC	Optional limiting force for retractor type 5. If zero, this option is ignored.

Remarks:

1. At least one sensor should be defined.

   Pretensioners allow modeling of seven types of active devices which tighten the belt during the initial stages of a crash. Types 1 and 5 represent a pyrotechnic device which spins the spool of a retractor, causing the belt to be reeled in. The user defines a pull-in versus time curve which applies once the pretensioner activates. Types 2 and 3 represent preloaded springs or torsion bars which move the buckle when released. The pretensioner is associated with any type of spring element including rotational. Note that the preloaded spring, locking spring and any restraints on the motion of the associated nodes are defined in the normal way; the action of the pretensioner is merely to cancel the force in one spring until (or after) it fires. With the second type, the force in the spring element is canceled out until the pretensioner is activated. In this case the spring in question is normally a stiff, linear spring which acts as a locking mechanism, preventing motion of the seat belt buckle relative to the vehicle. A preloaded spring is defined in parallel with the locking spring. This type avoids the problem of the buckle being free to 'drift' before the pretensioner is activated. Types 4, 6, and 7, force types, are described below.

   To activate the pretensioner, the following sequence of events must occur:

   1. Any one of up to four sensors must be triggered.

   2. Then a user-defined time delay occurs.

   3. Then the pretensioner acts.

2. In the 950 version of LS-DYNA, there are three types of seatbelt pretensioners that can be simulated. Types 2 and 3 are simple triggers for activating or deactivating springs, which

then pull on the buckle. No changes have been made to these, and they are not discussed here. The type 1 pretensioner is intended to simulate a pyrotechnic retractor. The user inputs a load curve describing the pull-in of the pretensioner as a function of time. This pretensioner type interacts with the retractor, forcing it to pull in the amount of belt indicated. It works well, and does exactly what it says it will do, but it can be difficult to use in practice. The reason for this is that it has no regard for the forces being exerted on the belt. If a pull-in of 20mm is specified at a particular time, then 20mm of belt will be pulled in, even if this results in unrealistic forces in the seatbelt. Furthermore, there was no explicit way to turn this pretensioner off. Once defined, it overrode the retractor completely, and the amount of belt passing into or out of the retractor depended solely on the load curve specified.

In the 970 version of LS-DYNA, the behavior of the type 1 pretensioner was changed due to user feedback regarding these shortcomings. The behavior now is fundamentally simpler, though a bit confusing to explain. Each retractor has a loading (and optional unloading) curve that describes the force on the belt element as a function of the amount of belt that has been pulled out of the retractor since the retractor locked. The new type 1 pretensioner acts as a shift of this retractor load curve. An example will make this clear. Suppose at a particular time that 5mm of belt material has left the retractor. The retractor will respond with a force corresponding to 5mm pull-out on it's loading curve. But suppose this retractor has a type 1 pretensioner defined, and at this instant of time the pretensioner specifies a pull-in of 20mm. The retractor will then respond with a force that corresponds to (5mm + 20mm) on it's loading curve. This results in a much larger force. The effect can be that belt material will be pulled in, but unlike in the 950 version, there is no guarantee. The benefit of this implementation is that the force vs. pull-in load curve for the retractor is followed and no unrealistic forces are generated. Still, it may be difficult to produce realistic models using this option, so two new types of pretensioners have been added. These are available in 970 versions 1300 and later.

The type 4 pretensioner takes a force vs. time curve, See Figure 14.3. Each time step, the retractor computes the desired force without regard to the pretensioner. If the resulting force is less than that specified by the pretensioner load curve, then the pretensioner value is used instead. As time goes on, the pretensioner load curve should drop below the forces generated by the retractor, and the pretensioner is then essentially inactive. This provides for good control of the actual forces, so no unrealistic values are generated. The actual direction and amount of belt movement is unspecified, and will depend on the other forces being exerted on the belt. This is suitable when the force the pretensioner exerts over time is known.

The type 5 pretensioner is essentially the same as the old type 1 pretensioner, but with the addition of a force limiting value. The pull-in is given as a function of time, and the belt is drawn into the retractor exactly as desired. However, if at any point the forces generated in the belt exceed the pretensioner force limit, then the pretensioner is deactivated and the retractor takes over. In order to prevent a large discontinuity in the force at this point, the loading curve for the retractor is shifted (in the abscissa) by the amount required to put the current (pull-out, force) on the load curve. For example, suppose the current force is 1000, and the current pull-out is -10 (10mm of belt has been pulled IN by the pretensioner). If the retractor would normally generate a force of 1000 after 25mm of belt had been pulled OUT, then the load curve is shifted to the left by 35,

and remains that way for the duration of the calculation. So that at the current pull-in of 10, it will generate the force normally associated with a pull out of 25. If the belt reaches a pull out of 5, the force will be as if it were pulled out 40 (5 + the shift of 35), and so on. This option is included for those who liked the general behavior of the old type 1 pretensioner, but has the added feature of the force limit to prevent unrealistic behavior.

The type 6 pretensioner is a variation of the type 4 pretensioner, with features of the type 5 pretensioner. A force vs. time curve is input and the pretensioner force is computed each cycle. The retractor linked to this pretensioner should specify a positive value for PULL, which is the distance the belt pulls out before it locks. As the pretensioner pulls the belt into the retractor, the amount of pull-in is tracked. As the pretensioner force decreases and drops below the belt tension, belt will begin to move back out of the retractor. Once PULL amount of belt has moved out of the retractor (relative to the maximum pull in encountered), the retractor will lock. At this time, the pretensioner is disabled, and the retractor force curve is shifted to match the current belt tension. This shifting is done just like the type 5 pretensioner. It is important that a positive value of PULL be specified to prevent premature retractor locking which could occur due to small outward belt movements generated by noise in the simulation.

The type 7 pretensioner is a simple combination of retractor and pretensioner. It is similar to the type 6 except for the following changes: when the retractor locks, the pretensioner is NOT disabled – it continues to exert force according to the force vs. time curve until the end of the simulation. (The force vs. time curve should probably drop to 0 at some time.) Furthermore, the retractor load curve is not shifted – the retractor begins to exert force according to the force vs. pull-out curve. These two forces are added together and applied to the belt. Thus, the pretensioner and retractor are essentially independent.

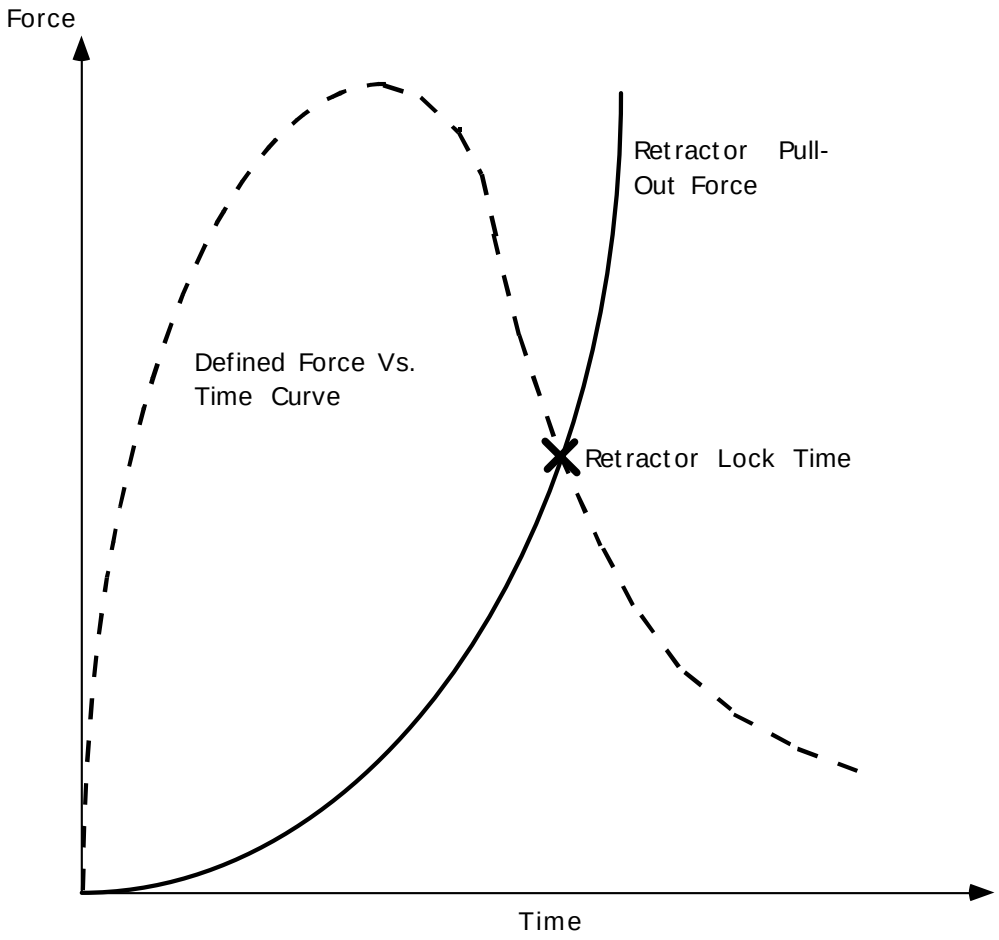


Figure 14.3. Force versus time pretensioner. At the intersection, the retractor locks.

*ELEMENT_SEATBELT_RETRACTOR

Purpose: Define seat belt retractor. See remarks below for seatbelt shell elements.

Card 1

Variable	SBRID	SBRNID	SBID	SID1	SID2	SID3	SID4	
Type	I	I	I	I	I	I	I	
Default	0	0	0	0	0	0	0	
Remarks		1,2	2	3				

Card 2

Variable	TDEL	PULL	LLCID	ULCID	LFED			
Type	F	F	I	I	F			
Default	0.0	0.0	0	0	0.0			
Remarks			4	5				

VARIABLE	DESCRIPTION
SBRID	Retractor ID. A unique number has to be used.
SBRNID	Retractor node ID
SBID	Seat belt element ID
SID1	Sensor ID 1
SID2	Sensor ID 2
SID3	Sensor ID 3
SID4	Sensor ID 4

VARIABLE	DESCRIPTION
TDEL	Time delay after sensor triggers.
PULL	Amount of pull-out between time delay ending and retractor locking, a length value.
LLCID	Load curve for loading (Pull-out, Force), see Figure 14.6.
ULCID	Load curve for unloading (Pull-out, Force), see Figure 14.6.
LFED	Fed length, see explanation below.

Remarks:

1. The retractor node should not be on any belt elements. The element defined should have one node coincident with the retractor node but should not be inside the retractor.

2. When SBRNID<0, this retractor is for shell-type seatbelt, -SBRNID is the *SET_NODE containing RN1, RN2, …RN5. SBID is then *SET_SHELL_LIST. Note that the numbering of –SBRNID, SBID has to be consistent in the direction of numbering. For example, if *SET_NODE for SBRNID has nodes of (RN1, RN2, RN3, RN4, RN5) then *SET_SHELL_LIST for SBID should have elem. of (RE1, RE2, RE3, RE4). See Figure 14.2.

3. At least one sensor should be defined.

4. The first point of the load curve should be (0, T_{min}). T_{min} is the minimum tension. All subsequent tension values should be greater than T_{min}.

5. The unloading curve should start at zero tension and increase monotonically (i.e., no segments of negative or zero slope).

   Retractors allow belt material to be paid out into a belt element. Retractors operate in one of two regimes: unlocked when the belt material is paid out, or reeled in under constant tension and locked when a user defined force-pullout relationship applies.

   The retractor is initially unlocked, and the following sequence of events must occur for it to become locked:

   1. Any one of up to four sensors must be triggered. (The sensors are described below.)

   2. Then a user-defined time delay occurs.

   3. Then a user-defined length of belt must be paid out (optional).

   4. Then the retractor locks and once locked, it remains locked.

In the unlocked regime, the retractor attempts to apply a constant tension to the belt. This feature allows an initial tightening of the belt and takes up any slack whenever it occurs. The tension value is taken from the first point on the force-pullout load curve. The maximum rate of pull out or pull in is given by 0.01 × fed length per time step. Because of this, the constant tension value is not always achieved.

In the locked regime, a user-defined curve describes the relationship between the force in the attached element and the amount of belt material paid out. If the tension in the belt subsequently relaxes, a different user-defined curve applies for unloading. The unloading curve is followed until the minimum tension is reached.

The curves are defined in terms of initial length of belt. For example, if a belt is marked at 10mm intervals and then wound onto a retractor, and the force required to make each mark emerge from the (locked) retractor is recorded, the curves used for input would be as follows:

0 Minimum tension (should be > zero)

10mm Force to emergence of first mark

20mm Force to emergence of second mark

. .

. .

. .

Pyrotechnic pretensions may be defined which cause the retractor to pull in the belt at a predetermined rate. This overrides the retractor force-pullout relationship from the moment when the pretensioner activates.

If desired, belt elements may be defined which are initially inside the retractor. These will emerge as belt material is paid out, and may return into the retractor if sufficient material is reeled in during unloading.

Elements e2, e3 and e4 are initially inside the retractor, which is paying out material into element e1. When the retractor has fed L_{crit} into e1, where

L_{crit} = fed length - 1.1 × minimum length

(minimum length defined on belt material input)

(fed length defined on retractor input)

Element e2 emerges with an unstretched length of 1.1 x minimum length; the unstretched length of element e1 is reduced by the same amount. The force and strain in e1 are unchanged; in e2, they are set equal to those in e1. The retractor now pays out material into e2.

If no elements are inside the retractor, e2 can continue to extend as more material is fed into it.

As the retractor pulls in the belt (for example, during initial tightening), if the unstretched length of the mouth element becomes less than the minimum length, the element is taken into the retractor.

To define a retractor, the user enters the retractor node, the 'mouth' element (into which belt material will be fed), e1 in Figure 12.3, up to 4 sensors which can trigger unlocking, a time delay, a payout delay (optional), load and unload curve numbers, and the fed length. The retractor node is typically part of the vehicle structure; belt elements should not be connected to this node directly, but any other feature can be attached including rigid bodies. The mouth element should have a node coincident with the retractor but should not be inside the retractor. The fed length would typically be set either to a typical element initial length, for the distance between painted marks on a real belt for comparisons with high speed film. The fed length should be at least three times the minimum length.

If there are elements initially inside the retractor (e2, e3 and e4 in the Figure) they should not be referred to on the retractor input, but the retractor should be identified on the element input for these elements. Their nodes should all be coincident with the retractor node and should not be restrained or constrained. Initial slack will automatically be set to $1.1 \times$ minimum length for these elements; this overrides any user-defined value.

Weblockers can be included within the retractor representation simply by entering a 'locking up' characteristic in the force pullout curve, see Figure 14.5. The final section can be very steep (but must have a finite slope).

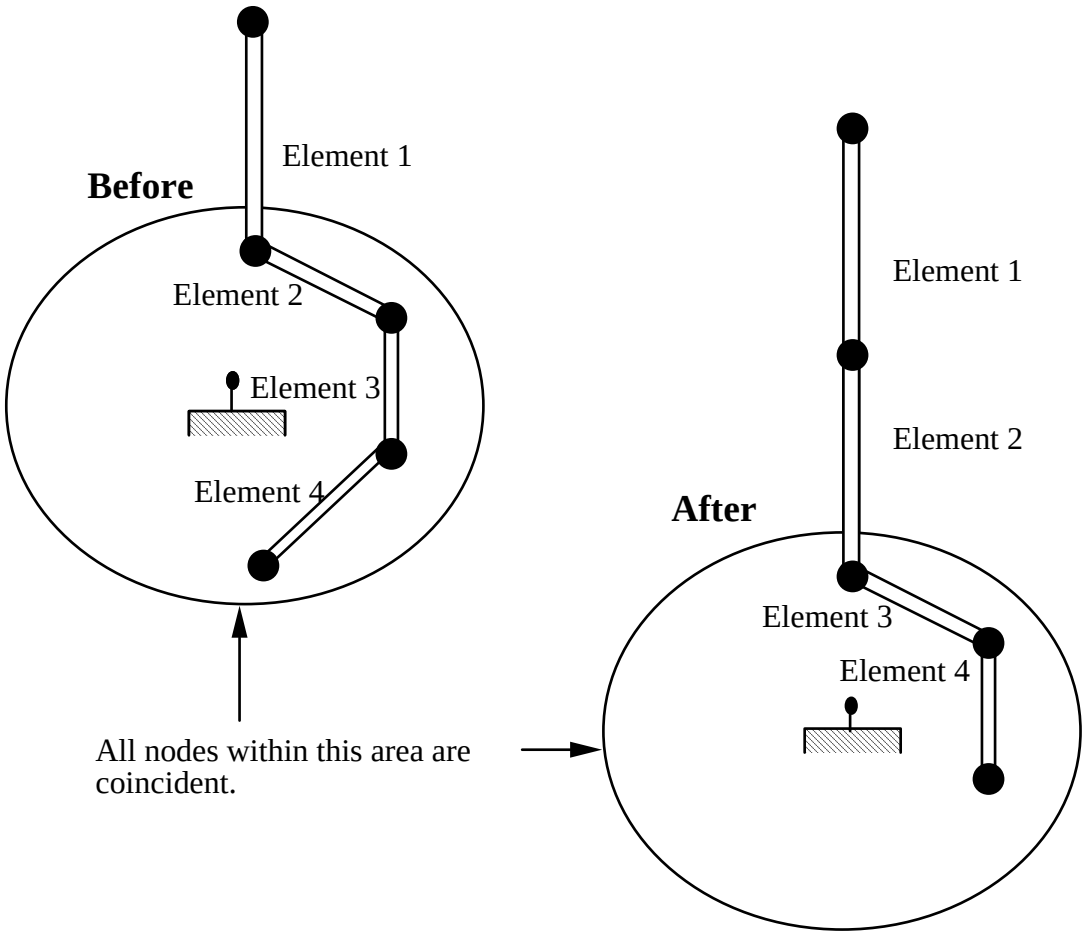


Figure 14.4. Elements in a retractor.

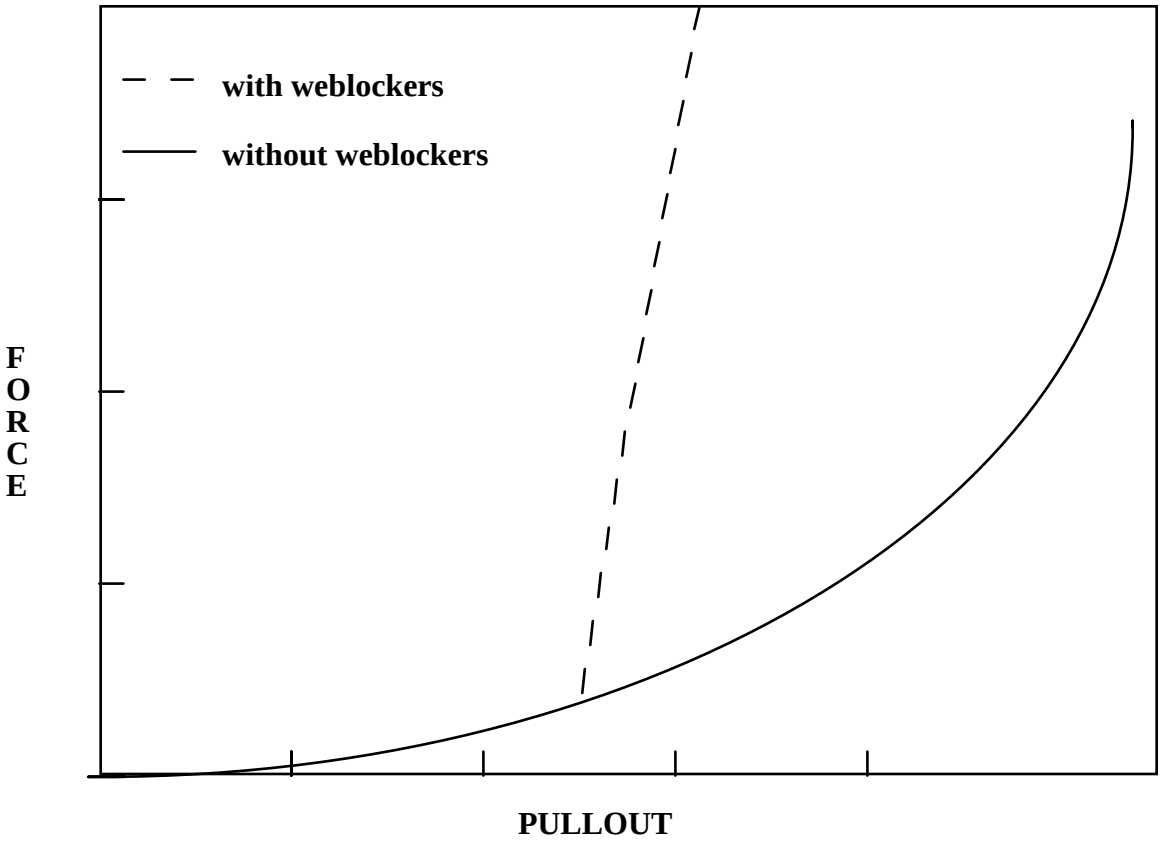


Figure 14.5. Retractor force pull characteristics.

*ELEMENT_SEATBELT_SENSOR

Purpose: Define seat belt sensor. Four types are possible, see explanation below.

Card 1	1	2	3	4	5	6	7	8
Variable	SBSID	SBSTYP	SBSFL					
Type	I	I	I					
Default	0	0	0					
Remarks								

Card 2 if SBSTYP=1

Card 2	1	2	3	4	5	6	7	8
Variable	NID	DOF	ACC	ATIME				
Type	I	I	F	F				
Default	0	0	0.0	0.0				
Remarks	1							

Card 2 if SBSTYP=2

Card 2	1	2	3	4	5	6	7	8
Variable	SBRID	PULRAT	PULTIM					
Type	I	F	F					
Default	0	0.0	0.0					
Remarks								

Card 2 if SBSTYP=3

Card 2	1	2	3	4	5	6	7	8
Variable	TIME							
Type	F							
Default	0.0							
Remarks								

Card 2 if SBSTYP=4

Card 2	1	2	3	4	5	6	7	8
Variable	NID1	NID2	DMX	DMN				
Type	I	I	F	F				
Default	0	0	0.0	0.0				
Remarks			2	2				

VARIABLE	DESCRIPTION
SBSID	Sensor ID. A unique number has to be used.
SBSTYP	Sensor type: EQ.1: acceleration of node, EQ.2: retractor pull-out rate, EQ.3: time, EQ.4: distance between nodes.
SBSFL	Sensor flag: EQ.0: sensor active during dynamic relaxation, EQ.1: sensor can be triggered during dynamic relaxation.
NID	Node ID of sensor
DOF	Degree of freedom: EQ.1: x, EQ.2: y, EQ.3: z.
ACC	Activating acceleration
ATIME	Time over which acceleration must be exceeded
SBRID	Retractor ID, see *ELEMENT_SEATBELT_RETRACTOR.
PULRAT	Rate of pull-out (length/time units)
PULTIM	Time over which rate of pull-out must be exceeded
TIME	Time at which sensor triggers
NID1	Node 1 ID
NID2	Node 2 ID
DMX	Maximum distance
DMN	Minimum distance

Remarks:

1. Node should not be on rigid body, velocity boundary condition, or other ‘imposed motion’ feature.

2. Sensor triggers when the distance between the two nodes is $d \geq d_{max}$ or $d \leq d_{min}$.

Sensors are used to trigger locking of retractors and activate pretensioners. Four types of sensors are available which trigger according to the following criteria:

Type 1 – When the magnitude of x-, y-, or z- acceleration of a given node has remained above a given level continuously for a given time, the sensor triggers. This does not work with nodes on rigid bodies.

Type 2 – When the rate of belt payout from a given retractor has remained above a given level continuously for a given time, the sensor triggers.

Type 3 – The sensor triggers at a given time.

Type 4 – The sensor triggers when the distance between two nodes exceeds a given maximum or becomes less than a given minimum. This type of sensor is intended for use with an explicit mass/spring representation of the sensor mechanism.

By default, the sensors are inactive during dynamic relaxation. This allows initial tightening of the belt and positioning of the occupant on the seat without locking the retractor or firing any pretensioners. However, a flag can be set in the sensor input to make the sensors active during the dynamic relaxation phase.

*ELEMENT_SEATBELT_SLIPRING

Purpose: Define seat belt slip ring.

Card	1	2	3	4	5	6	7	8
Variable	SBSRID	SBID1	SBID2	FC	SBRNID	LTIME	FCS	ONID
Type	I	I	I	F	I	F	F	I
Default	0	0	0	0.0	0	1.0E20	0.0	0

Define the following card if and only if an orientation node is specified.

Card 2	1	2	3	4	5	6	7	8
Variable	K	FUNCID	DIRECT	DC				
Type	F	I	I	F				
Default	0.	0	0	0				

VARIABLE	DESCRIPTION
SBSRID	Slipring ID. A unique number has to be used. See remarks below for the treatment of sliprings for shell belt elements.
SBID1	Seat belt element 1 ID
SBID2	Seat belt element 2 ID
FC	Coulomb dynamic friction coefficient. If less than zero, |FC| refers to a curve which defines the dynamic friction coefficient as a function of time.
SBRNID	Slip ring node, NID
LTIME	Slip ring lockup time. After this time no material is moved from one side of the slip ring to the other. This option is not active during dynamic relaxation.
FCS	Optional Coulomb static friction coefficient. . If less than zero, |FCS| refers to a curve which defines the static friction coefficient as a function of time.

VARIABLE	DESCRIPTION
ONID	Optional orientation node ID.
K	Optional coefficient for determining the Coulomb friction coefficient related to angle alpha
FUNCID	Function ID to determine friction coefficient.
DIRECT	Direction of belt movement: EQ. 0: if the belt can move along both directions. EQ.12: if the belt is only allowed to slip along the direction from SBID1 to SBID2 EQ.21: if the belt is only allowed to slip along the direction from SBID2 to SBID
DC	Optional decay constant to allow a smooth transition between the static and dynamic friction coefficients, i.e., $\mu_c = FC + (FCS - FC)e^{-DC\cdot\lvert v_{rel}\rvert}$

Remarks:

When SBRNID<0, this slipring is for shell-type seatbelt, -SBRNID is the *SET_NODE containing SN1, SN2, …SN5. SBID1 and SBID2 are then *SET_SHELL_LIST. Note that the numbering of -SBRNID, SBID1 and SBID2 has to be consistent in the direction of numbering. For example if, *SET_NODE for SBRNID has nodes of (SN1, SN2, SN3, SN4, SN5) then *SET_ SHELL_LIST for SBID1 should have elem. of (SRE11, SRE12, SRE13, SRE14) and *SET_SHELL_LIST for SBID2 should have elem. of (SRE21, SRE22, SRE23, SRE24). See Figure 14.2.

Elements 1 and 2 should share a node which is coincident with the slip ring node. The slip ring node should not be on any belt elements.

Sliprings allow continuous sliding of a belt through a sharp change of angle. Two elements (1 & 2 in Figure 14.6) meet at the slipring. Node B in the belt material remains attached to the slipring node, but belt material (in the form of unstretched length) is passed from element 1 to element 2 to achieve slip. The amount of slip at each time step is calculated from the ratio of forces in elements 1 and 2. The ratio of forces is determined by the relative angle between elements 1 and 2 and the coefficient of friction, FC. The tension in the belts are taken as T_1 and T_2, where T_2 is on the high tension side and T_1 is the force on the low tension side. Thus, if T_2 is sufficiently close to T_1, no slip occurs; otherwise, slip is just sufficient to reduce the ratio T_2/T_1 to $e^{FC\cdot\theta}$, where θ is the wrap angle, see Figures 14.7 and 14.8. No slip occurs if both elements are slack. The out-of-balance force at node B is reacted on the slipring node; the motion of node B follows that of slipring node.

If, due to slip through the slipring, the unstretched length of an element becomes less than the minimum length (as entered on the belt material card), the belt is remeshed locally: the short element passes through the slipring and reappears on the other side (see Figure 14.6). The new

unstretched length of e1 is 1.1 × minimum length. Force and strain in e2 and e3 are unchanged; force and strain in e1 are now equal to those in e2. Subsequent slip will pass material from e3 to e1. This process can continue with several elements passing in turn through the slipring.

To define a slipring, the user identifies the two belt elements which meet at the slipring, the friction coefficient, and the slipring node. The two elements must have a common node coincident with the slipring node. No attempt should be made to restrain or constrain the common node for its motion will automatically be constrained to follow the slipring node. Typically, the slipring node is part of the vehicle body structure and, therefore, belt elements should not be connected to this node directly, but any other feature can be attached, including rigid bodies.

If K is undefined, the limiting force ratio is taken as $e^{FC\cdot\theta}$. If K is defined, the maximum force ratio is computed as

$$e^{FC\cdot\theta\left(1+K\cdot\alpha^2\right)}$$

where alpha is the angle shown in Figure 14.9. If FUNCID is specified, the coefficients FC, FCS, and K are not used. The function is defined using the *DEFINE_FUNCTION keyword input. This function is a function of two variables, and the ratio is given by evaluating

$$\frac{T_2}{T_1} = FUNC(\theta,\alpha)$$

For example, the default behavior can be obtained using the function definition (assuming FC has a value of 0.025 and the function ID is unity):

```
*DEFINE_FUNCTION
   1,
f(theta,alpha) = exp(0.025*theta)
```

Behavior like default option can be obtained with (K=0.1):

```
*DEFINE_FUNCTION
  1,
f(theta,alpha) = exp(0.025*theta*(1.+0.1*alpha*alpha))
```

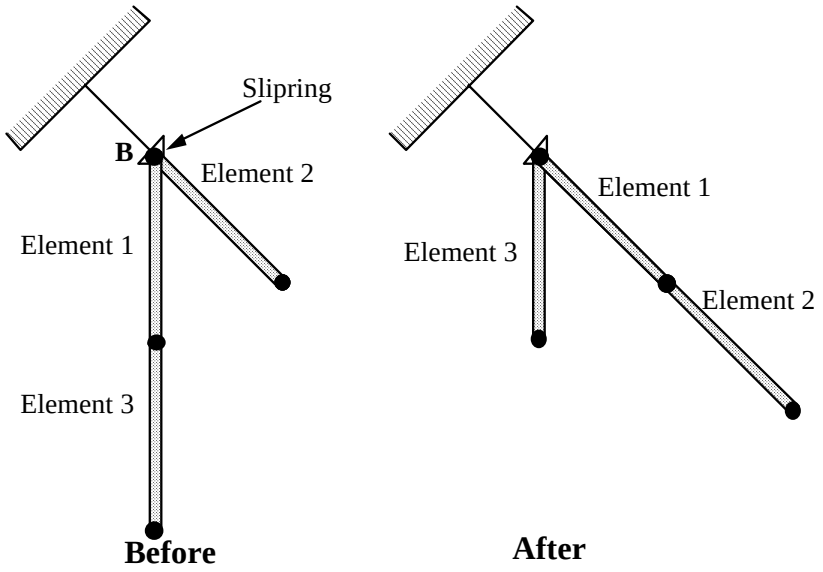


Figure 14.6. Elements passing through slipring.

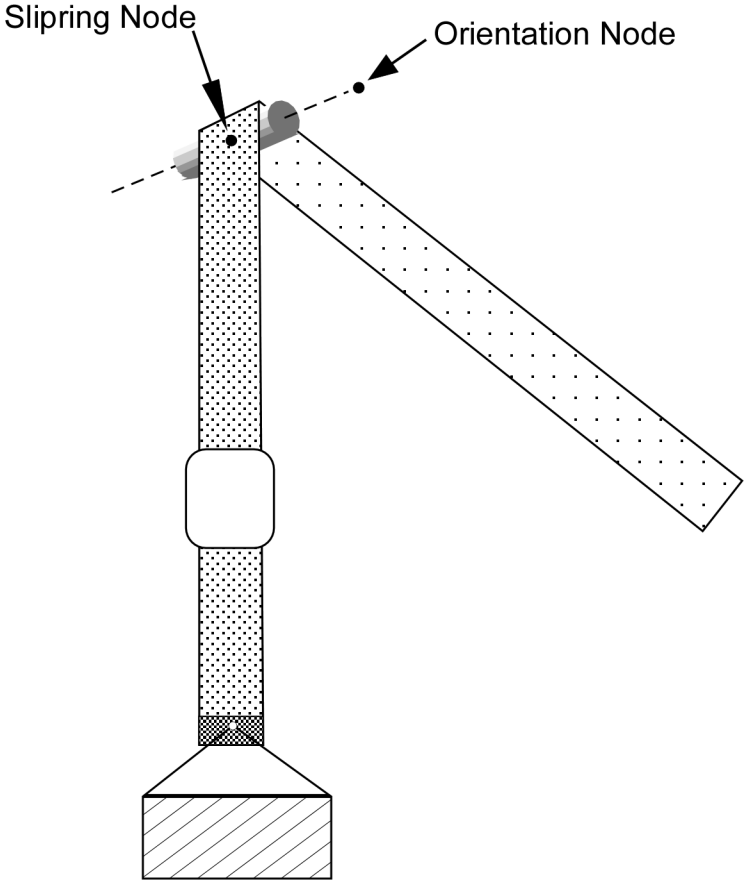


Figure 14.7. Orientation node.

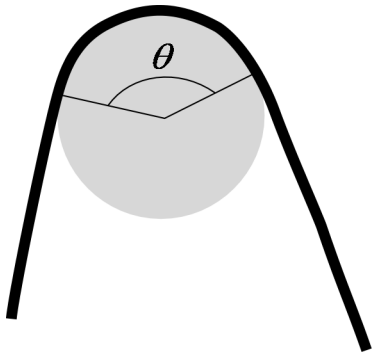


Figure 14.8. Front view showing wrap angle.

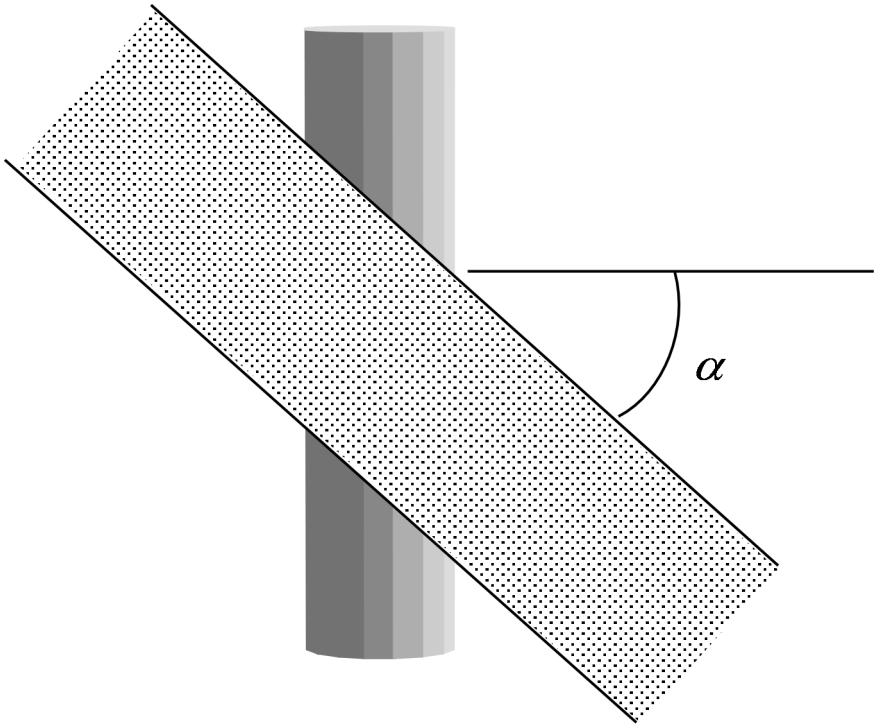


Figure 14.9. Top view shows orientation of belt relative to axis.

***ELEMENT_SHELL_**{*OPTION*}

Available options include:

<BLANK>

THICKNESS

BETA or MCID

OFFSET

DOF

Purpose: Define three, four, six, and eight node elements including 3D shells, membranes, 2D plane stress, plane strain, and axisymmetric solids. The type of the element and its formulation is specified through the part ID (see *PART) and the section ID (see *SECTION_SHELL). Also, the thickness of each element can be specified when applicable on the element cards or else a default thickness value is used from the section definition. For orthotropic and anisotropic materials a local material angle (variable PSI) can be defined which is cumulative with the integration point angles specified in *SECTION_SHELL. Alternatively, the angle PSI can be determined by defining a local coordinate system, MCID. An offset option is available for moving the shell reference surface from the nodal points that define the shell.

For the shell formulation that uses additional nodal degrees-of-freedom, the option DOF is available to connect the nodes of the shell to corresponding scalar nodes. Four scalar nodes are required for element type 25 to model the thickness changes that require 2 additional degrees-of-freedom per shell node. Defining these nodes is optional, if left undefined, they will be automatically created.

Card Format (10I8)

Card 1	1	2	3	4	5	6	7	8	9	10
Variable	EID	PID	N1	N2	N3	N4	N5	N6	N7	N8
Type	I	I	I	I	I	I	I	I	I	I
Default	none	none	none	none	none	none	0	0	0	0
Remarks			3	3	3	3				

Optional Card (Required if THICKNESS or BETA is specified after the keyword) (5E16.0)

Card	1	2	3	4	5	6	7	8	9	10
Variable	THIC1		THIC2		THIC3		THIC4		BETA or MCID	
Type	F		F		F		F		F	
Default	0.		0.		0.		0.		0.	
Remarks	1								2	

Optional Card (Required if OFFSET is specified after the keyword) (E16.0)

Card	1	2	3	4	5	6	7	8	9	10
Variable	OFFSET									
Type	F									
Default	0.									
Remarks	7									

Optional Card for scalar nodes (Required if DOF is specified after the keyword) (10I8)

Card	1	2	3	4	5	6	7	8	9	10
Variable			NS1	NS2	NS3	NS4				
Type			I	I	I	I				
Default										
Remarks			8	8	8	8				

VARIABLE	DESCRIPTION
EID	Element ID. Chose a unique number with respect to other elements.
PID	Part ID, see *PART.
N1	Nodal point 1
N2	Nodal point 2
N3	Nodal point 3
N4	Nodal point 4
N5-N8	Mid-side nodes for eight node shell
THIC1	Shell thickness at node 1
THIC2	Shell thickness at node 2
THIC3	Shell thickness at node 3
THIC4	Shell thickness at node 4
BETA	Orthotropic material base offset angle (see remarks 2 and 6 below). The angle is given in degrees. If blank the default is set to zero.
MCID	Material coordinate system ID. The angle BETA is taken as the angle from the element local x-axis (N1-to-N2) to the projection of the x-axis of the local coordinate system, MCID, onto the surface of the shell element.
OFFSET	The offset distance from the plane of the nodal points to the reference surface of the shell in the direction of the normal vector to the shell.
NS1	Scalar node 1, parameter NDOF on the *NODE_SCALAR is normally set to 2. If the thickness is constrained, set NDOF=0.
NS2	Scalar node 2
NS3	Scalar node 3
NS4	Scalar node 4

Remarks:

1. Default values in place of zero shell thicknesses are taken from the cross-section property definition of the PID, see *SECTION_SHELL.

2. BETA is defined only for orthotropic and anisotropic materials.

3. Counterclockwise node numbering determines the top surface, see Figure 14.7.

4. Stresses and strain output in the binary databases are by default given in the global coordinate system. Stress resultants are output in the local coordinate system for the shell element.

5. Interior angles must be less than 180 degrees.

6. To allow for an arbitrary orientation of the shell elements within the finite element mesh, each ply in the composite can have a unique material orientation angle which measures the offset from a reference system in the element. The reference system is determined by the AOPT and associated parameters in the *MAT input. The total offset for the i'th integration point through the element thickness consists of two parts, the base offset and the layer offset. We write this as

$$\theta_i = \beta + \beta_i$$

where β is the base offset and β_i is the layer offset. The element BETA input here overrides the BETA on *MAT input. The β_i angles are input either by *PART_COMPOSITE, or by using the ICOMP option on *SECTION_SHELL. Figures 14.8 and 14.9 depict these angles.

7. The parameter OFFSET gives the offset from the nodal points of the shell to the reference surface. This option applies to most shell formulations excluding two-dimensional elements, membrane elements, and quadratic shell elements. The reference surface offset given by OFFSET is not taken into account in the contact subroutines unless CNTCO is set to 1 in *CONTROL_SHELL.

8. The scalar nodes specified on the optional card refer to the scalar nodes defined by the user to hold additional degrees of freedom for shells with this capability. Scalar nodes are used with shell element type 25 and 26.

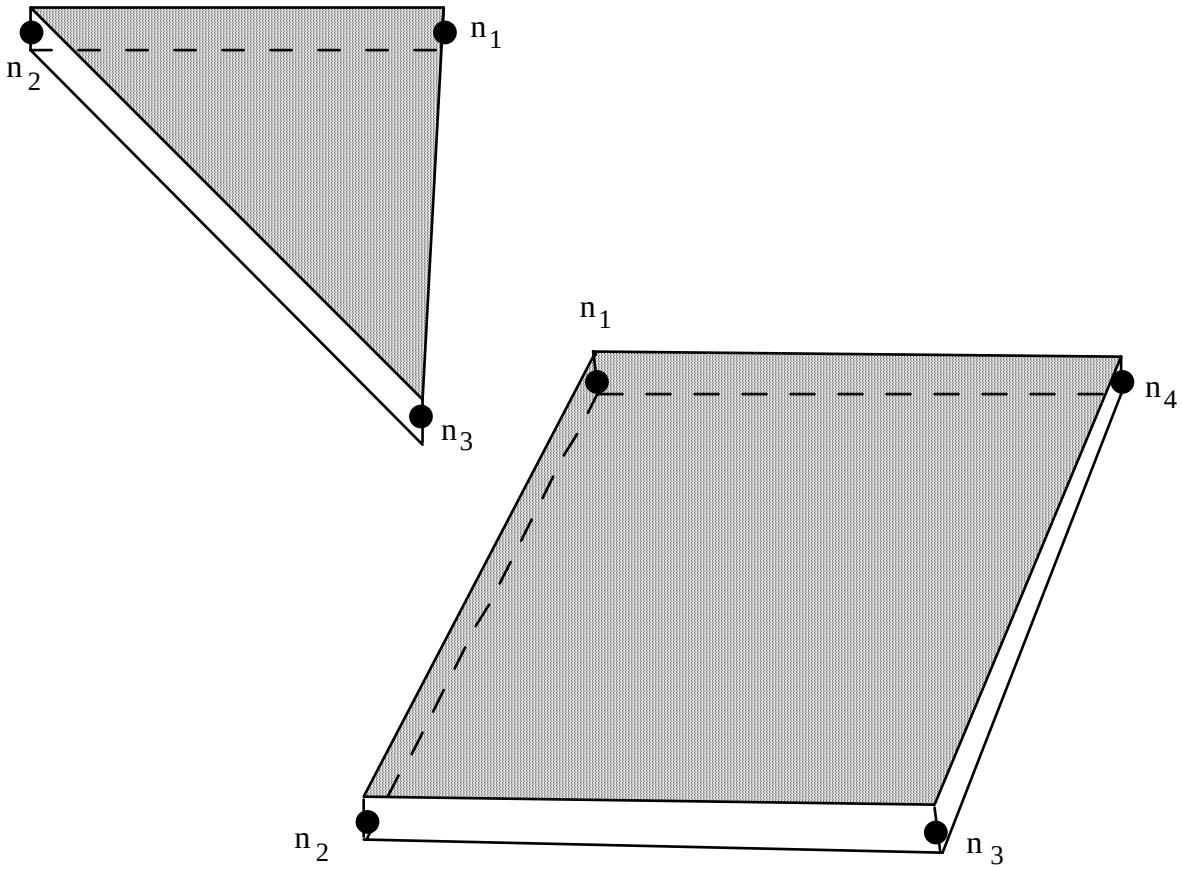


Figure 14.7. LS-DYNA shell elements. Counterclockwise node numbering determines the top surface.

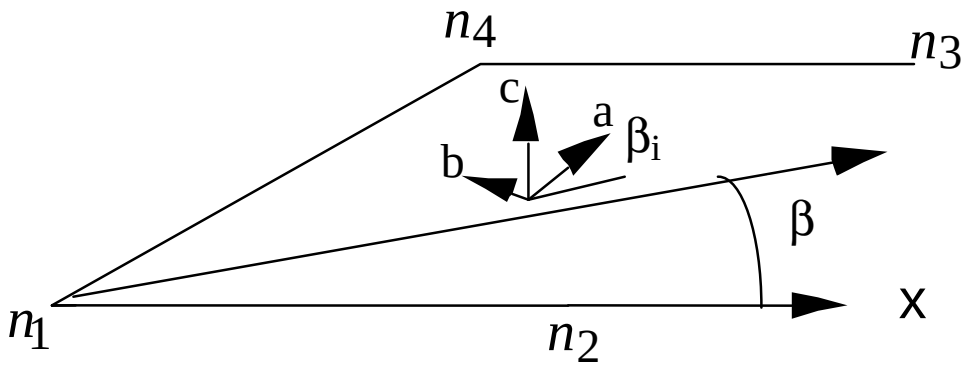


Figure 14.8. Orientation of material directions (shown relative to the 1-2 side as when AOPT=0 in *MAT).

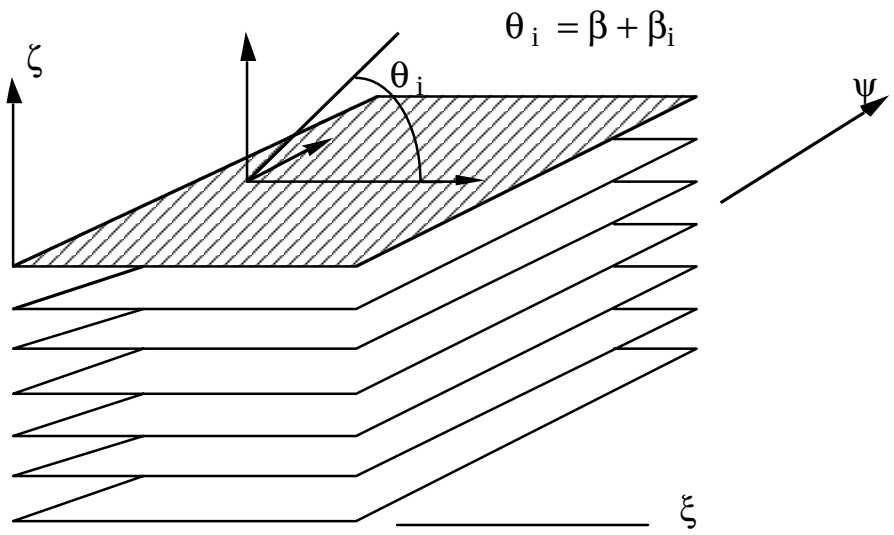


Figure 14.9. A multi-layer laminate can be defined. The angle β_i is defined for the i'th lamina (integration point), see *SECTION_SHELL.

*ELEMENT_SHELL_SOURCE_SINK

Purpose: Define a strip of shell elements of a single part ID to simulate a continuous forming operation. This option requires logical regular meshing of rectangular elements, which implies that the number of nodal points across the strip is constant along the length. Elements are created at the source and disappear at the sink. The advantage of this approach is that it is not necessary to define an enormous number of elements to simulate a continuous forming operation. Currently, only one source-sink definition is allowed. The boundary conditions at the source are discrete nodal point forces to keep the work piece in tension. At the sink, displacement boundary conditions are applied.

Card	1	2	3	4	5	6	7	8
Variable	NSSR	NSSK	PID					
Type	I	I	I					
Default	none	none	none					

VARIABLE	DESCRIPTION
NSSR	Node set at source. Provide an ordered set of nodes between corner nodes, which include the corner nodes.
NSSK	Node set at sink. Provide an ordered set of nodes between corner nodes, which include the corner nodes.
PID	Part ID of work piece.

***ELEMENT_SOLID_**{*OPTION*}

Available options include:

<BLANK>

ORTHO

DOF

TET4TOTET10

Purpose: Define three-dimensional solid elements including 4 noded tetrahedrons and 8-noded hexahedrons. The type of solid element and its formulation is specified through the part ID (see *PART) and the section ID (see *SECTION_SOLID_*OPTION*). Also, a local coordinate system for orthotropic and anisotropic materials can be defined by using the ORTHO option. If extra degrees of freedom are needed, the DOF option should be used. The option TET4TOTET10 converts 4 node tetrahedrons to 10 node tetrahedrons. See remarks below.

Card Format (2I8)

Card 1	1	2	3	4	5	6	7	8	9	10
Variable	EID	PID								
Type	I	I								
Default	None	none								
Remarks	2									

Card 2

Variable	N1	N2	N3	N4	N5	N6	N7	N8	N9	N10
Type	I	I	I	I	I	I	I	I	I	I
Default	None	none	none	none	none	None	none	none	none	none

Optional Cards (Required if ORTHO is specified after the keyword)

Optional card 1	1	2	3	4	5	6	7	8	9	10
Variable	A1		A2		A3					
Type	F		F		F					
Default	0.		0.		0.					
Remarks	3									

Optional card 2

Variable	D1		D2		D3					
Type	F		F		F					
Default	0.		0.		0.					
Remarks	3									

Optional Card (Required if DOF is specified after the keyword)

Card 2	1	2	3	4	5	6	7	8	9	10
Variable			NS1	NS2	NS3	NS4	NS5	NS6	NS7	NS8
Type			I	I	I	I	I	I	I	I
Default			none	none	none	None	none	none	none	none
Remarks										

VARIABLE	DESCRIPTION
EID	Element ID. A unique number has to be chosen.
PID	Part ID, see *PART.
N1	Nodal point 1
N2	Nodal point 2
N3	Nodal point 3
.	.
.	.
N10	Nodal point 10
A1	x-component of local material direction a, or else rotation angle in degrees (see remark 4).
A2	y-component of local material direction a.
A3	z-component of local material direction a.
D1	x-component of vector in the plane of the material vectors a and b.
D2	y-component of vector in the plane of the material vectors a and b.
D3	z-component of vector in the plane of the material vectors a and b.
NS1	Scalar node 1
NS2	Scalar node 2
NS3	Scalar node 3
NS4	Scalar node 4
NS5	Scalar node 5
NS6	Scalar node 6
NS7	Scalar node 7
NS8	Scalar node 8

Remarks:

1. The option TET4TOTET10 automatically converts 4 node tetrahedron solids to 10 node quadratic tetrahedron solids. Additional mid-side nodes are created which are shared by all tetrahedron elements that contain the edge. The user node ID's for these generated nodes are offset after the largest user node ID defined in the input file. When defining the *SECTION_SOLID keyword, the element type must be specified as either 16 or 17 which are the 10-noded tetrahedrons in LS-DYNA.

2. Four, six, and eight node elements are depicted in Figure 14.10 where the ordering of the nodal points is shown. This ordering must be followed or code termination with occur during the initialization phase with a negative volume message. The input of nodes on the element cards for the tetrahedron and pentahedron elements is given by:

   4-noded tetrahedron N1, N2, N3, N4, N4, N4, N4, N4, 0, 0

   6-noded pentahedron N1, N2, N3, N4, N5, N5, N6, N6, 0, 0

   If hexahedrons are mixed with tetrahedrons and pentahedrons in the input under the same part ID, degenerate tetrahedrons and pentahedrons are used. One problem with degenerate elements is related to an uneven mass distribution (node 4 of the tetrahedron has five times the mass of nodes 1-3) which can make these elements somewhat unstable with the default time step size. By using the control flag under the keyword, *CONTROL_SOLID, automatic sorting can be invoked to treat the degenerate elements as type 10 and type 15 tetrahedron and pentahedron elements, respectively.

   For elements with 4-8 nodes the card formats of LS-DYNA versions 940-970 are still valid. Card 2 is not defined in the older format.

Card Format (10I8)

Card 1	1	2	3	4	5	6	7	8	9	10
Variable	EID	PID	N1	N2	N3	N4	N5	N6	N7	N8
Type	I	I	I	I	I	I	I	I	I	I

3. For the orthotropic and anisotropic material models the local directions may be defined on the second card following the element connectivity definition. The local directions are then computed from the two vectors such that (see Figure 14.11):

$$\underset{\sim}{c} = \underset{\sim}{a} \times \underset{\sim}{d} \text{ and } \underset{\sim}{b} = \underset{\sim}{c} \times \underset{\sim}{a} .$$

   These vectors are internally normalized within LS-DYNA. If the material model uses AOPT=3, the *a* and *b* axes will be rotated about the *c* axis by the BETA angle on the material card.

4. Stress output for solid elements is in the global coordinate system by default.

5. If vector **d** is input as a zero length vector, then A1 is interpreted as an offset rotation angle in degrees which describes a rotation about the **c**-axis of the **a**-**b**-**c** coordinate system that is defined by AOPT and associated parameters on the *MAT input. This angle overrides the offset angle defined by BETA on the *MAT input.

6. The scalar nodes specified on the optional card refer to the scalar nodes defined by the user to hold additional degrees of freedom for solids with this capability. This option is primarily to be used with user defined solids.

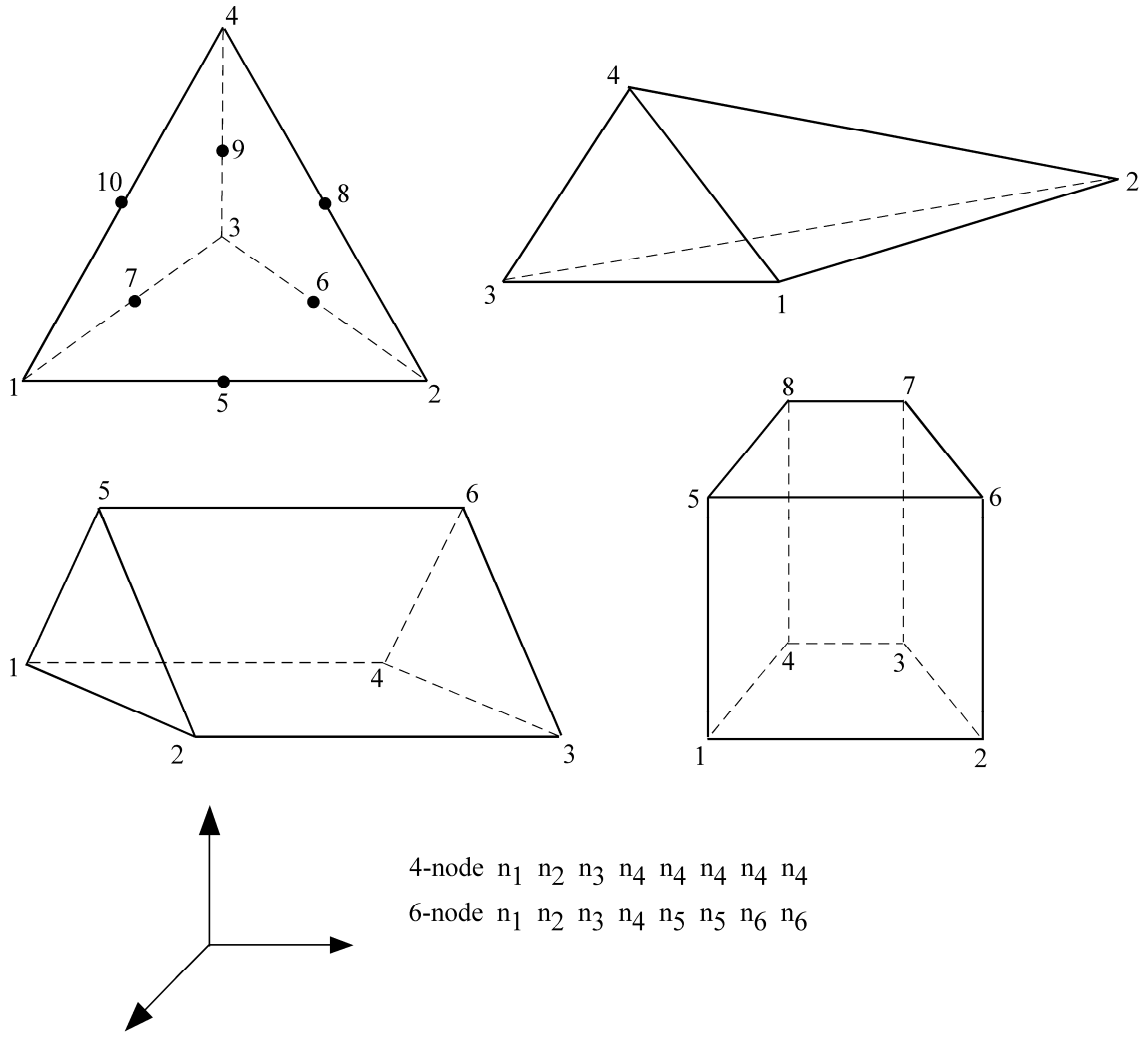


Figure 14.10. Four, six, and eight node solid elements. Nodes 1-4 are on the bottom surface.

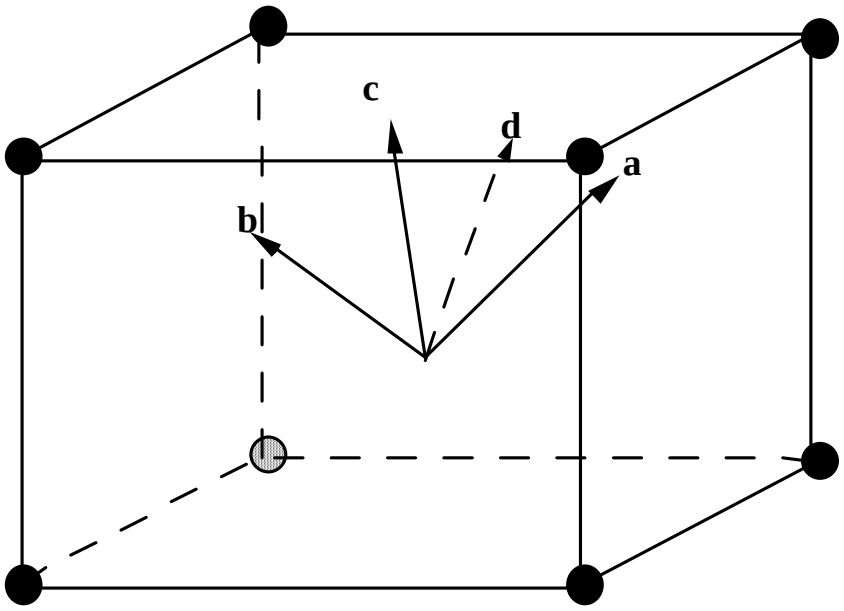


Figure 14.11. Two vectors **a** and **d** are defined and the triad is computed and stored.

*ELEMENT_SPH

Purpose: Define a lumped mass element assigned to a nodal point.

Card Format (2I8,E16.0)

Card	1	2	3	4	5	6	7	8	9	10
Variable	NID	PID	MASS							
Type	I	I	F							
Default	none	none	0.							
Remarks										

VARIABLE	DESCRIPTION
NID	Node ID and Element ID are the same for the SPH option.
PID	Part ID to which this node (element) belongs.
MASS	Mass value

***ELEMENT_TRIM**

Purpose: Define a part subset to be trimmed by *DEFINE_CURVE_TRIM.

Card Format (8I10)

Card 1	1	2	3	4	5	6	7	8
Variable	PSID							
Type	I							
Default	none							

VARIABLE	DESCRIPTION
PSID	Part set ID for trimming, see *SET_PART.

Remarks:

1. This keyword is used in combination with *DEFINE_CURVE_TRIM to trim the parts in PSID at time=0, i.e. before the simulation begins.

*ELEMENT_TSHELL

Purpose: Define an eight node thick shell element which is available with either fully reduced or selectively reduced integration rules. This plane stress element can be used as an alternative to the 4 node shell elements in cases where an 8-noded element is desired. Care must be taken in defining the element connectivity as N1 to N4 define the lower surface of the thick shell. The number of through-thickness integration points is defined by the user. The definition is completed by the *PART and *SECTION_TSHELL cards.

Card Format (10I8)

Card	1	2	3	4	5	6	7	8	9	10
Variable	EID	PID	N1	N2	N3	N4	N5	N6	N7	N8
Type	I	I	I	I	I	I	I	I	I	I
Default	none	none	none	none	none	none	none	none	none	none
Remarks			1							

VARIABLE	DESCRIPTION
EID	Element ID. Unique numbers have to be used.
PID	Part ID, see *PART.
N1	Nodal point 1
N2	Nodal point 2
N3	Nodal point 3
.	.
.	.
N8	Nodal point 8

Remarks:

1. The correct numbering of the nodes is essential for correct use. Nodes n_1 to n_4 define the lower surface, and nodes n_5 to n_8 define the upper surface. If one point integration is used (see *SECTION_TSHELL), the integration points then lie along the t-axis as depicted in Figure 14.12. Two by two selective reduced integration is also available. Extreme care must be used in defining the connectivity to insure proper orientation.

2. The stresses for this shell element are output in the global coordinate system.

3. To define a thick shell wedge element nodal pairs n3 & n4 and n7 & n8 are repeated. The ordering is then n1, n2, n3, n3, n4, n5, n6, n6, where nodes n1, n2, n3 form the lower triangular face and nodes n4, n5, n6 for the upper triangular face of the wedge.

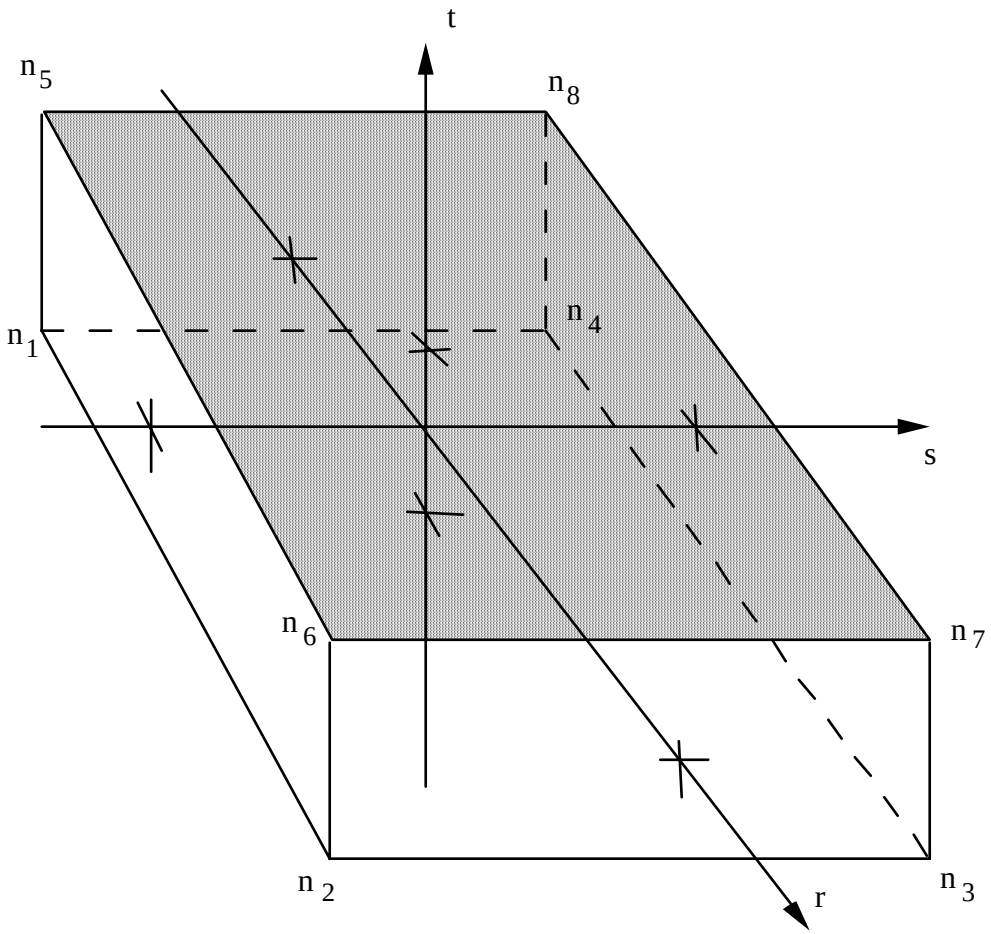


Figure 14.12. Solid 8-node Shell Element.

*EOS

LS-DYNA has historically referenced equations of state by type identifiers. Below these identifiers are given with the corresponding keyword name in the order that they appear in the manual. The equations of state can be used with a subset of the materials that are available for solid elements. Type 15 is linked to the type 2 thick shell element and can be used to model engine gaskets.

TYPE 1: ***EOS_LINEAR_POLYNOMIAL**

TYPE 2: ***EOS_JWL**

TYPE 3: ***EOS_SACK_TUESDAY**

TYPE 4: ***EOS_GRUNEISEN**

TYPE 5: ***EOS_RATIO_OF_POLYNOMIALS**

TYPE 6: ***EOS_LINEAR_POLYNOMIAL_WITH_ENERGY_LEAK**

TYPE 7: ***EOS_IGNITION_AND_GROWTH_OF_REACTION_IN_HE**

TYPE 8: ***EOS_TABULATED_COMPACTION**

TYPE 9: ***EOS_TABULATED**

TYPE 10: ***EOS_PROPELLANT_DEFLAGRATION**

TYPE 11: ***EOS_TENSOR_PORE_COLLAPSE**

TYPE 12: ***EOS_IDEAL_GAS**

TYPE 14: ***EOS_JWLB**

TYPE 15: ***EOS_GASKET**

TYPE 21-30: ***EOS_USER_DEFINED**

An additional option **_TITLE** may be appended to all the ***EOS** keywords. If this option is used then an additional line is read for each section in 80a format which can be used to describe the equation of state. At present LS-DYNA does not make use of the title. Inclusion of title simply gives greater clarity to input decks.

DEFINITIONS & NOTES ON SOME COMMONLY USED PARAMETERS:

In order to prescribe the boundary and/or initial thermodynamic condition, manual computations are often necessary. Conventions or definitions must be established to simplify this process. Some basic variables are defined in the following. Since many of these variables have already been denoted by different symbols, the notations used here are unique in this section only! They are presented to only clarify their usage. A corresponding SI unit set is also presented as an example.

First consider a few volumetric parameters since they are a measure of compression (or expansion).

Volume = $V \approx (m^3)$

Mass = $M \approx (Kg)$

Current specific volume (per mass) = $\upsilon = \frac{V}{M} = \frac{1}{\rho} \approx \left(\frac{m^3}{Kg} \right)$

Reference specific volume = $\upsilon_0 = \frac{V_0}{M} = \frac{1}{\rho_0} \approx \left(\frac{m^3}{Kg} \right)$

Relative volume = $\upsilon_r = \frac{V}{V_0} = \frac{(V/M)}{(V_0/M)} = \frac{\upsilon}{\upsilon_0} = \frac{\rho_0}{\rho}$

Current normalized volume increment = $\frac{d\upsilon}{\upsilon} = \frac{\upsilon - \upsilon_0}{\upsilon} = 1 - \frac{1}{\upsilon_r} = 1 - \frac{\rho}{\rho_0}$

A frequently used volumetric parameter is $\mu = \frac{1}{\upsilon_r} - 1 = \frac{\upsilon_0 - \upsilon}{\upsilon} = -\frac{d\upsilon}{\upsilon} = \frac{\rho}{\rho_0} - 1$

Sometimes another volumetric parameter is used: $\eta = \frac{\upsilon_0}{\upsilon} = \frac{\rho}{\rho_0}$

Thus the relation between μ and η is $\mu = \frac{\upsilon_0 - \upsilon}{\upsilon} = \eta - 1$

The following table summarizes these volumetric parameters.

VARIABLES	COMPRESSION	NO LOAD	EXPANSION
$\upsilon_r = \frac{\upsilon}{\upsilon_0} = \frac{\rho_0}{\rho}$	< 1	1	> 1
$\eta = \frac{1}{\upsilon_r} = \frac{\upsilon_0}{\upsilon} = \frac{\rho}{\rho_0}$	> 1	1	< 1
$\mu = \frac{1}{\upsilon_r} - 1 = \eta - 1$	> 0	0	< 0

V0 – INITIAL REALTIVE VOLUME

There are 3 definitions of density that must be distinguished from each other:

$\rho_0 = \rho_{ref}$ = Density at nominal/reference state, usually non-stress or non-deformed state.
$\rho|_{t=0}$ = Density at t = 0
ρ = Current density

Recalling the current relative volume, $\upsilon_r = \frac{\rho_0}{\rho} = \frac{\upsilon}{\upsilon_0}$, and the relative volume at time=0 is then $\upsilon_{r0} = \upsilon_r|_{t=0} = \frac{\rho_0}{\rho|_{t=0}} = \frac{\upsilon|_{t=0}}{\upsilon_0}$. Generally, the V0 input parameter in an *EOS_ card, refers to this υ_{r0}. ρ_0 is generally the density defined in the *MAT_ card. Hence, if a material is mechanically compressed at t=0, V0, or υ_{r0}, the initial relative volume, may be computed and input accordingly ($\upsilon_0 \neq V0$).

The "reference" state is a unique state with respect to which the material stress tensor is computed. Therefore υ_0 is very critical in computing the pressure level in a material. Incorrect choice of υ_0 would lead to incorrect pressure computed. In general, υ_0 is chosen such that at zero compression or expansion, the material should be in equilibrium with its ambient surrounding. In many of the equations shown in the EOS section, μ is frequently used as a measure of compression (or expansion). However, the users must clearly distinguish between μ and υ_{r0}.

E0 - INTERNAL ENERGY

Internal energy represents the thermal energy state (temperature dependent component) of a system. One definition for internal energy is

$$E = MC_v T \approx (Joule)$$

Note that the capital "E" here is the absolute internal energy. It is not the same as that used in the subsequent *EOS keyword input, or some equations shown for each *EOS_ card. This internal energy is often defined with respect to a mass or volume unit.

Internal energy per unit mass (also called specific internal energy):

$$e = \frac{E}{M} = C_V T \approx \left(\frac{Joule}{Kg} \right)$$

Internal energy per unit current volume:

$$e_V = \frac{M}{V} C_V T = \rho C_V T = \frac{C_V T}{\upsilon} \approx \left(\frac{Joule}{m^3} = \frac{N}{m^2} \right)$$

Internal energy per unit reference volume:

$$e_{V0} = \frac{M}{V_0} C_v T = \rho_0 C_v T = \frac{C_v T}{\upsilon_0} \approx \left(\frac{Joule}{m^3} = \frac{N}{m^2} \right).$$

e_{V0} typically refers to the capital "E" shown in some equations under this "EOS" section. Hence the initial "*internal energy per unit reference volume*", E0, a keyword input parameter in the *EOS section can be computed from

$$e_{V0}\big|_{t=0} = \rho_0 C_V \, T\big|_{t=0}$$

To convert from e_{V0} to e_V, simply divide e_{V0} by υ_r

$$e_V = \rho C_V T = [\rho_0 C_V T] \frac{\rho}{\rho_0} = \frac{e_{V0}}{\upsilon_r}$$

EQUATION OF STATE (EOS)

A thermodynamic state of a homogeneous material, not undergoing any chemical reactions or phase changes, may be defined by two state variables. This relation is generally called an equation of state. For example, a few possible forms relating pressure to two other state variables are

$$P = P(\rho, T) = P(\upsilon, e) = P(\upsilon_r, e_V) = P(\mu, e_{V0})$$

The last equation form is frequently used to compute pressure. The EOS for solid phase materials is sometimes partitioned into 2 terms, a cold pressure and a thermal pressure

$$P = P_c(\mu) + P_T(\mu, e_{V0})$$

$P_c(\mu)$ is the cold pressure hypothetically evaluated along a 0-degree-Kelvin isotherm. This is sometimes called a 0-K pressure-volume relation or cold compression curve. $P_T(\mu, e_{V0})$ is the thermal pressure component that depends on both volumetric compression and thermal state of the material.

Different forms of the EOS describe different types of materials and how their volumetric compression (or expansion) behaviors. The coefficients for each EOS model come from data-fitting, phenomenological descriptions, or derivations based on classical thermodynamics, etc.

LINEAR COMPRESSION

In low pressure processes, pressure is not significantly affected by temperature. When volumetric compression is within an elastic linear deformation range, a linear bulk modulus may be used to relate volume changes to pressure changes. Recalling the definition of an isotropic bulk modulus is [Fung 1965] $\frac{\Delta \upsilon}{\upsilon} = -\frac{P}{K}$. This may be rewritten as $P = K\left[-\frac{\Delta \upsilon}{\upsilon}\right] = K\mu$. The bulk modulus, K, thus is equivalent to C_1 in *EOS_LINEAR_POLYNOMIAL when all other coefficients are zero. This is a simplest form of an EOS. To initialize a pressure for such a material, only υ_{r0} must be defined.

INITIAL CONDITION SETTING

In general, a thermodynamic state must be defined by two state variables. The need to specify υ_{r0} and/or $e_{V0}\big|_{t=0}$ depends on the form of the EOS chosen. The user should review the equation term-by-term to establish what parameters to be initialized.

For many of the EOS available, pressure is specified (given), and the user must make an assumption on either $e_{V0}\big|_{t=0}$ or υ_{r0}. Consider two possibilities (a) $T\big|_{t=0}$ is defined or assumed from which $e_{V0}\big|_{t=0}$ may be computed, or (2) $\rho\big|_{t=0}$ is defined or assumed from which υ_{r0} may be obtained.

WHEN TO USE THE EOS

For small strains considerations, a total stress tensor may be partitioned into a deviatoric stress component and a mechanical pressure.

$$\sigma_{ij} = \sigma'_{ij} + \frac{\sigma_{kk}}{3}\delta_{ij} = \sigma'_{ij} - P\delta_{ij}$$

$$P = -\frac{\sigma_{kk}}{3} \Leftrightarrow \frac{\sigma_{kk}}{3} = -P$$

The pressure component may be written from the diagonal stress components.

Note that $\frac{\sigma_{kk}}{3} = \frac{[\sigma_{11} + \sigma_{22} + \sigma_{33}]}{3}$ is positive in tension while P is positive in compression.

Similarly the total strain tensor may be partitioned into a deviatoric strain component (volume-preserving deformation) and a volumetric deformation.

$$\varepsilon_{ij} = \varepsilon'_{ij} + \frac{\varepsilon_{kk}}{3}\delta_{ij}$$

where $\frac{\varepsilon_{kk}}{3}$ is called the mean normal strain, and ε_{kk} is called the dilatation or volume strain (change in volume per unit initial volume)

$$\varepsilon_{kk} = \frac{V - V_0}{V_0}$$

Roughly speaking, a typical convention may refer to the relation $\sigma'_{ij} = f(\varepsilon'_{ij})$ as a "constitutive equation", and $P = f(\mu, e_{V0})$ as an EOS. The use of an EOS may be omitted only when volumetric deformation is very small, and $|P| \ll |\sigma'_{ij}|$.

*EOS_LINEAR_POLYNOMIAL

Purpose: Define coefficients for linear polynomial EOS and initialize the initial thermodynamic state of the material. This is done by defining E0 and V0 below.

Card 1

Variable	EOSID	C0	C1	C2	C3	C 4	C5	C6
Type	A8	F	F	F	F	F	F	F

Card 2

Variable	E0	V0						
Type	F	F						

VARIABLE	DESCRIPTION
EOSID	Equation of state ID, a unique number or label not exceeding 8 characters must be specified.
C0	The 0^{th} polynomial equation coefficient.
C1	The 1^{st} polynomial equation coefficient (when used by itself, this is the **elastic** bulk modulus, i.e. it cannot be used for deformation that is beyond the elastic regime).
...	...
C6	The 6^{th} polynomial equation coefficient.
E0	Initial internal energy per unit reference specific volume (see the beginning of the *EOS section).
V0	Initial relative volume (see the beginning of the *EOS section).

Remarks:

1. The linear polynomial equation of state is linear in internal energy. The pressure is given by:

$$P = C_0 + C_1\mu + C_2\mu^2 + C_3\mu^3 + \left(C_4 + C_5\mu + C_6\mu^2\right)E.$$

where terms $C_2\mu^2$ and $C_6\mu^2$ are set to zero if $\mu < 0, \mu = \frac{\rho}{\rho_0} - 1$, and $\frac{\rho}{\rho_0}$ is the ratio of current density to reference density. ρ_0 is a nominal or reference density defined in the *MAT_NULL card.

The linear polynomial equation of state may be used to model gas with the gamma law equation of state. This may be achieved by setting:

$$C_0 = C_1 = C_2 = C_3 = C_6 = 0$$

and

$$C_4 = C_5 = \gamma - 1$$

where

$$\gamma = \frac{C_p}{C_v}$$

is the ratio of specific heats. Pressure for a perfect gas is then given by:

$$P = (\gamma - 1)\left\{\frac{\rho}{\rho_0}E\right\} = (\gamma - 1)\left\{\frac{e_{V0}}{\upsilon_r}\right\}$$

E has the unit of pressure (where $E = e_{V0}$ and $\upsilon_r = \rho_0/\rho$)

2. When $C_0 \neq 0$, it does not necessarily mean that the initial pressure is zero, $P_0 \neq C_0$! The initial pressure depends the values of all the coefficients and on $\mu|_{t=0}$ and $E|_{t=0}$. The pressure in a material is computed from the whole equation above, $P = P(\mu, E)$. It is always preferable to initialize the initial condition based on $\mu|_{t=0}$ and $E|_{t=0}$. The use of $C_0 \neq 0$ must be done with caution as it may change the form and behavior of the material. The safest way is to use the whole EOS equation to manually check for the pressure value. For example, for ideal gas, it is wrong to define $C_4 = C_5 = \gamma - 1$ and $C_0 \neq 0$ at the same time.

3. V0 and E0 defined in this card must be the same as the time-zero ordinates for the 2 load curves defined in the *BOUNDARY_AMBIENT_EOS card, if it is used. This is so that they would both consistently define the same initial state for a material.

*EOS_JWL

This is Equation of state Form 2.

Card	1	2	3	4	5	6	7	8
Variable	EOSID	A	B	R1	R2	OMEG	E0	VO
Type	A8	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
EOSID	Equation of state ID, a unique number or label not exceeding 8 characters must be specified.
A	
B	
R1	
R2	
OMEG	
E0	
V0	Initial relative volume.

Remarks:

The JWL equation of state defines the pressure as

$$p = A\left(1 - \frac{\omega}{R_1 V}\right)e^{-R_1 V} + B\left(1 - \frac{\omega}{R_2 V}\right)e^{-R_2 V} + \frac{\omega E}{V},$$

and is usually used for detonation products of high explosives.

***EOS_SACK_TUESDAY**

This is Equation of state Form 3.

Card	1	2	3	4	5	6	7	8
Variable	EOSID	A1	A2	A3	B1	B2	E0	V0
Type	A8	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
EOSID	Equation of state ID, a unique number or label not exceeding 8 characters must be specified.
A1	
A2	
A3	
B1	
B2	
E0	Initial internal energy
V0	Initial relative volume

Remarks:

The Sack equation of state defines pressure as

$$p = \frac{A_3}{V^{A_1}} e^{-A_2 V} \left(1 - \frac{B_1}{V}\right) + \frac{B_2}{V} E$$

and is used for detonation products of high explosives.

*EOS_GRUNEISEN

This is Equation of state Form 4.

Card	1	2	3	4	5	6	7	8
Variable	EOSID	C	S1	S2	S3	GAMAO	A	E0
Type	A8	F	F	F	F	F	F	F

Card 2

Variable	V0							
Type	F							

VARIABLE	DESCRIPTION
EOSID	Equation of state ID, a unique number or label not exceeding 8 characters must be specified.
C	
S1	
S2	
S3	
GAMAO	
A	
E0	Initial internal energy
V0	Initial relative volume

Remarks:

The Gruneisen equation of state with cubic shock velocity-particle velocity defines pressure for compressed materials as

$$p = \frac{\rho_0 C^2 \mu \left[1 + \left(1 - \frac{\gamma_0}{2} \right) \mu - \frac{a}{2} \mu^2 \right]}{\left[1 - \left(S_1 - 1 \right) \mu - S_2 \frac{\mu^2}{\mu+1} - S_3 \frac{\mu^3}{(\mu+1)^2} \right]^2} + \left(\gamma_0 + a \mu \right) E.$$

and for expanded materials as

$$p = \rho_0 C^2 \mu + \left(\gamma_0 + a\mu \right) E.$$

where C is the intercept of the v_s-v_p curve; S_1, S_2, and S_3 are the coefficients of the slope of the v_s-v_p curve; γ_0 is the Gruneisen gamma; a is the first order volume correction to $\gamma_{0;}$ and $\mu = \frac{\rho}{\rho_0} - 1.$

*EOS_RATIO_OF_POLYNOMIALS

This is Equation of state Form 5.

Card Format (I10) for card 1, **(4E20.0)** all following cards.

Card 1 1

Variable	EOSID
Type	A8

Card 2 1 2 3 4

Variable	A10	A11	A12	A13
Type	F	F	F	F

Card 3

Variable	A20	A21	A22	A23
Type	F	F	F	F

Card 4

Variable	A30	A31	A32	A33
Type	F	F	F	F

Card 5

Variable	A40	A41	A42	A43
Type	F	F	F	F

Card 6	1	2	3	4
Variable	A50	A51	A52	A53
Type	F	F	F	F

Card 7

Variable	A60	A61	A62	A63
Type	F	F	F	F

Card 8

Variable	A70	A71	A72	A73
Type	F	F	F	F

Card 9	1	2	3	4
Variable	A14	A24		
Type	F	F		

Card 10

Variable	ALPH	BETA	E0	V0
Type	F	F	F	F

VARIABLE	DESCRIPTION
EOSID	Equation of state ID, a unique number or label not exceeding 8 characters must be specified.
A10	
A11	
A12	
A13	
A20	
A21	
A22	
A23	
A30	
A31	
A32	
A33	
A40	
A41	
A42	
A43	
A50	
A51	
A52	
A53	
A60	
A61	
A62	
A63	
A70	

VARIABLE	DESCRIPTION
A71	
A72	
A73	
A14	
A24	
ALPHA	α
BETA	β
E0	Initial internal energy
V0	Initial relative volume

Remarks:

The ratio of polynomials equation of state defines the pressure as

$$p = \frac{F_1 + F_2 E + F_3 E^2 + F_4 E^3}{F_5 + F_6 E + F_7 E^2}(1 + \alpha\mu)$$

where

$$F_i = \sum_{j=0}^{n} A_{ij}\mu^j \quad n = 4 \text{if} i < 3$$

$$\mu = \frac{\rho}{\rho_0} - 1 \qquad n = 3 \text{if} i \geq 3$$

In expanded elements F_1 is replaced by $F_1' = F_1 + \beta\mu^2$. By setting coefficient $A_{10} = 1.0$, the delta-phase pressure modeling for this material will be initiated. The code will reset it to 0.0 after setting flags.

*EOS_LINEAR_POLYNOMIAL_WITH_ENERGY_LEAK

Card 1

	1	2	3	4	5	6	7	8
Variable	EOSID	C0	C1	C2	C3	C4	C5	C6
Type	A8	F	F	F	F	F	F	F

Card 2

Variable	E0	V0	LCID					
Type	F	F	I					

VARIABLE	DESCRIPTION
EOSID	Equation of state ID, a unique number or label not exceeding 8 characters must be specified.
C0	
C1	
C2	
C3	
C4	
C5	
C6	
E0	Initial internal energy
V0	Initial relative volume
LCID	Load curve ID defining the energy deposition rate

Remarks:

This polynomial equation of state, linear in the internal energy per initial volume, E, is given by

$$p = C_0 + C_1\mu + C_2\mu^2 + C_3\mu^3 + \left(C_4 + C_5\mu + C_6\mu^2\right)E$$

in which C_0, C_1, C_2, C_3, C_4, C_5, and C_6 are user defined constants and

$$\mu = \frac{1}{V} - 1.$$

where V is the relative volume. In expanded elements, we set the coefficients of μ^2 to zero, i.e.,

$$C_2 = C_6 = 0$$

Internal energy, E, is increased according to an energy deposition rate versus time curve whose ID is defined in the input.

*EOS_IGNITION_AND_GROWTH_OF_REACTION_IN_HE

Card 1	1	2	3	4	5	6	7	8
Variable	EOSID	A	B	XP1	XP2	FRER	G	R1
Type	A8	F	F	F	F	F	F	F

Card 2

Variable	R2	R3	R5	R6	FMXIG	FREQ	GROW1	EM
Type	F	F	F	F	F	F	F	F

Card 3

Variable	AR1	ES1	CVP	CVR	EETAL	CCRIT	ENQ	TMP0
Type	F	F	F	F	F	F	F	F

Card 4

Variable	GROW2	AR2	ES2	EN	FMXGR	FMNGR		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
EOSID	Equation of state ID, a unique number or label not exceeding 8 characters must be specified.
A	
B	
XP1	

VARIABLE	DESCRIPTION
XP2	
FRER	
G	
R1	
R2	
R3	
R5	
R6	
FMXIG	
FREQ	
GROW1	
EM	
AR1	
ES1	
CVP	Heat capacity of reaction products
CVR	Heat capacity of unreacted HE
EETAL	
CCRIT	
ENQ	Heat of reaction
TMP0	Initial temperature (°K)
GROW2	
AR2	
ES2	
EN	
FMXGR	
FMNGR	

Remarks:

Equation of State Form 7 is used to calculate the shock initiation (or failure to initiate) and detonation wave propagation of solid high explosives. It should be used instead of the ideal HE burn options whenever there is a question whether the HE will react, there is a finite time required for a shock wave to build up to detonation, and/or there is a finite thickness of the chemical reaction zone in a detonation wave. At relatively low initial pressures (<2-3 GPa), this equation of state should be used with material type 10 for accurate calculations of the unreacted HE behavior. At higher initial pressures, material type 9 can be used. A JWL equation of state defines the pressure in the unreacted explosive as

$$P_e = r_1 e^{-r5Ve} + r_2 e^{-r6Ve} + r_3 \frac{Te}{Ve} \left(r_3 = \omega_e \text{cvr} \right)$$

where Ve and Te are the relative volume and temperature, respectively, of the unreacted explosive. Another JWL equation of state defines the pressure in the reaction products as

$$P_p = ae^{-xp1Vp} + be^{-xp2Vp} + \frac{gTp}{Vp} \left(g = \omega_p \text{ cvp} \right)$$

where Vp and Tp are the relative volume and temperature, respectively, of the reaction products. As the chemical reaction converts unreacted explosive to reaction products, these JWL equations of state are used to calculate the mixture of unreacted explosive and reaction products defined by the fraction reacted F(F=O implies no reaction, F=1 implies complete reaction). The temperatures and pressures are assumed to be equal ($T_e = T_p$, $p_e = p_p$) and the relative volumes are additive, i.e.,

$$V = \left(1 - F\right) V_e + V_p$$

The chemical reaction rate for conversion of unreacted explosive to reaction products consists of three physically realistic terms: an ignition term in which a small amount of explosive reacts soon after the shock wave compresses it; a slow growth of reaction as this initial reaction spreads; and a rapid completion of reaction at high pressure and temperature. The form of the reaction rate equation is

$$\frac{\partial F}{\partial t} = freq\left(1 - F\right)^{frer} \left(Ve^{-1} - 1 - ccrit \right)^{eetal} \quad \text{(Ignition)}$$

$$+ \text{grow1}\left(1 - F\right)^{es1} F^{ar1} p^{em} \quad \text{(Growth)}$$

$$+ \text{grow2}\left(1 - F\right)^{es2} f^{ar2} p^{en} \quad \text{(Completion)}$$

The ignition rate is set equal to zero when F≥fmxig, the growth rate is set equal to zero when F≥fmxgr, and the completion rate is set equal to zero when F≤fmngr.

Details of the computational methods and many examples of one and two dimensional shock initiation and detonation wave calculation can be found in the references (Cochran and Chan [1979], Lee and Tarver [1980]). Unfortunately, sufficient experimental data has been obtained for only two solid explosives to develop very reliable shock initiation models: PBX-9504 (and the related HMX-based explosives LX-14,LX-10,LX-04, etc.) and LX-17 (the insensitive TATB-based explosive). Reactive flow models have been developed for other explosives (TNT, PETN, Composition B, propellants, etc.) but are based on very limited experimental data.

History variables 5 and 8 are temperature and burn fraction, respectively. See NEIPH in *DATABASE_EXTENT_BINARY if these output variables are desired in the databases for post-processing.

*EOS_TABULATED_COMPACTION

Card 1	1	2	3	4	5	6	7	8
Variable	EOSID	GAMA	E0	V0				
Type	A8	F	F	F				

Card Format (5E16.0)

Card 2	1	2	3	4	5
Variable	EV1	EV2	EV3	EV4	EV5
Type	F	F	F	F	F

Card 3

Variable	EV6	EV7	EV8	EV9	EV10
Type	F	F	F	F	F

Repeat Cards 2 and 3 for C_i, T_i, and K_i . A total of 9 cards must be defined.

VARIABLE	DESCRIPTION
EOSID	Equation of state ID, a unique number or label not exceeding 8 characters must be specified.
εV1,εV2,..εVN	Volumetric strain, ln V
C1,C2,..CN	
T1,T2,..TN	
K1,K2,..KN	Bulk unloading modulus
GAMA	γ
E0	Initial internal energy
V0	Initial relative volume

Remarks:

The tabulated compaction model is linear in internal energy. Pressure is defined by

$$p = C(\varepsilon_V) + \gamma T(\varepsilon_V)E$$

in the loading phase. The volumetric strain, ε_V is given by the natural logarithm of the relative volume V. Unloading occurs along the unloading bulk modulus to the pressure cutoff. Reloading always follows the unloading path to the point where unloading began, and continues on the loading path, see Figure 15.1. Up to 10 points and as few as 2 may be used when defining the tabulated functions. LS-DYNA will extrapolate to find the pressure if necessary.

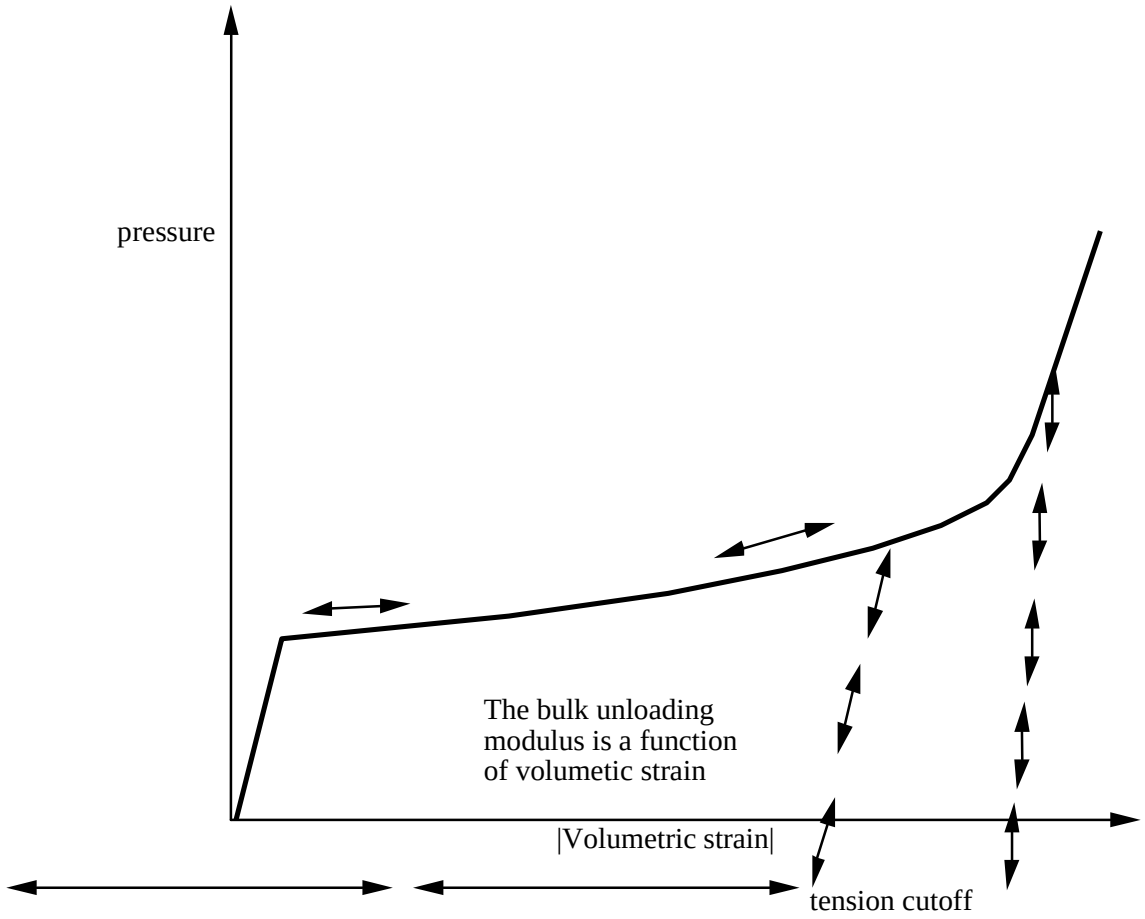


Figure 15.1. Pressure versus volumetric strain curve for Equation of state Form 8 with compaction. In the compacted states the bulk unloading modulus depends on the peak volumetric strain. Volumetric strain values should be input with correct sign (negative in compression) and in descending order. Pressure is positive in compression.

*EOS_TABULATED

This is Equation of state Form 9.

Card 1	1	2	3	4	5	6	7	8
Variable	EOSID	GAMA	E0	V0	LCC	LCT		
Type	A8	F	F	F	I	I		

Define 6 additional cards if and only if LCC and LCT equal zero. Card Format (5E16.0)

Card 2	1	2	3	4	5
Variable	EV1	EV2	EV3	EV4	EV5
Type	F	F	F	F	F

Card 3

Variable	EV6	EV7	EV8	EV9	EV10
Type	F	F	F	F	F

Repeat Cards 2 and 3 to define C_i and T_i.

VARIABLE	DESCRIPTION
EOSID	Equation of state ID, a unique number or label not exceeding 8 characters must be specified.
GAMA	γ
E0	Initial internal energy
V0	Initial relative volume
LCC	Load curve defining tabulated function C. See equation in Remarks. The abscissa values of LCC and LCT must *increase* monotonically. The definition can extend into the tensile regime.

VARIABLE	DESCRIPTION
LCT	Load curve defining tabulated function T. See equation in Remarks.
EV1, EV2, ...EVN	Volumetric strain, ln(V), where V is the relative volume. The first abscissa point, EV1, must be 0.0 or positive if the curve extends into the tensile regime with subsequent points *decreasing* monotonically.
C1,C2,..CN	Tabulated points for function C.
T1,T2,..TN	Tabulated points for function T.

Remarks:

The tabulated equation of state model is linear in internal energy. Pressure is defined by

$$P = C(\varepsilon_V) + \gamma T(\varepsilon_V) E$$

The volumetric strain, ε_V is given by the natural logarithm of the relative volume V. Up to 10 points and as few as 2 may be used when defining the tabulated functions. LS-DYNA will extrapolate to find the pressure if necessary.

*EOS_PROPELLANT_DEFLAGRATION

This Equation of state (10) has been added to model airbag propellants.

Card 1

1	2	3	4	5	6	7	8
EOSID	A	B	XP1	XP2	FRER		
A8	F	F	F	F	F		

Card 2

1	2	3	4	5	6	7	8
g	R1	R2	R3	R5			
F	F	F	F	F			

Card 3

1	2	3	4	5	6	7	8
R6	FMXIG	FREQ	GROW1	EM			
F	F	F	F	F			

Card 4

1	2	3	4	5	6	7	8
AR1	ES1	CVP	CVR	EETAL	CCRIT	ENQ	TMP0
F	F	F	F	F			

Card 5

1	2	3	4	5	6	7	8
GROW2	AR2	ES2	EN	FMXGR	FMNGR		
F	F	F	F	F	F		

VARIABLE	DESCRIPTION
EOSID	Equation of state ID, a unique number or label not exceeding 8 characters must be specified.
A	Product JWL coefficient
B	Product JWL coefficient
XP1	Product JWL coefficient
XP2	Product JWL coefficient
FRER	Unreacted Co-volume
G	Product wC_V
R1	Unreacted JWL coefficient
R2	Unreacted JWL coefficient
R3	Unreacted wC_V
R5	Unreacted JWL coefficient
R6	Unreacted JWL coefficient
FMXIG	Initial Fraction Reacted F_o
FREQ	Initial Pressure P_o
GROW1	First burn rate coefficient
EM	Pressure Exponent (1^{st} term)
AR1	Exponent on F (1^{st} term)
ES1	Exponent on (1-F) (1^{st} term)
CVP	Heat capacity products
CVR	Heat capacity unreacted
EETAL	Extra, not presently used
CCRIT	Product co-volume
ENQ	Heat of Reaction

VARIABLE	DESCRIPTION
TMP0	Initial Temperature (298°K)
GROW2	Second burn rate coefficient
AR2	Exponent on F (2nd term)
ES2	Exponent on (1-F) (2nd term)
EN	Pressure Exponent (2nd term)
FMXGR	Maximum F for 1st term
FMNGR	Minimum F for 2nd term

Remarks:

A deflagration (burn rate) reactive flow model requires an unreacted solid equation of state, a reaction product equation of state, a reaction rate law and a mixture rule for the two (or more) species. The mixture rule for the standard ignition and growth model [Lee and Tarver 1980] assumes that both pressures and temperatures are completely equilibrated as the reaction proceeds. However, the mixture rule can be modified to allow no thermal conduction or partial heating of the solid by the reaction product gases. For this relatively slow process of airbag propellant burn, the thermal and pressure equilibrium assumptions are valid. The equations of state currently used in the burn model are the JWL, Gruneisen, the van der Waals co-volume, and the perfect gas law, but other equations of state can be easily implemented. In this propellant burn, the gaseous nitrogen produced by the burning sodium azide obeys the perfect gas law as it fills the airbag but may have to be modeled as a van der Waal's gas at the high pressures and temperatures produced in the propellant chamber. The chemical reaction rate law is pressure, particle geometry and surface area dependant, as are most high-pressure burn processes. When the temperature profile of the reacting system is well known, temperature dependent Arrhenius chemical kinetics can be used.

Since the airbag propellant composition and performance data are company private information, it is very difficult to obtain the required information for burn rate modeling. However, Imperial Chemical Industries (ICI) Corporation supplied pressure exponent, particle geometry, packing density, heat of reaction, and atmospheric pressure burn rate data which allowed us to develop the numerical model presented here for their NaN_3 + Fe_2O_3 driver airbag propellant. The deflagration model, its implementation, and the results for the ICI propellant are presented in [Hallquist, et.al., 1990].

The unreacted propellant and the reaction product equations of state are both of the form:

$$p = Ae^{-R_1 V} + Be^{-R_2 V} + \frac{\omega C_v T}{V - d}$$

where p is pressure (in Mbars), V is the relative specific volume (inverse of relative density), ω is the Gruneisen coefficient, C_v is heat capacity (in Mbars -cc/cc°K), T is temperature in °K, d is

the co-volume, and *A, B, R_1* and *R_2* are constants. Setting *A*=*B*=0. yields the van der Waal's co-volume equation of state. The JWL equation of state is generally useful at pressures above several kilobars, while the van der Waal's is useful at pressures below that range and above the range for which the perfect gas law holds. Of course, setting *A*=*B*=*d*=0 yields the perfect gas law. If accurate values of ω and C_v plus the correct distribution between "cold" compression and internal energies are used, the calculated temperatures are very reasonable and thus can be used to check propellant performance.

The reaction rate used for the propellant deflagration process is of the form:

$$\frac{\partial F}{\partial t} = \underset{\text{for } 0<F<F_{limit1}}{Z(1-F)^y F^x p^w} + \underset{\text{for } F_{limit2}<F<1}{V(1-F)^u Frp^s}$$

where *F* is the fraction reacted (*F* = 0 implies no reaction, *F* = 1 is complete reaction), *t* is time, and *p* is pressure (in Mbars), *r*,*s*,*u*,*w*,*x*,*y*, F_{limit1} and F_{limit2} are constants used to describe the pressure dependence and surface area dependence of the reaction rates. Two (or more) pressure dependant reaction rates are included in case the propellant is a mixture or exhibited a sharp change in reaction rate at some pressure or temperature. Burning surface area dependencies can be approximated using the $(1\text{-F})^y$ F^x terms. Other forms of the reaction rate law, such as Arrhenius temperature dependent $e^{-E/RT}$ type rates, can be used, but these require very accurate temperatures calculations. Although the theoretical justification of pressure dependent burn rates at kilobar type pressures is not complete, a vast amount of experimental burn rate versus pressure data does demonstrate this effect and hydrodynamic calculations using pressure dependent burn accurately simulate such experiments.

The deflagration reactive flow model is activated by any pressure or particle velocity increase on one or more zone boundaries in the reactive material. Such an increase creates pressure in those zones and the decomposition begins. If the pressure is relieved, the reaction rate decreases and can go to zero. This feature is important for short duration, partial decomposition reactions. If the pressure is maintained, the fraction reacted eventually reaches one and the material is completely converted to product molecules. The deflagration front rates of advance through the propellant calculated by this model for several propellants are quite close to the experimentally observed burn rate versus pressure curves.

To obtain good agreement with experimental deflagration data, the model requires an accurate description of the unreacted propellant equation of state, either an analytical fit to experimental compression data or an estimated fit based on previous experience with similar materials. This is also true for the reaction products equation of state. The more experimental burn rate, pressure production and energy delivery data available, the better the form and constants in the reaction rate equation can be determined.

Therefore, the equations used in the burn subroutine for the pressure in the unreacted propellant

$$P_u = R1 \cdot e^{-R5 \cdot V_u} + R2 \cdot e^{-R6 \cdot V_u} + \frac{R3 \cdot T_u}{V_u - FRER}$$

where V_u and T_u are the relative volume and temperature respectively of the unreacted propellant. The relative density is obviously the inverse of the relative volume. The pressure *Pp* in the reaction products is given by:

$$P_p = A \cdot e^{-XP1 \cdot V_p} + B \cdot e^{-XP2 \cdot V_p} + \frac{G \cdot Tp}{V_p - CCRIT}$$

As the reaction proceeds, the unreacted and product pressures and temperatures are assumed to be equilibrated ($T_u = T_p = T$, $p = P_u = P_p$) and the relative volumes are additive:

$$V = (1 - F) \cdot V_u + F \cdot V_p$$

where *V* is the total relative volume. Other mixture assumptions can and have been used in different versions of DYNA2D/3D. The reaction rate law has the form:

$$\frac{\partial F}{\partial t} = \text{GROW1}(\text{P} + \text{FREQ})^{\text{EM}} (\text{F} + \text{FMXIG})^{\text{AR1}} (1 - \text{F} + \text{FMXIG})^{\text{ES1}}$$

$$+ \text{GROW2}(\text{P} + \text{FREQ})^{\text{EN}} (\text{F} + \text{FMXIG})^{\text{AR2}} (1 - \text{F} + \text{FMXIG})^{\text{ES2}}$$

If *F* exceeds FMXGR, the GROW1 term is set equal to zero, and, if *F* is less than FMNGR, the GROW2 term is zero. Thus, two separate (or overlapping) burn rates can be used to describe the rate at which the propellant decomposes.

This equation of state subroutine is used together with a material model to describe the propellant. In the airbag propellant case, a null material model (type #10) can be used. Material type #10 is usually used for a solid propellant or explosive when the shear modulus and yield strength are defined. The propellant material is defined by the material model and the unreacted equation of state until the reaction begins. The calculated mixture states are used until the reaction is complete and then the reaction product equation of state is used. The heat of reaction, ENQ, is assumed to be a constant and the same at all values of F but more complex energy release laws could be implemented.

*EOS_TENSOR_PORE_COLLAPSE

This is Equation of state Form 11.

Card 1	1	2	3	4	5	6	7	8
Variable	EOSID	NLD	NCR	MU1	MU2	IE0	EC0	
Type	A8	F	F	F	F	F	F	

VARIABLE	DESCRIPTION
EOSID	Equation of state ID, a unique number or label not exceeding 8 characters must be specified.
NLD	Virgin loading load curve ID
NCR	Completely crushed load curve ID
MU1	Excess Compression required before any pores can collapse
MU2	Excess Compression point where the Virgin Loading Curve and the Completely Crushed Curve intersect
IE0	Initial Internal Energy
EC0	Initial Excess Compression

Remarks:

The pore collapse model described in the TENSOR manual [23] is no longer valid and has been replaced by a much simpler method. This is due in part to the lack of experimental data required for the more complex model. It is desired to have a close approximation of the TENSOR model in the DYNA code to enable a quality link between them. The TENSOR model defines two curves, the virgin loading curve and the completely crushed curve as shown in Figure 15.2. It also defines the excess compression point required for pore collapse to begin (μ_1), and the excess compression point required to completely crush the material (μ_2). From this data and the maximum excess compression the material has attained (μ_{max}), the pressure for any excess compression (μ) can be determined.

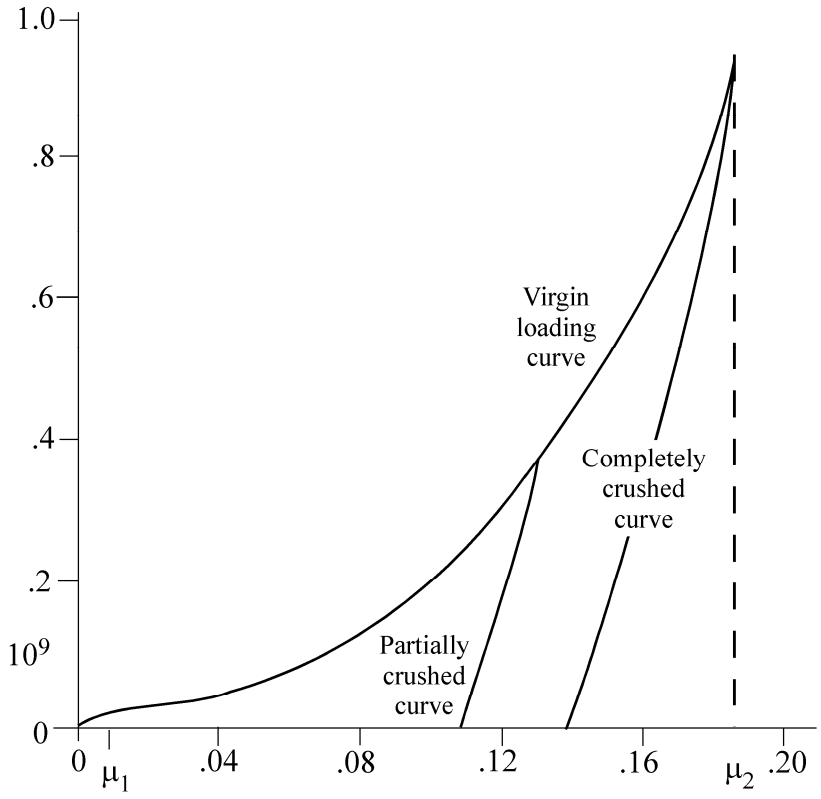


Figure 15.2. Pressure versus compaction curve.

Unloading occurs along the virgin loading curve until the excess compression surpasses μ_1. After that, the unloading follows a path between the completely crushed curve and the virgin loading curve. Reloading will follow this curve back up to the virgin loading curve. Once the excess compression exceeds μ_2, then all unloading will follow the completely crushed curve.

For unloading between μ_1 and μ_2 a partially crushed curve is determined by the relationship:

$$p_{pc}(\mu) = p_{cc}\left(\frac{(1+\mu_B)(1+\mu)}{1+\mu_{\max}} - 1\right).$$

where

$$\mu_B = P_{cc}^{-1}(P_{\max})$$

and the subscripts *pc* and *cc* refer to the partially crushed and completely crushed states, respectively. This is more readily understood in terms of the relative volume (*V*).

$$V = \frac{1}{1+\mu}$$

$$P_{pc}\ (V) = P_{cc}\left(\frac{V_B}{V_{\min}}V\right)$$

This representation suggests that for a fixed $V_{\min}\left(=\frac{1}{\mu_{\max}+1}\right)$ the partially crushed curve will separate linearly from the completely crushed curve as *V* increases to account for pore recovery in the material.

The bulk modulus *K* is determined to be the slope of the current curve times one plus the excess compression

$$K = \frac{\partial P}{\partial \mu}(1+\mu)$$

The slope $\frac{\partial P}{\partial \mu}$ for the partially crushed curve is obtained by differentiation as:

$$\frac{\partial P}{\partial \mu} = \frac{\partial P_{cc}\left(\frac{(1+\mu_B)(1+\mu)}{(1\mu_{\max})}\right)(1+\mu_B)}{\partial \mu \qquad (1+\mu_{\max})}$$

Simplifying,

$$K = \frac{\partial P_{cc}\left(\mu_a\right)}{\partial \mu}(1+\mu_a)$$

where

$$\mu_a = \frac{(1+\mu_B)(1+\mu)}{(1+\mu_{\max})} - 1.$$

The bulk sound speed is determined from the slope of the completely crushed curve at the current pressure to avoid instabilities in the time step.

The virgin loading and completely crushed curves are modeled with monotonic cubic-splines. An optimized vector interpolation scheme is then used to evaluate the cubic-splines. The bulk modulus and sound speed are derived from a linear interpolation on the derivatives of the cubic-splines.

*EOS_IDEAL_GAS

Purpose: This is equation of state form 12 for modeling ideal gas. It is an alternate approach to using *EOS_LINEAR_POLYNOMIAL with C4 = C5 = (γ-1) to model ideal gas. This has a slightly improved energy accounting algorithm.

Card 1	1	2	3	4	5	6	7	8
Variable	EOSID	CV0	CP0	CL	CQ	T0	V0	
Type	A8	F	F	F	F	F	F	

VARIABLE	DESCRIPTION
EOSID	Equation of state ID, a unique number or label not exceeding 8 characters must be specified.
CV0	Nominal constant-volume specific heat coefficient (at STP)
CP0	Nominal constant-pressure specific heat coefficient (at STP)
CL	Linear coefficient for the variations of Cv and Cp versus T
CQ	Quadratic coefficient for the variations of Cv and Cp versus T
T0	Initial temperature
V0	Initial relative volume (see the beginning of the *EOS section)

Remarks:

1. The pressure in the ideal gas law is defined as

$$p = \rho\left(C_p - C_v\right)T$$
$$C_p = C_{p0} + C_L T + C_Q T^2$$
$$C_v = C_{v0} + C_L T + C_Q T^2$$

where C_p and C_v are the specific heat capacities at constant pressure and at constant volume, respectively. ρ is the density. The relative volume, V0 parameter in the input, is defined as

$$\upsilon_r = \frac{V}{V_0} = \frac{(V/M)}{(V_0/M)} = \frac{\upsilon}{\upsilon_0} = \frac{\rho_0}{\rho}$$

where ρ_0 is a nominal or reference density defined in the *MAT_NULL card. The initial pressure can then be manually computed as

$$P\big|_{t=0} = \rho\big|_{t=0}\left(C_P - C_V\right)T\big|_{t=0}$$

$$\rho\big|_{t=0} = \left\{\frac{\rho_0}{\upsilon_r\big|_{t=0}}\right\}$$

$$P\big|_{t=0} = \left\{\frac{\rho_0}{\upsilon_r\big|_{t=0}}\right\}\left(C_P - C_V\right)T\big|_{t=0}$$

The initial relative volume, $\upsilon_r\big|_{t=0}$ (V0), initial temperature, $T\big|_{t=0}$ (T0), and heat capacity information are defined in the *EOS_IDEAL_GAS input. Note that the "reference" density is typically a density at a non-stressed or nominal stress state. The initial pressure should always be checked manually against simulation result.

2. When dealing with Eulerian/ALE models, the ideal gas model is implemented to preserve the adiabatic state during advection. The adiabatic state is conserved on the expense of a perfect internal energy conservation.

3. The ideal gas model is good for low density gas only. Deviation from the ideal gas behavior may be indicated by the compressibility factor defined as

$$Z = \frac{P\upsilon}{RT}$$

When Z deviates from 1, the gas behavior deviates from ideal.

4. V0 and T0 defined in this card must be the same as the time-zero ordinates for the 2 load curves defined in the *BOUNDARY_AMBIENT_EOS card, if it is used. This is so that they both would consistently define the same initial state for a material.

*EOS_JWLB

This is Equation of state Form 14. The JWLB (Jones-Wilkens-Lee-Baker) equation of state, developed by Baker [1991] and further described by Baker and Orosz [1991], describes the high pressure regime produced by overdriven detonations while retaining the low pressure expansion behavior required for standard acceleration modeling. The derived form of the equation of state is based on the JWL form due to its computational robustness and asymptotic approach to an ideal gas at high expansions. Additional exponential terms and a variable Gruneisen parameter have been added to adequately describe the high-pressure region above the Chapman-Jouguet state.

Card 1

Variable	EOSID	A1	A2	A3	A4	A5		
Type	A8	F	F	F	F	F		

Card 2

Variable	R1	R2	R3	R4	R5			
Type	F	F	F	F	F			

Card 3

Variable	AL1	AL2	AL3	AL4	AL5			
Type	F	F	F	F	F			

Card 4

Variable	BL1	BL2	BL3	BL4	BL5			
Type	F	F	F	F	F			

Card 5	1	2	3	4	5	6	7	8
Variable	RL1	RL2	RL3	RL4	RL5			
Type	F	F	F	F	F			

Card 6

Variable	C	OMEGA	E	V0				
Type	I	F	F	F				

VARIABLE	DESCRIPTION
EOSID	Equation of state ID, a unique number or label not exceeding 8 characters must be specified.
A1	Equation of state coefficient, see below.
A2	Equation of state coefficient, see below.
A3	Equation of state coefficient, see below.
A4	Equation of state coefficient, see below.
A5	Equation of state coefficient, see below.
R1	Equation of state coefficient, see below.
R2	Equation of state coefficient, see below.
R3	Equation of state coefficient, see below.
R4	Equation of state coefficient, see below.
R5	Equation of state coefficient, see below.
AL1	$A_{\lambda 1}$, equation of state coefficient, see below.
AL2	$A_{\lambda 2}$, equation of state coefficient, see below.
AL3	$A_{\lambda 3}$, equation of state coefficient, see below.

VARIABLE	DESCRIPTION
AL4	$A_{\lambda 4}$, equation of state coefficient, see below.
AL5	$A_{\lambda 5}$, equation of state coefficient, see below.
BL1	$B_{\lambda 1}$, equation of state coefficient, see below.
BL2	$B_{\lambda 2}$, equation of state coefficient, see below.
BL3	$B_{\lambda 3}$, equation of state coefficient, see below.
BL4	$B_{\lambda 4}$, equation of state coefficient, see below.
BL5	$B_{\lambda 5}$, equation of state coefficient, see below.
RL1	$R_{\lambda 1}$, equation of state coefficient, see below.
RL2	$R_{\lambda 2}$, equation of state coefficient, see below.
RL3	$R_{\lambda 3}$, equation of state coefficient, see below.
RL4	$R_{\lambda 4}$, equation of state coefficient, see below.
RL5	$R_{\lambda 5}$, equation of state coefficient, see below.
C	Equation of state coefficient, see below.
OMEGA	Equation of state coefficient, see below.
E	Energy density per unit initial volume
V0	Initial relative volume.

Remarks:

The JWLB equation-of-state defines the pressure as

$$p = \sum_{i=1}^{5} A_i \left(1 - \frac{\lambda}{R_i V}\right) e^{-R_i V} + \frac{\lambda E}{V} + C\left(1 - \frac{\lambda}{\omega}\right) V^{-(\omega+1)}$$

$$\lambda = \sum_{i=1}^{5} \left(A_{\lambda i} V + B_{\lambda i}\right) e^{-R_{\lambda i} V} + \omega$$

where V is the relative volume, E is the energy per unit initial volume, and A_i, R_i, $A_{\lambda i}$, $B_{\lambda i}$, $R_{\lambda i}$, C, and ω are input constants defined above.

JWLB input constants for some common explosives as found in Baker and Stiel [1997] are given in the following table.

	TATB	LX-14	PETN	TNT	Octol 70/30
ρ0 (g/cc)	1.800	1.821	1.765	1.631	1.803
E0 (Mbar)	.07040	.10205	.10910	.06656	.09590
*D*CJ (cm/μs)	.76794	.86619	.83041	.67174	.82994
*P*CJ (Mbar)	.23740	.31717	.29076	.18503	.29369
A1 (Mbar)	550.06	549.60	521.96	490.07	526.83
A2 (Mbar)	22.051	64.066	71.104	56.868	60.579
A3 (Mbar)	.42788	2.0972	4.4774	.82426	.91248
A4 (Mbar)	.28094	.88940	.97725	.00093	.00159
R_1	16.688	34.636	44.169	40.713	52.106
R_2	6.8050	8.2176	8.7877	9.6754	8.3998
R_3	2.0737	20.401	25.072	2.4350	2.1339
R_4	2.9754	2.0616	2.2251	.15564	.18592
C (Mbar)	.00776	.01251	.01570	.00710	.00968
ω	.27952	.38375	.32357	.30270	.39023
$A\lambda_1$	1423.9	18307.	12.257	.00000	.011929
$B\lambda_1$	14387.	1390.1	52.404	1098.0	18466.
$R\lambda_1$	19.780	19.309	43.932	15.614	20.029
$A\lambda_2$	5.0364	4.4882	8.6351	11.468	5.4192
$B\lambda_2$	-2.6332	-2.6181	-4.9176	-6.5011	-3.2394
$R\lambda_2$	1.7062	1.5076	2.1303	2.1593	1.5868

*EOS_GASKET

This model works with solid elements and the thick shell using selective reduced 2 x 2 integration (ELFORM=2 on SECTION_TSHELL) to model the response of gaskets. For the thick shell only, it is completely decoupled from the shell material, i.e., in the local coordinate system of the shell, this model defines the normal stress, σ_{zz}, and doesn't change any of the other stress components. The model is a reduction of the *MAT_GENERAL_NONLINEAR_6DOF_DISCRETE_BEAM.

Card 1

1	2	3	4	5	6	7	8
Variable	EOSID	LCID1	LCID2	LCID3	LCID4			
Type	A8	I	I	I	I			

Card 2

Variable	UNLOAD	ET	DMPF	TFS	CFS	LOFFSET	IVS	
Type	F	F	F	F	F	F	F	

VARIABLE	DESCRIPTION
EOSID	Equation of state ID, a unique number or label not exceeding 8 characters must be specified.
LCID1	Load curve for loading.
LCID2	Load curve for unloading.
LCID3	Load curve for damping as a function of volumetric strain rate.
LCID4	Load curve for scaling the damping as a function of the volumetric strain.
UNLOAD	Unloading option (See Volume II, Figure 119.1.): EQ.0.0: Loading and unloading follow loading curve EQ.1.0: Loading follows loading curve, unloading follows unloading curve. The unloading curve ID if undefined is taken as the loading curve. EQ.2.0: Loading follows loading curve, unloading follows unloading stiffness, KT or KR, to the unloading curve. The loading and unloading curves may only intersect at the origin of the axes.

VARIABLE	DESCRIPTION
	EQ.3.0: Quadratic unloading from peak displacement value to a permanent offset.
DMPF	Damping factor for stability. Values in the neighborhood of unity are recommended. The damping factor is properly scaled to eliminate time step size dependency. Also, it is active if and only if ET is defined.
TFS	Tensile failure strain.
CFS	Compressive failure strain.
OFFSET	Offset factor between 0 and 1.0 to determine permanent set upon unloading if the UNLOAD=3.0. The permanent sets in compression and tension are equal to the product of this offset value and the maximum compressive and tensile displacements, respectively.
IVS	Initial volume strain.

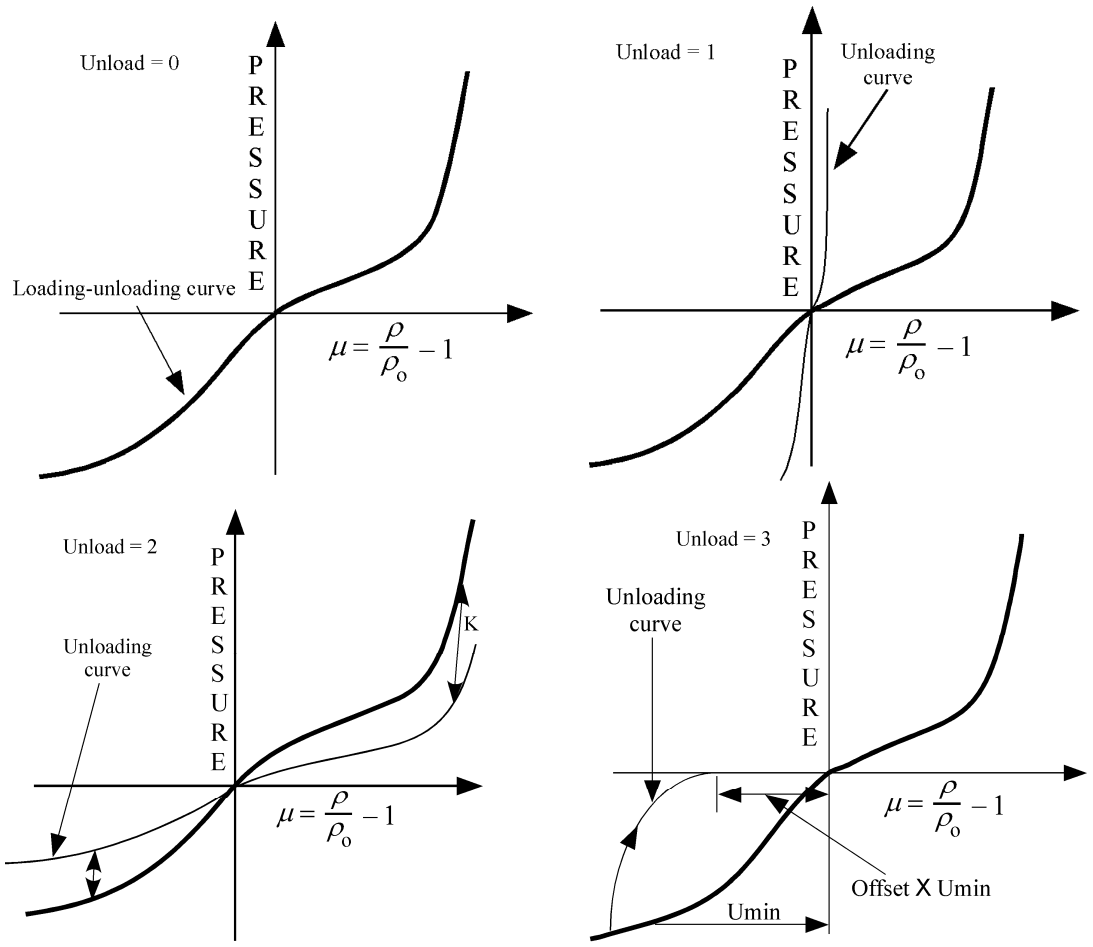


Figure 15.3. Load and Unloading behavior.

*EOS_USER_DEFINED

These are equations of state 21-30. The user can supply his own subroutines. See also Appendix B. The keyword input has to be used for the user interface with data.

Card 1	1	2	3	4	5	6	7	8
Variable	EOSID	EOST	LMC	NHV	IVECT	EO	VO	BULK
Type	A8	I	I	I	I	F	F	F

Define LMC material parameters using 8 parameters per card.

Card 2	1	2	3	4	5	6	7	8
Variable	P1	P2	P3	P4	P5	P6	P7	P8
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
EOSID	Equation of state ID, a unique number or label not exceeding 8 characters must be specified.
EOST	User equation of state type (21-30 inclusive). A number between 21 and 30 has to be chosen.
LMC	Length of material constant array which is equal to the number of material constants to be input. (LMC ≤ 48)
NHV	Number of history variables to be stored, see Appendix D.
IVECT	Vectorization flag (on=1). A vectorized user subroutine must be supplied.
EO	Initial internal energy.
V0	Initial relative volume.
BULK	Bulk modulus. This value is used in the calculation of the contact surface stiffness.

VARIABLE	DESCRIPTION
P1	First material parameter.
P2	Second material parameter.
P3	Third material parameter.
P4	Fourth material parameter.
.	.
.	.
.	.
PLMC	LMCth material parameter.

*HOURGLASS

***HOURGLASS**

Purpose: Define hourglass and bulk viscosity properties which are referenced via HGID in the *PART command. Properties specified here, when invoked for a particular part, override those in *CONTROL_HOURGLASS and *CONTROL_BULK_VISCOSITY.

An additional option **_TITLE** may be appended to ***HOURGLASS** keywords. If this option is used then an additional line is read for each section in 80a format which can be used to describe the section. At present LS-DYNA does not make use of the title. Inclusion of titles gives greater clarity to input decks.

Card 1	1	2	3	4	5	6	7	8
Variable	HGID	IHQ	QM	IBQ	Q1	Q2	QB/VDC	QW
Type	A8	I	F	I	F	F	F	F
Default	0		.10	0	1.5	0.06	QM/0.	QM
Remark		1,6	2,4,7	3	3	3	5	5

VARIABLE — **DESCRIPTION**

HGID — Hourglass ID. A unique number or label not exceeding 8 characters must be specified. This ID is referenced by HGID in the *PART command.

IHQ — Hourglass control type. For solid elements six options are available. For quadrilateral shell and membrane elements the hourglass control is based on the formulation of Belytschko and Tsay, i.e., options 1-3 are identical, and options 4-6 are identical:

EQ.0: default=1 regardless of IHQ in *CONTROL_HOURGLASS,
EQ.1: standard LS-DYNA viscous form,
EQ.2: Flanagan-Belytschko viscous form,
EQ.3: Flanagan-Belytschko viscous form with exact volume integration for solid elements,
EQ.4: Flanagan-Belytschko stiffness form,

VARIABLE	DESCRIPTION
	EQ.5: Flanagan-Belytschko stiffness form with exact volume integration for solid elements. EQ.6: Belytschko-Bindeman [1993] assumed strain co-rotational stiffness form for 2D and 3D solid elements only. This form is available for explicit and IMPLICIT solution methods. In fact, type 6 or 7 is mandatory for the implicit options. EQ.7: Linear total strain form of type 6 hourglass control. This form is available for explicit and implicit solution method (See remark 6 below). EQ:8: Applicable to the type 16 fully integrated shell element. IHQ=8 activates the full projection warping stiffness for accurate solutions. A speed penalty of 25% is common for this option. EQ.9: Puso [2000] enhanced assumed strain stiffness form for 3D hexahedral elements. This form is available for explicit and implicit solution methods, hence it is an alternative to the Belytschko-Bindeman hourglass type 6 for implicit simulations.
	A discussion of the viscous and stiffness hourglass control for shell elements follows at the end of this section.
QM	Hourglass coefficient. Values of QM that exceed .15 may cause instabilities. The recommended default applies to all options except for IHQ=6. The stiffness forms, however, can stiffen the response especially if deformations are large and therefore should be used with care. For the shell and membrane elements QM is taken as the membrane hourglass coefficient, the bending as QB, and warping as QW. These coefficients can be specified independently, but generally, QM=QB=QW, is adequate. For type 6 solid element hourglass control, see remark 4 below. For hourglass type 9, see Remark 8.
IBQ	Bulk viscosity type (See Remark 3 below.): EQ.1: standard LS-DYNA.
Q1	Quadratic bulk viscosity coefficient.
Q2	Linear bulk viscosity coefficient.
QB	Hourglass coefficient for shell bending. The default: QB=QM. (See Remark 4).
VDC	Viscous damping coefficient for types 6 and 7 hourglass control.
QW	Hourglass coefficient for shell warping. The default: QB=QW.

Remarks:

1. Viscous hourglass control is recommended for problems deforming with high velocities. Stiffness control is often preferable for lower velocities, especially if the number of time

steps are large. For solid elements the exact volume integration provides some advantage for highly distorted elements.

2. For automotive crash the stiffness form of the hourglass control with a coefficient of 0.05 is preferred by many users.

3. Bulk viscosity is necessary to propagate shock waves in solid materials and therefore applies only to solid elements. Generally, the default values are okay except in problems where pressures are very high, larger values may be desirable. In low density foams, it may be necessary to reduce the viscosity values since the viscous stress can be significant. It is not advisable to reduce it by more than an order of magnitude.

4. Type 6 hourglass control is for 2D and 3D solid elements only. Based on elastic constants and an assumed strain field, it produces accurate coarse mesh bending results for elastic material when QM=1.0. For plasticity models with a yield stress tangent modulus that is much smaller than the elastic modulus, a smaller value of QM (0.001 to 0.1) may produce better results. For any material, keep in mind that the stiffness is based on the elastic constants, so if the material softens, a QM value smaller than 1.0 may work better. For anisotropic materials, an average of the elastic constants is used. For fluids modeled with null material, type 6 hourglass control is viscous and is scaled to the viscosity coefficient of the material (see *MAT_NULL).

5. In part, the computational efficiency of the Belytschko-Lin-Tsay and the under integrated Hughes-Liu shell elements are derived from their use of one-point quadrature in the plane of the element. To suppress the hourglass deformation modes that accompany one-point quadrature, hourglass viscous or stiffness based stresses are added to the physical stresses at the local element level. The discussion of the hourglass control that follows pertains to all one point quadrilateral shell and membrane elements in LS-DYNA.

   The hourglass shape vector τ_I is defined as

$$\tau_I = h_I - \left(h_J \hat{x}_{aJ}\right) B_{aI}$$

   where, $\hat{x}_{aJ}$ are the element coordinates in the local system at the Ith element node, B_{aI} is the strain displacement matrix, and hourglass basis vector is:

$$h = \begin{bmatrix} +1 \\ -1 \\ +1 \\ -1 \end{bmatrix}$$

   is the basis vector that generates the deformation mode that is neglected by one-point quadrature. In the above equations and the reminder of this subsection, the Greek subscripts have a range of 2, e.g., $\hat{x}_{aI} = \left(\hat{x}_{1I}, \hat{x}_{2I}\right) = \left(\hat{x}_I, \hat{y}_I\right)$.

   The hourglass shape vector then operates on the generalized displacements to produce the generalized hourglass strain rates

$$\dot{q}_\alpha^M = \tau_I \hat{v}_{\alpha I}$$

$$\dot{q}_\alpha^B = \tau_I \hat{\theta}_{\alpha I}$$

$$\dot{q}_3^W = \tau_I \hat{v}_{zI}$$

where the superscripts M, B, and W denote membrane, bending, and warping modes, respectively. The corresponding hourglass stress rates are then given by

$$\dot{Q}_\alpha^M = \frac{QM \cdot EtA}{8} B_{\beta I} B_{\beta I} \dot{q}_\alpha^M$$

$$\dot{Q}_\alpha^B = \frac{QB \cdot Et^3 A}{192} B_{\beta I} B_{\beta I} \dot{q}_\alpha^B$$

$$\dot{Q}_3^W = \frac{QW \cdot \kappa G t^3 A}{12} B_{\beta I} B_{\beta I} \dot{q}_3^B$$

where t is the shell thickness. The hourglass coefficients: QM, QB, and QW are generally assigned values between 0.05 and 0.10.

Finally, the hourglass stresses which are updated using the time step, Δt, from the stress rates in the usual way, i.e.,

$$Q^{n+1} = Q^n + \Delta t \dot{Q}$$

and the hourglass resultant forces are then

$$\hat{f}_{\alpha I}^H = \tau_I Q_\alpha^M$$

$$\hat{m}_{\alpha I}^H = \tau_I Q_\alpha^B$$

$$\hat{f}_{3I}^H = \tau_I Q_3^W$$

where the superscript H emphasizes that these are internal force contributions from the hourglass deformations.

6. IHG=7 is a linear total strain formulation of the Belytschko-Bindeman [1993] stiffness form for 2D and 3D solid elements. This linear form was developed for visco-elastic material and guarantees that an element will spring back to its initial shape regardless of the severity of deformation.

7. The default value for QM is 0.1 unless superseded by a nonzero value of QH in *CONTROL_HOURGLASS. A nonzero value of QM supersedes QH.

8. Hourglass type 9 is available for hexahedral elements and is based on physical stabilization using an enhanced assumed strain method. In performance it is similar to the Belytschko-Bindeman hourglass formulation (type 6) but gives more accurate results for distorted meshes, e.g., for skewed elements. If QM=1.0, it produces accurate coarse bending results for elastic materials. The hourglass stiffness is by default based on elastic properties, hence the QM parameter should be reduced to about 0.1 for plastic materials in order not to stiffen the structure during plastic deformation. For materials 3, 18 and 24 there is the option to use a negative value of QM. With this option, the hourglass stiffness is based on the current material properties, i.e., the plastic tangent modulus, and scaled by $|QM|$.

*HOURGLASS

*INCLUDE

Purpose: The keyword *INCLUDE provides a means of reading independent input files containing model data. The file contents are placed directly at the location of the *INCLUDE line.

***INCLUDE_{***OPTION***}**

INCLUDE_COMPENSATION_OPTION*

***INCLUDE_{*OPTION*}**

Available options include:

<BLANK>

BINARY

NASTRAN

PATH

STAMPED_PART_{*OPTION1*}_{*OPTION2*}_{*OPTION3*}

OPTION1: **_SET**

OPTION2: **_MATRIX**

OPTION3: **_INVERSE**

STAMPED_SET

TRANSFORM

TRANSFORM_BINARY

The BINARY and TRANSFORM_BINARY options specify that the initial stress file, DYNAIN, is written in a binary format. See the keyword *INTERFACE_SPRINGBACK.

The PATH option defines a directory in which to look for the include files. The program always searches the local directory first. If an include file is not found and the filename has no path, the program will search for it in all the directories defined by *INCLUDE_PATH. Multiple paths can defined with one *INCLUDE_PATH definition, i.e.,

```
*INCLUDE_PATH
Directory_path1
Directory_path2
Directory_path3
```

Directory paths are read until the next "*" card is encountered. A directory path can have up to 80 characters.

The STAMPED_PART option allows the plastic strain and thickness distribution of the stamping simulation to be mapped onto a part in the crash model.

1. When option 1, _SET is used, the PID will be part set ID. All the parts included in this set will be considered in this mapping.
2. When option 2, _MATRIX is used, translation matrix will be read directly and the orientation nodes will be ignored.
3. When option 3, _INVERSE (must be used with _MATRIX) is used, the matrix will be reversed first.

When STAMPED_SET is used, the target is a part set ID. Between the stamped part and the crash part, note the following points:

1. The outer boundaries of the parts do not need to match since only the regions of the crash part which overlap the stamped part are initialized.
2. Arbitrary mesh patterns are assumed.
3. Element formulations can change.
4. Three nodes on each part are used to reorient the stamped part for the mapping of the strain and thickness distributions. After reorientation, the three nodes on each part should approximately coincide.
5. The number of in plane integrations points can change.
6. The number of through thickness integration points can change. Full interpolation is used.
7. The node and element ID's between the stamped part and the crash part do not need to be unique.

The TRANSFORM option allows for node, element, and set ID's to be offset and for coordinates and constitutive parameters to be transformed and scaled.

The card is required.

Card 1 1

Variable	FILENAME
Type	C

If the *INCLUDE command is used without options, multiple filenames can be specified, i.e.,

*INCLUDE
Filename1
Filename2
Filename3

which are processed sequentially. Filenames are read until the next "*" card is encountered.

If the NASTRAN option is active then define the following input line.
NASTRAN option

Card 2	1	2	3	4	5	6	7	8
Variable	BEAMDF	SHELLDF	SOLIDDF					
Type	I	I	I					
Default	2	21	18					

If the STAMPED_PART option is active then define the following input.
STAMPED_PART option

Card 2	1	2	3	4	5	6	7	8
Variable	PID	THICK	PSTRN	STRAIN	STRESS	INCOUT		RMAX
Type	I	I	I	I	I	I		F
Default	none	0	0	0	0	0		10.0

If the _MATRIX option is inactive define the follow card.

Card 3	1	2	3	4	5	6	7	8
Variable	N1S	N2S	N3S	N1C	N2C	N3C	TENSOR	THKSCL
Type	I	I	I	I	I	I	I	F
Default	0	0	0	0	0	0	0	1.0
Remarks	2	2	2	2	2	2	4	

If the _MATRIX option is active define the follow three cards.

Card 3	1	2	3	4	5	6	7	8
Variable	R11	R12	R13	XP				
Type	F	F	F	F				
Default	0	0	0	0				
Remarks	2	2	2	2				

Card 4

Variable	R21	R22	R23	YP				
Type	F	F	F	F				
Default	0	0	0	0				
Remarks	2	2	2	2				

Card 5	1	2	3	4	5	6	7	8
Variable	R31	R32	R33	ZP				
Type	F	F	F	F				
Default	0	0	0	0				
Remarks	2	2	2	2				

Optional Cards

Card 4	1	2	3	4	5	6	7	8
Variable	ISYM	IAFTER	PERCELE					
Type	I	I	F					

Card 5

Variable	X01	Y01	Z01					
Type	F	F	F					

Card 6

Variable	X02	Y02	Z02	X03	Y03	Z03		
Type	F	F	F	F	F	F		

If the TRANSFORM option is active then define the following input
TRANSFORM option

Card 2

Variable	IDNOFF	IDEOFF	IDPOFF	IDMOFF	IDSOFF	IDFOFF	IDDOFF	
Type	I	I	I	I	I	I	I	

Card 3

Variable	IDROFF							
Type	I							

Card 4

Variable	FCTMAS	FCTTIM	FCTLEN	FCTTEM	INCOUT1			
Type	F	F	F	A	I			

Card 5

Variable	TRANID							
Type	I							
Default	0							

VARIABLE	DESCRIPTION
FILENAME	File name of file to be included in this keyword file, 80 characters maximum. If the STAMPED_PART option is active, this is the DYNAIN file containing the results from metal stamping.
BEAMDF	LS-DYNA beam element type. Defaults to type 2.
SHELLDF	LS-DYNA shell element type. Defaults to type 21.

VARIABLE	DESCRIPTION
SOLIDDF	LS-DYNA solid element type. Defaults to type 18.
PID	Part ID of crash part for remapping.
THICK	Thickness remap: EQ.0: map thickness EQ.1: do not map thickness EQ.2: average value inside a circle defined by RMAX
PSTRN	Plastic strain remap: EQ.0: map plastic strain EQ.1: do not plastic strain EQ.2: average value inside a circle defined by RMAX
STRAIN	Strain remap: EQ.0: map strains EQ.1: do not map strains
STRESS	Stress tensor remap: EQ.0: map stress tensor EQ.1: do not map stress tensor, only history EQ.-1: map stress tensor in an internal large format (binary files) EQ.-3: do not map stress tensor in an internal large format, only history (binary files)
THKSCL	Thickness scale factor.
INCOUT	EQ.1: to save the mapped data to a file called dyna.inc, which contains the mapped data for the part that is being mapped. This option is useful to do mapping using INCLUDE_STAMPED_PART and then save the mapped data for future use. When INCOUT is set to 2, the output file is in dynain format and the file name is dynain_xx (xx is the part or part set id); and when INCOUT is set to 3, the output file is in NASTRAN format, and the file name is: nastran_xx. EQ.2: to save the mapped data for the specified part (PID) to a file called dynain_PID. EQ.3: to save the mapped data for the specified part (PID) to a file called nastran_PID (in nastran format)
RMAX	Search radius
N1S	First of 3 nodes needed to reorient the stamped part.
N2S	Second of 3 nodes needed to reorient the stamped part.

VARIABLE	DESCRIPTION
N3S	Third of 3 nodes needed to reorient the stamped part.
N1C	First of 3 nodes needed to reorient the crash model part.
N2C	Second of 3 nodes needed to reorient the crash model part.
N3C	Third of 3 nodes needed to reorient the crash model part.
TENSOR	Tensor remap: EQ.0: map tensor data from history variables. (See Remark 4.) EQ.1: do not map tensor data from history variables
R11,R12,…R33	Components of the transformation matrix.
XP,YP,ZP	Translational distance.
X03,Y03,Z03	Third point in the symmetric plane
ISYM	Symmetric switch EQ.0: no symmetric mapping EQ.1: yz plane symmetric mapping EQ.2: zx plane symmetric mapping EQ.3: zx and yz planes symmetric mapping EQ.4: user defined symmetric plane mapping
IAFTER	Mirroring sequence switch EQ.0: generate a symmetric part before transformation EQ.1: generate a symmetric part after transformation
PERCELE	Percentage of elements that should be mapped to proceed (default=0); otherwise an error termination occurs.
X01, Y01, Z01	First point in the symmetric plane (required if ISYM.NE.0)
X02,Y02,Z02	Second point in the symmetric plane
X03,Y03,Z03	Third point in the symmetric plane
IDNOFF	Offset to node ID.
IDEOFF	Offset to element ID.
IDPOFF	Offset to part ID, nodal rigid body ID, constrained nodal set ID, Rigidwall ID, and *DATABASE_CROSS_SECTION.
IDMOFF	Offset to material ID and equation of state ID.
IDSOFF	Offset to set ID.

IDFOFF	Offset to function ID or table ID.
IDDOFF	Offset to any ID defined through DEFINE except the FUNCTION option.
IDROFF	Used for all offsets except for those listed above.
FCTMAS	Mass transformation factor. For example, FCTMAS=1000. When the original mass units are in tons and the new unit is kg.
FCTTIM	Time transformation factor. For example, FCTTIM=.001 when the original time units are in milliseconds and the new time unit is seconds.

VARIABLE	DESCRIPTION
INCOUT1	Set to 1 for the creation of a file, DYNA.INC, which contains the transformed data. The data in this file can be used in future include files and should be checked to ensure that all the data was transformed correctly.
TRANID	Transformation ID, if 0 no transformation will be applied. See the input DEFINE_TRANSFORMATION.
FCTLEN	Length transformation factor.
FCTTEM	Temperature transformation factor consisting of a four character flag: FtoC (Fahrenheit to Centigrade), CtoF, FtoK, KtoF, KtoC, and CtoK.

Remarks:

1. To make the input file easy to maintain, this keyword allows the input file to be split into subfiles. Each subfile can again be split into sub-subfiles and so on. This option is beneficial when the input data deck is very large. Consider the following example:

```
*TITLE
full car model
*INCLUDE
carfront.k
*INCLUDE
carback.k
*INCLUDE
occupantcompartment.k
*INCLUDE
dummy.k
*INCLUDE
bag.k
*CONTACT
• • •
*END
```

Note that the command *END terminates the include file.

The carfront.k file can again be subdivided into rightrail.k, leftrail.k, battery.k, wheel-house.k, shotgun.k, etc.. Each *.k file can include nodes, elements, boundary conditions, initial conditions, and so on.

```
*INCLUDE
rightrail.k
*INCLUDE
leftrail.k
*INCLUDE
battery.k
*INCLUDE
wheelhouse.k
*INCLUDE
shotgun.k
• • •
```

. . .
*END

2. When defining *INCLUDE_STAMPED_PART the target mesh must be read in before the include stamped part.

   n1s, n2s, n3s, n1c, n2c, n3c are used for transforming the stamped part to the crashed part, such that it is in the same position as the crashed part. If the stamped part is in the same position as the crashed part then n1s, n2s, n3s, n4s, n1c, n2c, n3c can all be set to 0. Note: If these 6 nodes are input as 0, LS-DYNA will not transform the stamped part.

   When symmetric mapping is used (ISYM is not zero), the three points should not be in one line.

   If ISYM = 0, 1, 2, or 3, only the first point (X01,Y01, Z01) is needed

   If ISYM = 4, all the three points are needed

3. All filenames and paths are limited to a total of 240 characters in length, and to a limit of 80 characters per line. To continue a filename or path over more than one line use '+' (note space before plus sign) on the end of the line.

4. Certain material models (notably Material 190) have tensor data stored within the history variables. Within material subroutines this data is typically stored in element local coordinate systems. In order to properly map this information between models it is necessary to have the tensor data present on the *INITIAL_STRESS_SHELL card and have it stored in global coordinates. During mapping the data is then converted into the local coordinate system of the crash mesh. This data can be dumped into the dynain file that is created at termination time if the parameter FTENSOR is set to 0 on the *INTERFACE_SPRINGBACK_DYNA3D card. Currently, the only material model that supports mapping of element history tensor data is Material 190.

INCLUDE_COMPENSATION_OPTION*

Purpose: To include geometry information for springback compensation. This keyword must be used with *INTERFACE_COMPENSATION_NEW.

Options available include:

BLANK_BEFORE_SPRINGBACK

BLANK_AFTER_SPRINGBACK

DESIRED_BLANK_SHAPE

COMPENSATED_SHAPE

CURRENT_TOOLS

*INCLUDE_COMPENSATION_BLANK_BEFORE_SPRINGBACK

Card 1 1

Variable	FILENAME
Type	C
Default	./blank0.tmp

*INCLUDE_COMPENSATION_BLANK_AFTER_SPRINGBACK

Card 2 1

Variable	FILENAME
Type	C
Default	./spbk.tmp

*INCLUDE_COMPENSATION_DESIRED_BLANK_SHAPE

Card 3 1

Variable	FILENAME
Type	C
Default	./reference0.dat

INCLUDE_COMPENSATION_COMPENSATED_SHAPE

Card 4 1

Variable	FILENAME
Type	C
Default	./reference1.dat

*INCLUDE_COMPENSATION_CURRENT_TOOLS

Card 5 1

Variable	FILENAME
Type	C
Default	./rigid.tmp

Remarks:

When the option BLANK_BEFORE_SPRINGBACK is used, the included file is the ‘dynain’ file just before springback prediction. For the first iteration, it might be the same as reference0.dat. It can also be the mesh after coarsening and before springback.

When the option BLANK_AFTER_SPRINBACK is used, the included file is the ‘dynain’ file just after springback prediction.

When the option DESIRED_BLANK_SHAPE is used, the included file is the ‘dynain’ file after trimming in the first iteration.

When the option COMPENSATED_SHAPE is used, the included file is the ‘dynain’ file. For the first iteration, it is the same as reference0.dat; and for the following iterations, this file is obtained from the ‘disp.tmp’ which is generated as an output file during the previous compensation iteration.

When the option CURRENT_TOOLS is used, the included file is the file containing the tool mesh. The tool mesh after each forming simulation. The draw bead nodes have to be included in this file so that they will be modified with the rigid tools. If the file is named rigid0.tmp the elements of the tools get refined along the outline of the part.

*INITIAL

The keyword *INITIAL provides a way of initializing velocities and detonation points. The keyword control cards in this section are defined in alphabetical order:

***INITIAL_ALE_MAPPING**

***INITIAL_AXIAL_FORCE_BEAM**

***INITIAL_DETONATION**

***INITIAL_FOAM_REFERENCE_GEOMETRY**

***INITIAL_GAS_MIXTURE**

***INITIAL_MOMENTUM**

***INITIAL_PWP_DEPTH**

***INITIAL_STRAIN_SHELL**

***INITIAL_STRAIN_SOLID**

***INITIAL_STRESS_BEAM**

***INITIAL_STRESS_DEPTH**

***INITIAL_STRESS_SECTION**

***INITIAL_STRESS_SHELL**_{*OPTION*}

***INITIAL_STRESS_SOLID**

***INITIAL_STRESS_TSHELL**

***INITIAL_TEMPERATURE**_*OPTION*

***INITIAL_VEHICLE_KINEMATICS**

Two mutually exclusive methods are available for initial velocity generation:

***INITIAL_VELOCITY**

***INITIAL_VELOCITY_NODE**

***INITIAL_VELOCITY_RIGID_BODY**

and:

***INITIAL_VELOCITY_GENERATION**

The latter is convenient for specifying initial rotational velocities about arbitrary axes. These methods for velocity generation must not be mixed in a single input deck.

***INITIAL_VOID**_*OPTION*

***INITIAL_VOLUME_FRACTION**

***INITIAL_VOLUME_FRACTION_GEOMETRY**

*INITIAL_ALE_MAPPING

Purpose: This card initializes the current ALE run with data from the last cycle of a previous ALE run. Data are read from a mapping file called by the prompt "map=" on the command line (see remarks 4 and 5). To map data histories (not just the last cycle) to a region of selected elements (not all the ALE domain) see *BOUNDARY_ALE_MAPPING.

Card Format

Card 1

Variable	PID	TYP	AMMSID					
Type	I	I	I					
Default	none	none	none					

Card 2

Variable	XO	YO	ZO	VECID	ANGLE			
Type	F	F	F	I	F			
Default	0.0	0.0	0.0	none	none			

VARIABLE	DESCRIPTION
PID	Part ID or part set ID.
TYP	Type of "PID" (see remark 1): EQ.0: part set ID (PSID). EQ.1: part ID (PID).
AMMSID	Set ID of ALE multi-material groups defined in *SET_MULTI-MATERIAL_GROUP. See remark 1.
XO	Origin position in global X-direction. See remark 2.
YO	Origin position in global Y-direction. See remark 2.
ZO	Origin position in global Z-direction. See remark 2.

VECID	ID of the symmetric axis defined by *DEFINE_VECTOR. See remark 3.
ANGLE	Angle of rotation around an axis defined by *DEFINE_VECTOR for the 3D to 3D mapping. See remark 6.

Remarks:

1. The routines of this card need to know which mesh will be initialized with the mapping data and more specifically which multi-material groups. The first 2 parameters (PID and TYP) defines the mesh and the third one (AMMSID) calls the *SET_MULTI-MATERIAL_GROUP_LIST card. This card will define a list of material groups in the current run. The rank in this list should match the rank of the multi-material groups from the previous run (as a reminder the ranks of multi-material groups are defined by *ALE_MULTI-MATERIAL_GROUP). Let's take an example. For instance if the previous model has 3 groups, the current one has 5 groups and the following mapping is wanted:
   - The 1st group from the previous run is mapped on the 3rd one in the current run,
   - The 2nd group from the previous run is mapped on the 5th one in the current run,
   - The 3rd group from the previous run is mapped on the 4th one in the current run,

   The *SET_MULTI-MATERIAL_GROUP_LIST card should be set as follows:

   ```
   *SET_MULTI-MATERIAL_GROUP_LIST
   300
   3,5,4
   ```

2. The data can be mapped in different parts of the mesh by defining the origin of the coordinate system (XO,YO,ZO).

3. For a mapping between a 2D axisymmetric model and a 3D run the symmetric axis orientation in the 3D model needs to be defined. For a mapping from 2D to 2D the vector is not used but *DEFINE_VECTOR is still read. For a 3D to 3D mapping the vector is used if the parameter ANGLE is defined (see Remark 6).

4. To create the mapping file: only the prompt "map=" is necessary. If the keyword INITIAL_ALE_MAPPING is not in the input deck and the prompt "map=" is in the command line, the ALE data of the last cycle are written in the mapping file.

5. To make several successive mapping: the prompt “map1=” is necessary. If the keyword INITIAL_ALE_MAPPING is in the input deck and the prompt “map=” is in the command line, the ALE data are read from the mapping file defined by “map=” to initialize the run. Data of the last cycle are written in the mapping file defined by “map1=”.

6. For a mapping from a previous 3D run to a current 3D model, the parameter ANGLE allows to rotate the previous 3D domain around an axis identified by VECID and defined by *DEFINE_VECTOR.

*INITIAL_AXIAL_FORCE_BEAM

Purpose: Initialize the axial force resultants in beam elements that are used to model bolts. This option works with *MAT_SPOTWELD with beam type 9, a Hughes-Liu type beam. The beam elements listed in this section will be active in contact unlike the spot weld beam elements, which are automatically excluded in the automatic contact treatments.

Card 1	1	2	3	4	5	6	7	8
Variable	BSID	LCID	SCALE					
Type	I	I	F					
Default	none	none	1.0					

VARIABLE	DESCRIPTION
BSID	Beam set ID.
LCID	Load curve ID defining preload force versus time. When the load curve ends or goes to zero, the initialization is assumed to be completed. See remark 2 below.
SCALE	Scale factor on load curve.

Remarks:

1. To achieve convergence during explicit dynamic relaxation, the application of the damping options is very important. If contact is active, contact damping is recommended with a value between 10-20 percent. Additional damping, via the option DAMPING_ PART_STIFFNESS also speeds convergence where a coefficient of 0.10 is effective. If damping is not used, convergence may not be possible.

2. When defining the load curve, LCID, a ramp starting at the origin should be used to increase the force to the desired value. The time duration of the ramp should produce a quasi-static response. When the end of the load curve is reached, or when the value of the load decreases from its maximum value, the initialization stops. If the load curve begins at the desired force value, i.e., no ramp, convergence will take much longer, since the impulsive like load created by the initial force can excite nearly every frequency in the structural system where force is initialized.

*INITIAL_DETONATION

Purpose: Define points to initiate the location of high explosive detonations in part ID's which use the material (type 8) *MAT_HIGH_EXPLOSIVE_BURN. Also see *CONTROL_ EXPLOSIVE_SHADOW.

Card 1	1	2	3	4	5	6	7	8
Variable	PID	X	Y	Z	LT			
Type	I	F	F	F	F			
Default	all HE	0.	0.	0.	0,			

Optional card required if and only if PID=-1.

Card 2	1	2	3	4	5	6	7	8
Variable	PEAK	DECAY	XS	YS	ZS	NID		
Type	F	F	F	F	F	I		
Remark	1	1						

VARIABLE	DESCRIPTION
PID	Part ID of high explosive material to be lit, see *PART. However, two other options are available: EQ.-1: an acoustic boundary, also, *BOUNDARY_USA_SURFACE, EQ. 0: all high explosive materials are considered.
X	x-coordinate of detonation point, see Figure 18.1.
Y	y-coordinate of detonation point.
Z	z-coordinate of detonation point.

VARIABLE	DESCRIPTION
LT	Lighting time for detonation point. This time is ignored for an acoustic boundary.
PEAK	Peak pressure, p_o, of incident pressure pulse, see remark below.
DECAY	Decay constant, τ
XS	x-coordinate of standoff point, see Figure 18.1.
YS	y-coordinate of standoff point
ZS	z-coordinate of standoff point
NID	Reference node ID near structure

Remarks:

For solid elements (not acoustic) two options are available. If the control card option, *CONTROL_EXPLOSIVE_SHADOW, is not used the lighting time for an explosive element is computed using the distance from the center of the element to the nearest detonation point, L_d; the detonation velocity, *D*; and the lighting time for the detonator, t_d:

$$t_L = t_d + \frac{L_d}{D}$$

The detonation velocity for this default option is taken from the element whose lighting time is computed and does not account for the possibilities that the detonation wave may travel through other explosives with different detonation velocities or that the line of sight may pass outside of the explosive material.

If the control card option, *CONTROL_EXPLOSIVE_SHADOW, is defined, the lighting time is based on the shortest distance through the explosive material. If inert obstacles exist within the explosive material, the lighting time will account for the extra time required for the detonation wave to travel around the obstacles. The lighting times also automatically accounts for variations in the detonation velocity if different explosives are used. No additional input is required for this option but care must be taken when setting up the input. This option works for two and three-dimensional solid elements. It is recommended that for best results:

1. Keep the explosive mesh as uniform as possible with elements of roughly the same dimensions.

2. Inert obstacle such as wave shapers within the explosive must be somewhat larger than the characteristic element dimension for the automatic tracking to function properly. Generally, a factor of two should suffice. The characteristic element dimension is found by checking all explosive elements for the largest diagonal.

3. The detonation points should be either within or on the boundary of the explosive. Offset points may fail to initiate the explosive.

4. Check the computed lighting times in the post processor LS-PREPOST. The lighting times may be displayed at time=0., state 1, by plotting component 7 (a component normally reserved for plastic strain) for the explosive material. The lighting times are stored as negative numbers. The negative lighting time is replaced by the burn fraction when the element ignites.

Line detonations may be approximated by using a sufficient number of detonation points to define the line. Two many detonation points may result in significant initialization cost.

The pressure versus time curve for the acoustic option is defined by:

$$p(t) = p_o e^{-\frac{t}{\tau}}.$$

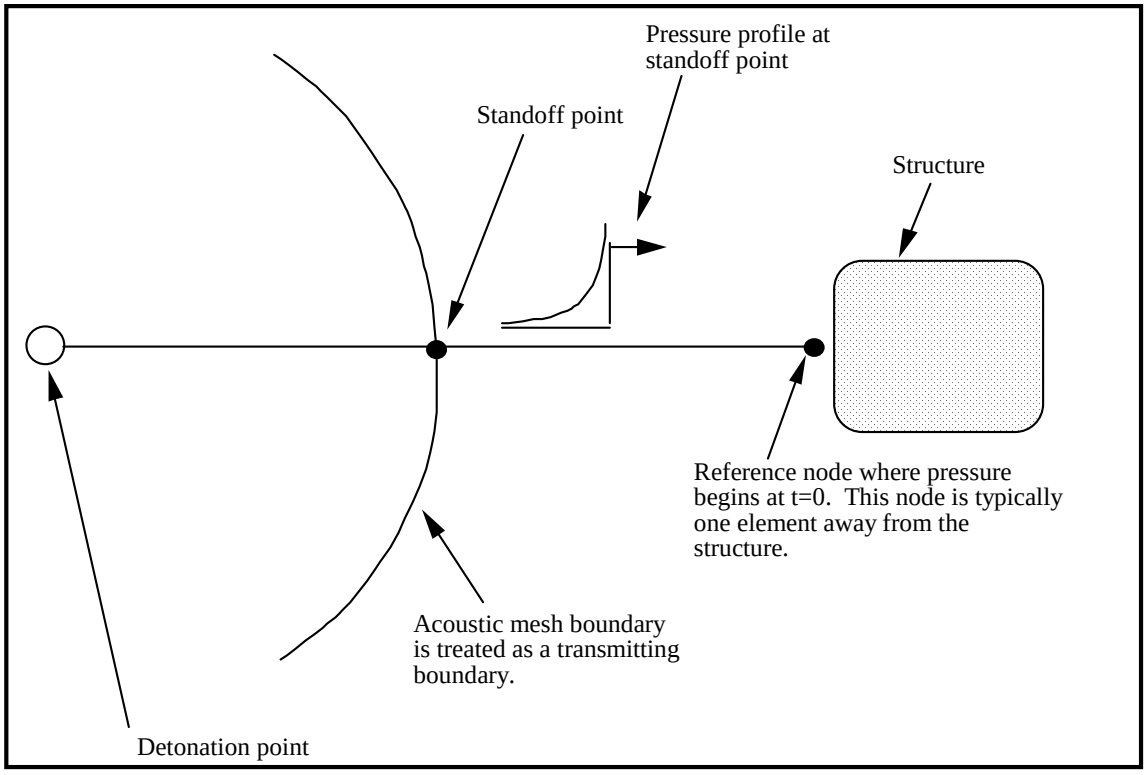


Figure 18.1 Initialization of the initial pressures due to an explosive disturbance is performed in the acoustic media. LS-DYNA automatically determines the acoustic mesh boundary and applies the pressure time history to the boundary. This option is only applicable to the acoustic element formulation, see *SECTION_SOLID.

*INITIAL_FOAM_REFERENCE_GEOMETRY

Purpose: The reference configuration allows stresses to be initialized in the following hyperelastic material models: 2, 5, 7, 21, 23, 27, 31, 38, 57, 73, 77, 83, 132, 179, 181, 183, and 189. Supported solid elements are the constant stress hexahedron (#1), the fully integrated S/R hexahedron (#2), the tetrahedron (#10), and the pentahedron (#15).

To use this option, the geometry of the foam material is defined in a deformed configuration. The stresses in the low density foam then depend only on the deformation gradient matrix F_{ij}:

$$F_{ij} = \frac{\partial x_i}{\partial X_j}$$

where x_i is the deformed configuration and X_j is the undeformed configuration. By using this option, dynamic relaxation can be avoided once a deformed configuration is obtained usually on the first run of a particular problem.

Card Format (I8,3E16.0)

Card 1,...	1	2	3	4	5	6	7	8	9	10
Variable	NID	X		Y		Z				
Type	I	F		F		F				
Default	none	0.		0.		0.				
Remarks										

VARIABLE	DESCRIPTION
NID	Node number
X	x coordinate in reference configuration
Y	y coordinate in reference configuration
Z	z coordinate in reference configuration

***INITIAL_GAS_MIXTURE**

Purpose: This command is used to specify (a) which ALE multi-material groups may be present inside an ALE mesh set at time zero, and (b) the corresponding reference gas temperature and density which define the initial thermodynamic state of the gases. The order of the species in the gas mixture corresponds to the order of different gas species defined in the associated *MAT_GAS_MIXTURE card. This card must be used together with a *MAT_GAS_MIXTURE (or equivalently, a *MAT_ALE_GAS_MIXTURE) card.

Card 1

Variable	SID	STYPE	MMGID	TEMP				
Type	I	I	I	F				
Default	none	0	none	none				

Card 2

Variable	RO1	RO2	RO3	RO4	RO5	RO6	RO7	RO8
Type	F	F	F	F	F	F	F	F
Default	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0

VARIABLE	DESCRIPTION
SID	Set ID for initialization. This SID defines the ALE mesh within which certain ALE multi-material group(s) may be present at t=0.
STYPE	Set type for the SID above: EQ.0: SID is a part set ID EQ.1: SID is a part ID
MMGID	ALE Multi-material group ID of the material that may be present at t=0 in the ALE mesh set defined by SID.
TEMP	Initial static temperature of the gas species occupying the ALE mesh. Note that all species in the mixture are assumed to be in thermal equilibrium (having the same T).

VARIABLE	DESCRIPTION
RO1-RO8	Initial densities of the ALE material(s) which may be occupying some region (or all) of the aforementioned ALE mesh, for up to eight different gas species. The order of the density input corresponds to the order of the materials defined in associated *MAT_GAS_MIXTURE card.

Remarks:

1. Please see the example under the *MAT_GAS_MIXTURE card definition for an application of the *INITIAL_GAS_MIXTURE card.

2. The temperature is assumed to be the initial temperature which together with the gas density, will define the initial pressure of the gas species via the perfect gas law $\left(P\big|_{t=0} = \rho\big|_{t=0}\left(C_P - C_V\right)T\big|_{t=0}\right)$. The user should manually check the initial pressure for consistency.

3. Given an ALE mesh, this mesh may initially be occupied by one or more ALE multi-material groups (AMMG). For example, a background ALE mesh (H1) containing AMMG 1 may be partially filled with AMMG 2 via the volume filling command *INITIAL_VOLUME_FRACTION_GEOMETRY. Then there are 2 AMMGs to be initialized for this mesh H1. The commands look like the following.

```
$---------------------------------------------------------------------------
$ One card is defined for each AMMG that will occupy some elements of a mesh set
*INITIAL_GAS_MIXTURE
$      SID     STYPE     MMGID        T0
         1         1         1    298.15
$     RHO1      RHO2      RHO3      RHO4      RHO5      RHO6      RHO7      RHO8
    1.0E-9
*INITIAL_GAS_MIXTURE
$      SID     STYPE     MMGID        T0
         1         1         2    298.15
$     RHO1      RHO2      RHO3      RHO4      RHO5      RHO6      RHO7      RHO8
    1.2E-9

$---------------------------------------------------------------------------
```

***INITIAL_INTERNAL_DOF_SOLID_{OPTION}**

Valid OPTIONS are TYPE3 and TYPE4.

Purpose: Initialize the internal degrees of freedom for solid element types 3 and 4.

Card 1	1	2	3	4	5	6	7	8
Variable	LID							
Type	I							
Default	none							
Card 2 to N	1	2	3	4	5	6	7	8
Variable	VALX	VALY	VALZ					
Type	F	F	F					
Default	none	none	none					

VARIABLE	DESCRIPTION
LID	Element ID.
LCID	Load curve ID defining preload force versus time. When the load curve ends or goes to zero, the initialization is assumed to be completed. See remark 2 below.
VALX	X component of internal degree of freedom.
VALY	Y component of internal degree of freedom.
VALZ	Z component of internal degree of freedom.

REMARKS:

1. Type 3 solids require 12 cards (N=13) to initialize the internal degrees of freedom, and the type 4 solids require 6 cards (N=7).

2. The internal degrees of freedom are specified in terms of the displacements of the corresponding mid-side nodes of the 20 node hex and the 10 node tet that are the basis of the type 3 and 4 solid elements, respectively.

*INITIAL_MOMENTUM

Purpose: Define initial momentum to be deposited in solid elements. This option is to crudely simulate an impulsive type of loading.

Card	1	2	3	4	5	6	7	8
Variable	EID	MX	MY	MZ	DEPT			
Type	I	F	F	F	F			
Default	none	0.	0.	0.	0,			

VARIABLE	DESCRIPTION
EID	Element ID
MX	Initial x-momentum
MY	Initial y-momentum
MZ	Initial z-momentum
DEPT	Deposition time

*INITIAL_PWP_DEPTH

Purpose: Initialize pore water pressure in solid elements where a non-hydrostatic profile is required.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	PID	LC						
Type	I	I						
Default	none	none						

VARIABLE	DESCRIPTION
PID	Part ID
LC	Curve of pore water pressure head (length units) vs Z-coordinate

Notes: This feature overrides the automatically calculated hydrostatic pressure profile. The points in the curve must be ordered with the most negative z-coordinate first – this order looks "upside-down" on the page.

If a part has pore fluid but no *INITIAL_PWP_DEPTH is defined, the default initial pressure profile is hydrostatic.

***INITIAL_STRAIN_SHELL_{*OPTION*}**

The availables options include:

<BLANK>

SET

Purpose: Initialize strain tensor and inner and outer through thickness integration points at element center. This option is primarily for multi-stage metal forming operations where the accumulated strain is of interest.

Define as many shell elements in this section as desired. The input is assumed to terminate when a new keyword is detected. These strain tensors are defined at the inner and outer integration points and are used for post-processing only. There is no interpolation with this option and the strains are defined in the global cartesian coordinate system. The *DATABASE_EXTENT_BINARY flag STRFLG must be set to unity for this option to work.

Card 1	1	2	3	4	5	6	7	8
Variable	EID							
Type	I							
Default	none							

Define two cards below, one for the inner integration point and the other for the outer integration point, respectively.

Card 2,...	1	2	3	4	5	6	7	8
Variable	EPSxx	EPSyy	EPSzz	EPSxy	EPSyz	EPSzx		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
EID	Element ID or shell element set ID when the **SET** option is used.
EPSij	Define the ij strain component. The strains are defined in the GLOBAL cartesian system.

***INITIAL_STRAIN_SOLID_{***OPTION***}**

The availables options include:

<BLANK>

SET

Purpose: Initialize strain tensor at element center. This option can be used for multi-stage metal forming operations where the accumulated strain is of interest. This option is available starting in Release 3 of version 971.

Define as many solid elements in this section as desired. The input is assumed to terminate when a new keyword is detected. These strain tensors are defined at the element center and are used for post-processing only. The strains are defined in the global cartesian coordinate system. The *DATABASE_EXTENT_BINARY flag STRFLG must be set to unity for this option to work. This capability is not available for the cohesive element since it is based on displacements, not strains.

Card 1	1	2	3	4	5	6	7	8
Variable	EID							
Type	I							
Default	none							

Define one card below.

Card 2,...	1	2	3	4	5	6	7	8
Variable	EPSxx	EPSyy	EPSzz	EPSxy	EPSyz	EPSzx		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
EID	Element ID or solid element set ID when the **SET** option is used.
EPSij	Define the ij strain component. The strains are defined in the GLOBAL cartesian system.

***INITIAL_STRESS_BEAM**

Purpose: Initialize stresses and plastic strains in the Hughes-Liu beam elements.

Define as many beams in this section as desired. The input is assumed to terminate when a new keyword is detected.

Card 1	1	2	3	4	5	6	7	8
Variable	EID	RULE	NPTS	LOCAL				
Type	I	I	I	I				
Default	none	none	none	0				

Define NTPS cards below, one per integration point.

Card 2,...	1	2	3	4	5	6	7	8
Variable	SIG11	SIG22	SIG33	SIG12	SIG23	SIG31	EPS	
Type	F	F	F	F	F	F	F	

VARIABLE	DESCRIPTION
EID	Element ID
RULE	Integration rule type number: EQ.1.0: 1×1 Gauss quadrature EQ.2.0: 2×2 Gauss quadrature (default beam), EQ.3.0: 3×3 Gauss quadrature, EQ.4.0: 3×3 Lobatto quadrature, EQ.5.0: 4×4 Gauss quadrature.
NPTS	Number of integration points output.
LOCAL	Coordinate system for stresses: EQ.0: stress components are defined in the global coordinate system. EQ.1: stress components are defined in the local beam system. In the local system components SIG22, SIG33, and SIG23 are set to 0.0.

VARIABLE	DESCRIPTION
SIGIJ	Define the IJ stress component.
EPS	Effective plastic strain

***INITIAL_STRESS_DEPTH**

Purpose: Initialize solid element stresses where stress is a function of depth.

Note: This keyword card will be available starting in release 3 of version 971.

Card 1	1	2	3	4	5	6	7	8
Variable	PID	RO_G	ZDATUM	KFACT	LC	LCH		
Type	I	F	F	F	I	I		
Default	none	none	none	0.0	none	none		

VARIABLE	DESCRIPTION
PID	Part ID
RO_G	Stress per unit elevation above datum (usually = density x gravity)
ZDATUM	Z-coordinate of datum
KFACT	X- and Y-stress = KFACT x Z-stress
LC	Optional curve of stress vs z-coordinate (ZDATUM is ignored with this option)
LCH	Optional curve of horizontal stress versus z-coordinate (KFACT is ignored with this option)

Remarks:

Z-stress = $\text{RO\_G} \times (\text{Z}_{\text{element}} - \text{ZDATUM})$. To generate compressive stresses, the datum should be above the highest element – usually at the surface of the soil in geotechnics simulations. If the curve is present, it overrides RO_G and ZDATUM. Note that the points in the curve should be ordered with most negative z-coordinate first – this order looks "upside-down" on the page.

*INITIAL_STRESS_SECTION

Purpose: Initialize the stress in solid elements that are part of a section definition to create a preload. The stress component in the direction normal to the cross-section plane is initialized. This option works with a subset of materials that are incrementally updated including the elastic, viscoelastic, and elastoplastic materials. Rubbers, foams, and materials that are combined with equations-of-state cannot be initialized by this approach. NEW: Hyperelastic materials # 57, 73 and 83 can be initialized with this approach.

Card 1	1	2	3	4	5	6	7	8
Variable	ISSID	CSID	LCID	PSID	VID			
Type	I	A8	I	I	I			
Default	none	none	none	none	none			

VARIABLE	DESCRIPTION
ISSID	Section stress initialization ID.
CSID	Cross-section ID. See *DATABASE_CROSS_SECTION.
LCID	Load curve ID defining preload stress versus time. When the load curve ends or goes to zero, the initialization is assumed to be completed. See remark 2 below.
PSID	Part set ID.
VID	Vector ID defining the direction normal to the cross section. This vector must be defined if *DATABASE_CROSS_SECTION_SET is used to define the cross section. If the cross section is defined using the PLANE option, the normal used in the definition of the plane is used if VID is left undefined.

Remarks:

1. To achieve convergence during explicit dynamic relaxation, the application of the damping options is very important. If contact is active, contact damping is recommended with a value between 10-20 percent. Additional damping, via the option DAMPING_ PART_STIFFNESS also speeds convergence where a coefficient of 0.10 is effective. If damping is not used, convergence may not be possible.

2. When defining the load curve, LCID, a ramp starting at the origin should be used to increase the stress to the desired value. The time duration of the ramp should produce a quasi-static response. When the end of the load curve is reached, or when the value of the load decreases from its maximum value, the initialization stops. If the load curve begins at the desired stress value, i.e., no ramp, convergence will take much longer, since the impulsive like load created by the initial stress can excite nearly every frequency in the structural system where stress is initialized.

3. This option currently applies only to materials that are incrementally updated. Hyperelastic materials and materials that require an equation-of-state are not currently supported.

4. Solid elements types 1, 2, 3, 4, 9, 10, 13, 15, 16, 17, and 18 are supported. ALE elements are not supported.

***INITIAL_STRESS_SHELL_**{*OPTION*}

Available options include:

<BLANK>

SET

Purpose: Initialize stresses, history variables, and the effective plastic strain for shell elements.

Define as many shell elements or shell element sets in this section as desired. The input is assumed to terminate when a new keyword is detected. It is not necessary for the location of the through thickness integration points to match those used in the elements which are initialized. The data will be interpolated by LS-DYNA.

Card 1 1 2 3 4 5 6 7 8

Variable	EID/SID	NPLANE	NTHICK	NHISV	NTENSR	LARGE	NTHINT	NTHHSV
Type	I	I	I	I	I	I	I	I
Default	none	none	none	0	0	0	0	0

Define NPLANE X NTHICK cards below, one (if NHISV=0 and LARGE=0) or two (if NHISV=0 and LARGE=1) per integration point. Include optional cards as necessary to define the NHISV history variable. For each through thickness point define NPLANE points. NPLANE should be either 1 or 4 corresponding to either 1 or 4 Gauss integration points. If four integration points are specified, they should be ordered such that their in plane parametric coordinates are at:

$$\left(-\frac{\sqrt{3}}{3},-\frac{\sqrt{3}}{3}\right),\left(\frac{\sqrt{3}}{3},-\frac{\sqrt{3}}{3}\right),\left(\frac{\sqrt{3}}{3},\frac{\sqrt{3}}{3}\right),\left(-\frac{\sqrt{3}}{3},\frac{\sqrt{3}}{3}\right),$$

respectively.

If LARGE=0, then define the following cards (8E10.0).

Card 2… 1 2 3 4 5 6 7 8

Variable	T	SIGXX	SIGYY	SIGZZ	SIGXY	SIGYZ	SIGZX	EPS
Type	F	F	F	F	F	F	F	F

Optional

	1	2	3	4	5	6	7	8
Variable	HISV1	HISV2	HISV3	HISV4	HISV5	HISV6	HISV7	HISV8
Type	F	F	F	F	F	F	F	F

Optional

Variable	TENXX	TENYY	TENZZ	TENXY	TENYZ	TENZX		
Type	F	F	F	F	F	F		

If LARGE=1, then define the following cards (5E16.0).

Card 2

	1	2	3	4	5
Variable	T	SIGXX	SIGYY	SIGZZ	SIGXY
Type	F	F	F	F	F

Card 3…

Variable	SIGYZ	SIGZX	EPS		
Type	F	F	F		

Optional

Variable	HISV1	HISV2	HISV3	HISV4	HISV5
Type	F	F	F	F	F

Optional

	1	2	3	4	5
Variable	TENXX	TENYY	TENZZ	TENXY	TENYZ
Type	F	F	F	F	F

Optional

Variable	TENZX				
Type	F				

Define NTHINT sets of cards with NTHHSV number of parameters for each set on the following format (LARGE=1)

Cards for integration point 1

Card 1

	1	2	3	4	5
Variable	THHSV_1_1	THHSV_1_2	THHSV_1_3	THHSV_1_4	THHSV_1_5
Type	F	F	F	F	F

Card 2…

Variable	THHSV_1_6	THHSV_1_7	THHSV_1_8	…	
Type	F	F	F		

Cards for integration point 2

Card 2	1	2	3	4	5
Variable	THHSV_2_1	THHSV_2_2	THHSV_2_3	THHSV_2_4	THHSV_2_5
Type	F	F	F	F	F

Card 2…

Variable	THHSV_2_6	THHSV_2_7	THHSV_2_8	…	
Type	F	F	F		

VARIABLE	DESCRIPTION
EID/SID	Element ID or shell set ID, see *SET_SHELL_....
NPLANE	Number of in plane integration points being output.
NTHICK	Number of integration points through the thickness.
NHISV	Number of additional history variables.
NTENSR	Number of components of tensor data taken from the element history variables stored.
LARGE	Format size (0:off or 1:on). See cards above.
T	Parametric coordinate of through thickness integration point between -1 and 1 inclusive.
SIGij	Define the *ij* stress component. The stresses are defined in the GLOBAL cartesian system.
EPS	Effective plastic strain
HISVn	Define the nth history variable.
TENij	Define the *ij*th component of the tensor taken from the history variables. The tensor is defined in the GLOBAL Cartesian system. Define enough lines to provide a total of NTENSOR components, stored six components per line. This applies to material 190 only.

VARIABLE	DESCRIPTION
NTHINT	Number of thermal integration points
NTHHSV	Number of thermal history variables per thermal integration point.
THHSV_M_N	N:th history variable at the M:th thermal integration point

***INITIAL_STRESS_SOLID**

Purpose: Initialize stresses and plastic strains for solid elements. This command is not applicable to hyperelastic materials or any material model based on a Total Lagrangian formulation. Furthermore, for *mat_005, *mat_014, and any material that requires an equation-of-state (*EOS), the initialized stresses are deviatoric stresses, not total stresses.

Define as many solid elements in this section as desired. The input is assumed to terminate when a new keyword is detected. If eight points are defined for 1 point LS-DYNA solid elements, the average value will be taken.

Card 1

Variable	EID	NINT	NHISV	LARGE	IVEFLG		NTHINT	NTHHSV
Type	I	I	I	I	I		I	I
Default	none	none	0	0	0		0	0

Define NINT cards below, one per integration point. NINT should be either 1 or 8. If eight Gauss integration points are specified, they should be ordered such that their parametric coordinates are located at:

$$\left(-\frac{\sqrt{3}}{3},-\frac{\sqrt{3}}{3},-\frac{\sqrt{3}}{3}\right),\ \left(\frac{\sqrt{3}}{3},-\frac{\sqrt{3}}{3},-\frac{\sqrt{3}}{3}\right),\ \left(\frac{\sqrt{3}}{3},\frac{\sqrt{3}}{3},-\frac{\sqrt{3}}{3}\right),\ \left(-\frac{\sqrt{3}}{3},\frac{\sqrt{3}}{3},-\frac{\sqrt{3}}{3}\right),$$

$$\left(-\frac{\sqrt{3}}{3},-\frac{\sqrt{3}}{3},\frac{\sqrt{3}}{3}\right),\ \left(\frac{\sqrt{3}}{3},-\frac{\sqrt{3}}{3},\frac{\sqrt{3}}{3}\right),\ \left(\frac{\sqrt{3}}{3},\frac{\sqrt{3}}{3},\frac{\sqrt{3}}{3}\right),\ \left(-\frac{\sqrt{3}}{3},\frac{\sqrt{3}}{3},\frac{\sqrt{3}}{3}\right),$$

respectively.

If LARGE=0, then define the following cards, no history variables are allowed.

Card 2...

Variable	SIGXX	SIGYY	SIGZZ	SIGXY	SIGYZ	SIGZX	EPS	
Type	F	F	F	F	F	F	F	

If LARGE=1, then define the following card sets for each integration point. If NHISV>3 define as many additional cards as necessary. (5E16.0). If IVEFLG equals 1 or 2 the last history variables are the initial element volume, IVEFLG=1, or the initial element volume plus the internal energy density per unit initial volume, IVEFLG=2. The specification of energy provides way of initializing pressure in elements which use constitutive models which require energy dependent equations-of-state.

Card 2

1	2	3	4	5
SIGXX	SIGYY	SIGZZ	SIGXY	SIGYZ
F	F	F	F	F

Card 3…

1	2	3	4	5
SIGZX	EPS	HISV1	HISV2	HISV3
F	F	F	F	F

Card 4…

1	2	3	4	5
HISV…	HISV…	HISVn-2	INITVOL	ENERGY
F	F	F	F	F

Define NTHINT sets of cards with NTHHSV number of parameters for each set on the following format (LARGE=1) . Cards for integration point 1:

Card 1

1	2	3	4	5
THHSV_1_1	THHSV_1_2	THHSV_1_3	THHSV_1_4	THHSV_1_5
F	F	F	F	F

Card 2…

1	2	3	4	5
THHSV_1_6	THHSV_1_7	THHSV_1_8	…	
F	F	F		

Cards for integration point 2

Card 1	1	2	3	4	5
Variable	THHSV_2_1	THHSV_2_2	THHSV_2_3	THHSV_2_4	THHSV_2_5
Type	F	F	F	F	F

Card 2…

Variable	THHSV_2_6	THHSV_2_7	THHSV_2_8	…	
Type	F	F	F		

VARIABLE	DESCRIPTION
EID	Element ID
NINT	Number of integration points either 1 or 8.
NHISV	Number of additional history variables, which is typically equal to the number of history variables stored at the integration point + IVEFLG. If NHISV exceeds the number of integration point history variables required by the constitutive model, only the number required is output; therefore, if in doubt, set NHISV to a large number.
LARGE	Format size, if zero, NHISV must also be set to zero (this is the format used by LS-DYNA versions 970 and earlier) and, if set to 1, a larger format is used and NHISV is used.
IVEFLG	Initial Volume/energy flag (**only used in large format**) EQ.0:last history variable is used as normal, EQ.1:last history variable is used as the initial volume of the element. One additional history variable is required if IVFLG=1 EQ.2:last two history variables are used to define the initial volume and the internal energy per unit initial volume. Two additional history variables are must be allocated, see NHISV above, if IVFLG=2. If the initial volume is set to zero, the actual element volume is used.
SIGij	Define the *ij*th stress component. Stresses are defined in the GLOBAL Cartesian system.

VARIABLE	DESCRIPTION
EPS	Effective plastic strain.
HISV1…n-1	Define n-1 history variables.
INITVOL	Initial volume (HISVn)
NTHINT	Number of thermal integration points
NTHHSV	Number of thermal history variables per thermal integration point.
THHSV_M_N	N:th history variable at the M:th thermal integration point

REMARKS:

1. The elastic material model for cohesive elements is a total Lagrangian formulation, and the initial stress will therefore be ignored for it.

***INITIAL_STRESS_TSHELL**

Purpose: Initialize stresses and plastic strains for thick shell elements.

Define as many thick shell elements in this section as desired. The input is assumed to terminate when a new keyword is detected. It is not necessary for the location of the through thickness integration points to match those used in the elements which are initialized. The data will be interpolated by LS-DYNA.

Card 1	1	2	3	4	5	6	7	8
Variable	EID	NPLANE	NTHICK	NHISV	LARGE			
Type	I	I	I	I	I			
Default	none	none	none	0	0			

Define NPLANE*NTHICK cards below, one per integration point for LARGE=0, or 2 cards for each integration point for LARGE=1. Include optional cards as necessary to define the NHISV history variables. For each through thickness point define NPLANE points. NPLANE should be either 1 or 4 corresponding to either 1 or 4 Gauss integration points. If four integration points are specified, they should be ordered such that their in plane parametric coordinates are at:

$$\left(-\frac{\sqrt{3}}{3},-\frac{\sqrt{3}}{3}\right),\left(\frac{\sqrt{3}}{3},-\frac{\sqrt{3}}{3}\right),\left(\frac{\sqrt{3}}{3},\frac{\sqrt{3}}{3}\right),\left(-\frac{\sqrt{3}}{3},\frac{\sqrt{3}}{3}\right),$$

respectively.

If LARGE=0, then define the following cards for each integration point. Provide only as many optional cards as needed if NHISV>0. (8E10.0).

Card 2	1	2	3	4	5	6	7	8
Variable	T	SIGXX	SIGYY	SIGZZ	SIGXY	SIGYZ	SIGZX	EPS
Type	F	F	F	F	F	F	F	F

Optional 1 2 3 4 5 6 7 8

Variable	HISV1	HSIV2	HSIV3	HSIV4	HSIV5	HSIV6	HSIV7	HSIV8
Type	F	F	F	F	F	F	F	F

If LARGE=1, then define the following card sets for each integration point. Provide only as many optional cards as needed if NHISV>0. (5E16.0)

Card 2 1 2 3 4 5

Variable	T	SIGXX	SIGYY	SIGZZ	SIGXY
Type	F	F	F	F	F

Card 3

Variable	SIGYZ	SIGZX	EPS		
Type	F	F	F		

Optional

Variable	HISV1	HISV2	HISV3	HISV4	HISV5
Type	F	F	F	F	F

VARIABLE	DESCRIPTION
EID	Element ID
NPLANE	Number of in plane integration points.
NTHICK	Number of integration points through the thickness.
T	Parametric coordinate of through thickness integration point. between -1 and 1 inclusive.
NHISV	Number of additional history variables.

VARIABLE	DESCRIPTION
LARGE	Format size (0:off or 1:on). See cards above.
SIGij	Define the ij stress component. The stresses are defined in the GLOBAL cartesian system.
EPS	Effective plastic strain

INITIAL_TEMPERATURE_OPTION*

Available options include:

NODE

SET

Purpose: Define initial nodal point temperatures using nodal set ID's or node numbers. These initial temperatures are used in a thermal only analysis or a coupled thermal/structural analysis. See also *CONTROL_THERMAL_SOLVER, *CONTROL_THERMAL_TIMESTEP, and CONTROL_ THERMAL_NONLINEAR.

For thermal loading in a structural only analysis, see *LOAD_THERMAL_*OPTION*.

Card 1	1	2	3	4	5	6	7	8
Variable	NSID/NID	TEMP	LOC					
Type	I	I	I					
Default	none	0.	0					
Remark	1							

VARIABLE	DESCRIPTION
NSID/NID	Nodal set ID or nodal point ID, see also *SET_NODES: EQ.0: all nodes are included (set option only).
TEMP	Temperature at node or node set.
LOC	Application of surface for thermal shell elements, see parameter, TSHELL, in the *CONTROL_SHELL input: EQ.-1: lower surface of thermal shell element EQ. 0: middle surface of thermal shell element EQ. 1: upper surface of thermal shell element

Remarks:

1. If a nodal temperature is specified on more than one input card, then the last set input will determine its temperature unless it is specified on a *INITIAL_TEMPERATURE_NODE card.

*INITIAL_VEHICLE_KINEMATICS

Purpose: Define initial kinematical information for a vehicle. In its initial orientation, the vehicle's yaw, pitch, and roll axes must be aligned with the global axes. Successive simple rotations are taken about these body fixed axes.

Card 1

Variable	GRAV	PSID	XO	YO	ZO	XF	YF	ZF
Type	I	I	F	F	F	F	F	F
Default	none	none	0.	0.	0.	0.	0.	0.

Card 2

Variable	VX	VY	VZ	AAXIS	BAXIS	CAXIS		
Type	F	F	F	I	I	I		
Default	0.	0.	0.	0	0	0		

Card 3

Variable	AANG	BANG	CANG	WA	WB	WC		
Type	F	F	F	F	F	F		
Default	0.	0.	0.	0.	0.	0.		

VARIABLE	DESCRIPTION
GRAV	Gravity direction code. EQ. 1: Global +x direction. EQ.-1: Global -x direction. EQ. 2: Global +y direction. EQ.-2: Global -y direction. EQ. 3: Global +z direction. EQ.-3: Global -z direction. Note: this must be the same for all vehicles present in the model.
PSID	Part set ID.
XO	x-coordinate of initial position of mass center.
YO	y-coordinate of initial position of mass center.
ZO	z-coordinate of initial position of mass center.
XF	x-coordinate of final position of mass center.
YF	y-coordinate of final position of mass center.
ZF	z-coordinate of final position of mass center.
VX	x-component of mass center velocity.
VY	y-component of mass center velocity.
VZ	z-component of mass center velocity.
AAXIS	First rotation axis code. EQ.1: Initially aligned with global x-axis. EQ.2: Initially aligned with global y-axis. EQ.3: Initially aligned with global z-axis.
BAXIS	Second rotation axis code.
CAXIS	Third rotation axis code.
AANG	Rotation angle about the first rotation axis (degrees).
BANG	Rotation angle about the second rotation axis (degrees).
CANG	Rotation angle about the third rotation axis (degrees).
WA	Angular velocity component for the first axis (radian/second).

VARIABLE	DESCRIPTION
WB	Angular velocity component for the second axis (radian/second).
WC	Angular velocity component for the third axis (radian/second).

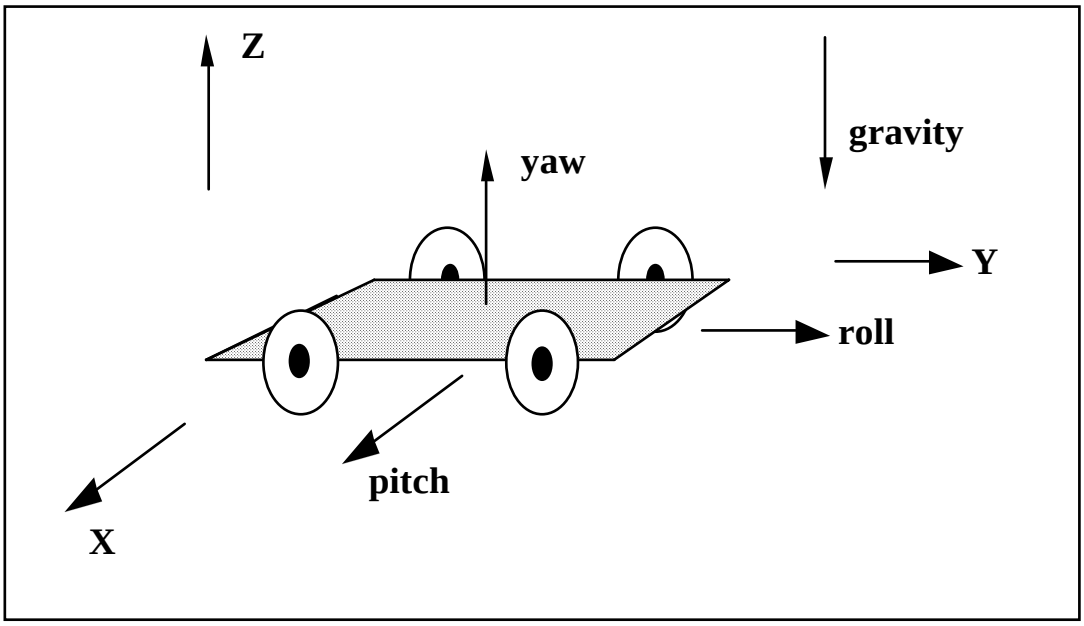


Figure 18.2. The vehicle pictured is to be oriented with a successive rotation sequence about the yaw, pitch, and roll axes, respectively. Accordingly, AAXIS=3, BAXIS=1, and CAXIS=2. The direction of gravity is given by GRAV=-3.

*INITIAL_VELOCITY

Purpose: Define initial nodal point translational velocities using nodal set ID's. This may also be used for sets in which some nodes have other velocities. See NSIDEX below.

Card 1

Variable	NSID	NSIDEX	BOXID	IRIGID	ICID			
Type	I	I	I	I	I			
Default	none	0	0	0	0			
Remark	1							

Card 2

Variable	VX	VY	VZ	VXR	VYR	VZR		
Type	F	F	F	F	F	F		
Default	0.	0.	0.	0.	0.	0.		

Define the following card if and only if NSIDEX>0.

Card 3

Variable	VXE	VYE	VZE	VXRE	VYRE	VZRE		
Type	F	F	F	F	F	F		
Default	0.	0.	0.	0.	0.	0.		

VARIABLE	DESCRIPTION
NSID	Nodal set ID, see *SET_NODES, containing nodes for initial velocity: If NSID = 0 the initial velocity is applied to all nodes.
NSIDEX	Nodal set I, see *SET_NODES, containing nodes that are exempted from the imposed velocities and may have other initial velocities.
BOXID	All nodes in box which belong to NSID are initialized. Nodes outside the box are not initialized. Exempted nodes are initialized to velocities defined by VXE, VYE, and VZE below regardless of their location relative to the box.
IRIGID	Option to overwrite rigid body velocities defined on *PART_INERTIA and *CONSTRAINED_NODAL_RIGID_BODY_INERTIA cards. GE.1: part set ID, containing ID of parts to overwrite. Center of gravity of part must lie within box BOXID. If BOXID is not defined then all parts defined in the set are overwritten. EQ.-1: Overwrite velocities for all *PART_INERTIA's and *CONSTRAINED_NODAL_RIGID_BODY_INERTIA's with a center of gravity within box BOXID. If BOXID is not defined then all are overwritten. EQ.-2: Overwrite velocities for all *PART_INERTIA's and *CONSTRAINED_NODAL_RIGID_BODY_INERTIA's.
ICID	Local coordinate system ID. The initial velocity is specified in the local coordinate system if ICID is greater than zero.
VX	Initial velocity in x-direction
VY	Initial velocity in y-direction
VZ	Initial velocity in z-direction
VXR	Initial rotational velocity about the x-axis
VYR	Initial rotational velocity about the y-axis
VZR	Initial rotational velocity about the z-axis
VXE	Initial velocity in x-direction of exempted nodes
VYE	Initial velocity in y-direction of exempted nodes
VZE	Initial velocity in z-direction of exempted nodes
VXRE	Initial rotational velocity in x-direction of exempted nodes
VYRE	Initial rotational velocity in y-direction of exempted nodes

VZRE	Initial rotational velocity in z-direction of exempted nodes

Remarks:

1. This generation input must not be used with *INITIAL_VELOCITY_GENERATION keyword.

2. If a node is initialized on more than one input card set, then the last set input will determine its velocity. However, if the nodal velocity is also specified on a *INITIAL_VELOCITY_NODE card, then the velocity specification on this card will be used.

3. Unless the option IRIGID is specified rigid bodies, initial velocities given in *PART_INERTIA will overwrite generated initial velocities. The IRIGID option will cause the rigid body velocities specified on the *PART_INERTIA input to be overwritten. To directly specify the motion of a rigid body without using the keyword, *PART_INERTIA, which also requires the definition of the mass properties, use the keyword option, *INITIAL_VELOCITY_RIGID_BODY.

4. Nodes which belong to rigid bodies must have motion consistent with the translational and rotational velocity of the center of gravity (c.g.) of the rigid body. During initialization the rigid body translational and rotational rigid body momentum's are computed based on the prescribed nodal velocity field. From this rigid body momentum, the translational and rotational velocities of the nodal points are computed and reset to the new values. These new values may or may not be the same as the values prescribed for the nodes that make up the rigid body. Sometimes this occurs in single precision due to numerical round-off. If a problem like this occurs specify the velocity using the keyword: *INITIAL_VELOCITY_ RIGID_BODY.

*INITIAL_VELOCITY_NODE

Purpose: Define initial nodal point velocities for a node.

Card	1	2	3	4	5	6	7	8
Variable	NID	VX	VY	VZ	VXR	VYR	VZR	ICID
Type	I	F	F	F	F	F	F	I
Default	none	0.	0.	0.	0.	0.	0.	0

VARIABLE	DESCRIPTION
NID	Node ID
VX	Initial translational velocity in x-direction
VY	Initial translational velocity in y-direction
VZ	Initial translational velocity in z-direction
VXR	Initial rotational velocity about the x-axis
VYR	Initial rotational velocity about the y-axis
VZR	Initial rotational velocity about the z-axis
ICID	Local coordinate system ID. The specified velocities are in the local system if ICID is greater than zero.

See Remarks on *INITIAL_VELOCITY card.

*INITIAL_VELOCITY_RIGID_BODY

Purpose: Define the initial translational and rotational velocities at the center of gravity (c.g.) for a rigid body or a nodal rigid body. This input overrides all other velocity input for the rigid body and the nodes which define the rigid body.

Card	1	2	3	4	5	6	7	8
Variable	PID	VX	VY	VZ	VXR	VYR	VZR	ICID
Type	I	F	F	F	F	F	F	I
Default	none	0.	0.	0.	0.	0.	0.	0

VARIABLE	DESCRIPTION
PID	Part ID of the rigid body or the nodal rigid body.
VX	Initial translational velocity at the c.g. in global x-direction.
VY	Initial translational velocity at the c.g. in global y-direction.
VZ	Initial translational velocity at the c.g. in global z-direction.
VXR	Initial rotational velocity at the c.g. about the global x-axis.
VYR	Initial rotational velocity at the c.g. about the global y-axis.
VZR	Initial rotational velocity at the c.g. about the global z-axis.
ICID	Local coordinate system ID. The specified velocities are in the local system if ICID is greater than zero.

See remarks 3 and 4 on the *INITIAL_VELOCITY input description.

*INITIAL_VELOCITY_GENERATION

Purpose: Define initial velocities for rotating and translating bodies. Caution: Rigid body velocities cannot be reinitialized after dynamic relaxation by setting PHASE=1 since rigid body velocities are always restored to the values that existed prior to dynamic relaxation. Reinitialization of velocities after dynamic relaxation is only for nodal points of deformable bodies; therefore, if rigid bodies are present in the part set ID, this input should be defined twice, once for IPHASE=0 and again for IPHASE=1.

Card 1

Variable	ID	STYP	OMEGA	VX	VY	VZ	IVATN	ICID
Type	I	I	F	F	F	F	I	I
Default	none	none	0.	0.	0.	0.	0	0

Card 2

Variable	XC	YC	ZC	NX	NY	NZ	PHASE	IRIGID
Type	F	F	F	F	F	F	I	I
Default	0.	0.	0.	0.	0.	0.	0	0

VARIABLE	DESCRIPTION
ID	Part ID, part set ID, or node set ID if zero STYP is ignored and all velocities are set. WARNING if IVATN=0: If a part ID of a rigid body is specified only the nodes that belong to elements of the rigid body are initialized. Nodes defined under the keyword. *CONSTRAINED_ EXTRA_NODES are not initialized. Set IVATN=1 to initialize velocities of slaved nodes and parts.
STYP	Set type: EQ.1: part set ID, see *SET_PART, EQ.2: part ID, see *PART, EQ.3: node set ID, see *SET_NODE.
OMEGA	Angular velocity about the rotational axis.
VX	Initial translational velocity in global x-direction.

VARIABLE	DESCRIPTION
VY	Initial translational velocity in global y-direction.
VZ	Initial translational velocity in global z-direction.
IVATN	Flag for setting the initial velocities of slave nodes and parts: EQ.0: slaved parts are ignored. EQ.1: slaved parts and slaved nodes of the master parts will be assigned initial velocities like the master part.
ICID	Local coordinate system ID. The specified velocities are in the local system if ICID is greater than zero. Note that this requires XC, YC, and ZC to also be specified in the local coordinate system.
XC	x-coordinate on rotational axis.
YC	y-coordinate on rotational axis.
ZC	z-coordinate on rotational axis.
NX	x-direction cosine.
NY	y-direction cosine.
NZ	z-direction cosine.
PHASE	Flag specifying phase of the analysis the velocities apply to: EQ.0. Velocities are applied immediately, EQ.1. Velocities are applied after reaching the start time, STIME, which is after dynamic relaxation, if active, is completed. See the keyword: *INITIAL_VELOCITY_GENERATION_START_TIME. STIME defaults to zero.
IRIGID	Option to overwrite or automatically set rigid body velocities defined on the *PART_INERTIA and *CONSTRAINED_NODAL_RIGID_BODY _INERTIA cards. EQ.1: Reset the rigid body velocites for *PART ID or all parts in *SET_PART ID. This option does not apply for STYP=3.

Remarks:

1. This generation input must not be used with *INITIAL_VELOCITY or *INITIAL_ VELOCITY_NODE options.

2. The velocities are initialized in the order the *INITIAL_VELOCITY_GENERATION input is defined. Later input via the *INITIAL_VELOCITY_GENERATION keyword may overwrite the velocities previously set.

3. For rigid bodies, initial velocities given in *PART_INERTIA will overwrite generated initial velocities.

4. Nodes which belong to rigid bodies must have motion consistent with the translational and rotational velocity of the rigid body. During initialization the rigid body translational and rotational rigid body momentum's are computed based on the prescribed nodal velocities. From this rigid body motion the velocities of the nodal points are computed and reset to the new values. These new values may or may not be the same as the values prescribed for the node.

*INITIAL_VELOCITY_GENERATION_START_TIME

Purpose: Define a time to initialize velocities after time zero. Time zero starts after dynamic relaxation if used for initialization. This option can be applied if and only if PHASE=1 is specified for at least one *INITIAL_VELOCITY_GENERATION definitions.

Card 1	1	2	3	4	5	6	7	8
Variable	STIME							
Type	I							
Default	0.0							

VARIABLE	DESCRIPTION
STIME	Start time.

INITIAL_VOID_OPTION*

Available options include:

PART

SET

Purpose: Define initial voided part set ID's or part numbers. Void materials cannot be created during the calculation. Fluid elements which are evacuated, e.g., by a projectile moving through the fluid, during the calculation are approximated as fluid elements with very low densities. The constitutive properties of fluid materials used as voids must be identical to those of the materials which will fill the voided elements during the calculation. Mixing of two fluids with different properties is not permitted with this option.

Card 1	1	2	3	4	5	6	7	8
Variable	PSID/PID							
Type	I							
Default	none							
Remark	1							

VARIABLE	DESCRIPTION
PSID/PID	Part set ID or part ID, see also *SET_PART:

Remarks:

This void option and multiple materials per element, see *ALE_MULTI-MATERIAL_GROUP are incompatible and cannot be used together in the same run.

*INITIAL_VOLUME_FRACTION

Purpose: Define initial volume fractions of different materials in multi-material ALE elements.

Card 1	1	2	3	4	5	6	7	8
Variable	EID	VF1	VF2	VF3	VF4	VF5	VF6	VF7
Type	I	F	F	F	F	F	F	F
Default	none	0.0	0.0	0.0	0.0	0.0	0.0	0.0

VARIABLE	DESCRIPTION
EID	Element ID.
VF1	Volume fraction of multi-material group 1, AMMGID=1.
VF2	Volume fraction of multi-material group 2. Only needed in simulations with 3 material groups. Otherwise VF2=1-VF1.
VF3	Volume fraction of multi-material group 3, AMMGID=3.
VF4	Volume fraction of multi-material group 4, AMMGID=4.
VF5	Volume fraction of multi-material group 5, AMMGID=5.
VF6	Volume fraction of multi-material group 6, AMMGID=6.
VF7	Volume fraction of multi-material group 7, AMMGID=7.

*INITIAL_VOLUME_FRACTION_GEOMETRY

Purpose: This is a volume-filling command for defining the volume fractions of various ALE multi-material groups (AMMG) that can occupy certain regions in some specified ALE mesh set. It is applied only for multi-material ALE model. See Remark 5.

Defines the background ALE mesh set & an AMMGID that initially fills it.

Card 1	1	2	3	4	5	6	7	8
Variable	FMSID	FMIDTYP	BAMMG	NTRACE				
Type	I	I	I	I				
Default	none	0	0	3				

VARIABLE	DESCRIPTION
FMSID	A background ALE (fluid) mesh SID to be initialized or filled with various AMMG's. This set ID refers to one or more ALE parts.
FMIDTYP	ALE mesh set ID type: EQ.0: FMSID is an ALE part set ID (PSID). EQ.1: FMSID is an ALE part ID (PID).
BAMMG	The background fluid group ID or ALE Multi-Material group ID (AMMGID) that initially fills all ALE mesh region defined by FMSID.
NTRACE	Number of sampling points for volume filling detection. Typically NTRACE ranges from 3 to maybe 10 (or more). The higher it is, the finer the ALE element is divided so that small gaps between 2 Lagrangian shells may be filled in. See Remark 6.

Defines the container type and the AMMGID that fills inside or outside it.

Card a	1	2	3	4	5	6	7	8
Variable	CONTTYP	FILLOPT	FAMMG					
Type	I	I	I					
Default	none	0	none					

VARIABLE **DESCRIPTION**

CONTTYP A "container" defines a Lagrangian surface boundary of a spatial region, inside (or outside) of which, an AMMG would fill up. CONTTYP defines the container geometry type of this surface boundary (or shell structure).

EQ.1: The container geometry is defined by a part ID (**PID**) or a part set ID (**PSID**), where the parts should be defined by shell elements (see *PART or *SET_PART).

EQ.2: The container geometry is defined by a **segment set** (SGSID).

EQ.3: The container geometry is defined by a **plane**: a point and a normal vector.

EQ.4: The container geometry is defined by a **conical surface**: 2 end points and 2 corresponding radii.

EQ.5: The container geometry is defined by a **cuboid or rectangular box**: 2 opposing end points, minimum to maximum coordinates.

EQ.6: The container geometry is defined by a **sphere**: 1 center point, and a radius.

FILLOPT A flag to indicate which side of the container surface the AMMG is supposed to fill. The "head" side of a container surface/segment is defined as the side pointed to by the heads of the normal vectors of the segments ("tail" side refers to opposite direction to "head"). See Remark 7.

EQ.0: The "head" side of the geometry defined above will be filled with fluid (default).

EQ.1: The "tail" side of the geometry defined above will be filled with fluid.

FAMMG This defines the fluid group ID or ALE Multi-Material group ID (AMMGID) which will fill up the interior (or exterior) of the space defined by the "container". The order of AMMGIDs are defined by the order in which they are listed under *ALE_MULTI-MATERIAL_GROUP card. For example, that card defines AMMGID=1 on its first line, and AMMGID=2 on its second line, etc.

CONTTYP = 1 for container defined by a shell PID or PSID

Card b-1	1	2	3	4	5	6	7	8
Variable	SID	STYPE	NORMDIR	XOFFSET				
Type	I	I	I	F				
Default	none	0	0	0.0				
Remark			obsolete					

VARIABLE	DESCRIPTION
SID	A Set ID pointing to a part ID (PID) or part set ID (PSID) of the Lagrangian shell element structure defining the "container" geometry to be filled (see *PART or *SET_PART).
SSTYPE	Set ID type: EQ.0: Container SID is a Lagrangian part set ID (PSID). EQ.1: Container SID is a Lagrangian part ID (PID).
NORMDIR	Obsolete (see Remark 7).
XOFFSET	Absolute length unit for offsetting the fluid interface from the nominal fluid interface LS-DYNA would otherwise define by default. This parameter only applies to GEOTYPE=1 (4^{th} column) and GEOTYPE=2 (3^{rd} column). This is applicable to cases in which high pressure fluid is contained within a container. The offset allows LS-DYNA time to prevent leakage. In general, this may be set to roughly 5-10% of the ALE elm width. It may be important only for when ILEAK is turned ON to give the code time to "catch" the leakage. If ILEAK is not ON, this may not be necessary.

CONTTYP = 2 for container defined by a SGSID

Card b-2	1	2	3	4	5	6	7	8
Variable	SGSID	NORMDIR	XOFFSET					
Type	I	I	F					
Default	none	0	0.0					
Remark		obsolete						

VARIABLE	DESCRIPTION
SGSID	Segment Set ID defining the "container", see *SET_SEGMENT.
NORMDIR	Obsolete (see Remark 7).
XOFFSET	Absolute length unit for offsetting the fluid interface from the nominal fluid interface LSDYNA would otherwise define by default. This parameter only applies to GEOTYPE=1 (4^{th} column) and GEOTYPE=2 (3^{rd} column). This is applicable to cases in which high pressure fluid is contained within a container. The offset allows LS-DYNA time to prevent leakage. In general, this may be set to roughly 5-10% of the ALE elm width. It may be important only for when ILEAK is turned ON to give the code time to "catch" the leakage. If ILEAK is not ON, this may not be necessary.

CONTTYP = 3 for container defined by a PLANE

Card b-3	1	2	3	4	5	6	7	8
Variable	X0	Y0	Z0	XCOS	YCOS	ZCOS		
Type	F	F	F	F	F	F		
Default	none	none	none	none	none	none		

VARIABLE	DESCRIPTION
X0, Y0, Z0	X, Y and Z coordinate of a spatial point on the plane.
X1, Y1, Z1	X, Y and Z direction cosines of the plane normal vector. The filling will occur on the side pointed to by the plane normal vector (or "head" side).

CONTTYP = 4 for container defined by a Cylinder and Cone.

Card b-4

1	2	3	4	5	6	7	8
X0	Y0	Z0	X1	Y1	Z1	R1	R2
F	F	F	F	F	F	F	F
none	none	none	none	none	none	none	none

VARIABLE	DESCRIPTION
X0, Y0, Z0	X, Y and Z coordinate of the center of the 1st base of the cone.
X1, Y1, Z1	X, Y and Z coordinate of the center of the 2nd base of the cone.
R1	Radius of the 1st base of the cone
R2	Radius of the 2nd base of the cone

CONTTYP = 5 for container defined by a Rectangular Box

Card b-5

1	2	3	4	5	6	7	8
X0	Y0	Z0	X1	Y1	Z1		
F	F	F	F	F	F		
none	none	none	none	none	none		

VARIABLE	DESCRIPTION
X0, Y0, Z0	X, Y and Z coordinate of the maximum coordinate of the box.
X1, Y1, Z1	X, Y and Z coordinate of the minimum coordinate of the box.

CONTTYP = 6 for container defined by a Sphere

Card b-6	1	2	3	4	5	6	7	8
Variable	X0	Y0	Z0	R0				
Type	F	F	F	F				
Default	none	none	none	none				

VARIABLE	DESCRIPTION
X0, Y0, Z0	X, Y and Z coordinate of the center of the sphere.
R0	Radius of the sphere

Remarks:

1. After card **1** defining the basic mesh filled by certain fluid group (AMMGID), each "filling action" will require 2 additional lines of input (cards **a** and **b-#**, where **#** is the CONTTYP value). At the minimum there will be 3 cards required for this command (**1**, **a** and **b-#**) for 1 "filling action".

2. There can be one or more "filling actions" prescribed for each definition of this command. The "filling actions" take place in the prescribed order and the effects are accumulative. The latter filling actions over-write the previous ones. Therefore any complex filling logics will require some planning. For example, the following card sequence, with 2 "filing actions", is allowable:

   1

   a (CONTTYP=1)

   b-1

   a (CONTTYP=3)

   b-3

   This sequence of cards prescribes a system of background ALE mesh with 2 "filing actions" to be executed. The 1st is a filling of a CONTTYP=1, and the 2nd of CONTTYP=3.

3. Card **a** is required for all container geometry types (CONTTYP). Card **b-#** defines the container actual geometry and corresponds to each of the CONTTYP choice.

4. If ELFORM=12 (in *SECTION_SOLID) for the ALE mesh to be filled, i.e. single-material-and-void element formulation is used, then the non-void material is automatically referred to as AMMG 1 and the void as AMMG 2. The multi-material groups are implied even though no *ALE_MULTI-MATERIAL_GROUP card is required.

5. A simple ALE background mesh (for example, a cuboid mesh) can be constructed enveloping some Lagrangian shell structure (or container). The ALE region inside this Lagrangian shell container may be filled with one multi-material group (AMMG1), and the outside region with another (AMMG2). This approach simplifies the mesh generation requirements for ALE material parts with complex geometries.

6. Default is NTRACE=3 ➔ total number is $(2*NTRACE+1)^3$ = (7X7X7). This means an ALE element is subdivided into 7X7X7 regions. Each is to be filled in with the appropriate AMMG. An example of this application would be the filling of initial gas between multiple layers of Lagrangian airbag shell elements sharing the same ALE element.

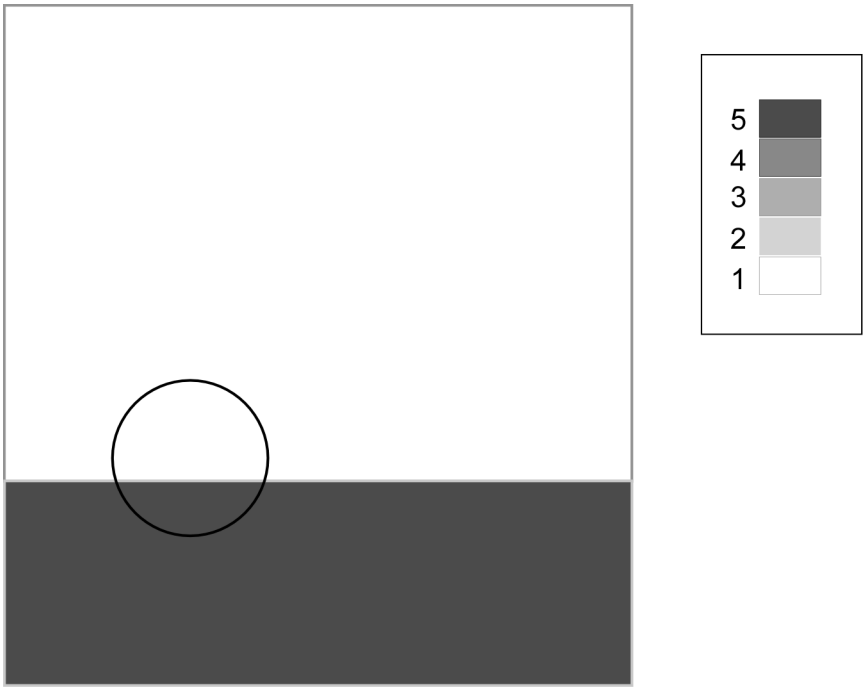
7. The simplest approach is to:
   (a) Set "NORMDIR=0" (default).
   (b) Define the shell (or segment) container with inward normals.
   (c) Using only FILLOPT in card **a** to control which side of the container to fill.

Example:

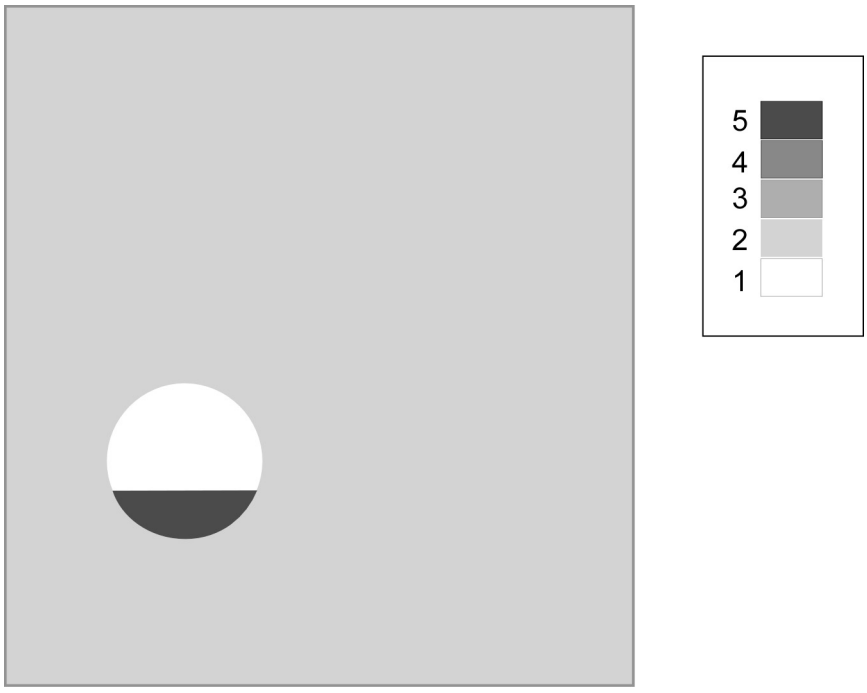
Consider a simple ALE model with ALE parts H1-H5 (5 AMMGs possible) and 1 Lagrangian shell (container) part S6. Only parts H1 and S6 initially have their meshes defined. We will perform 4 "filling actions". The volume filling results after each step will be shown below to clarify the concept used. The input for the volume filling looks like this.

```
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|....8
$ H1 = AMMG 1 = fluid 1 initially occupying whole ALE mesh= background mesh
$ H5 = AMMG 5 = fluid 5 fills below a plane = filling action 1 = CONTTYP=3
$ H2 = AMMG 2 = fluid 2 fills outside S5    = filling action 2 = CONTTYP=1
$ H3 = AMMG 3 = fluid 3 fills inside a cone = filling action 3 = CONTTYP=4
$ H4 = AMMG 4 = fluid 4 fills inside a box  = filling action 4 = CONTTYP=5
$ S6 =          Lagrangian shell container
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|....8
*ALE_MULTI-MATERIAL_GROUP
         1         1
         2         1
         3         1
         4         1
         5         1
*INITIAL_VOLUME_FRACTION_GEOMETRY
$ The 1st card fills the whole pid H1 with AMMG 1=background ALE mesh
$    FMSID   FMIDTYP     BAMMG     <=== card 1: background fluid
         1         1         1
$ filling action 1 = AMMG 5 fill all elms below a plane
$ CONTTYPE   FILLOPT FILAMMGID     <=== card a  : container: CONTTYPE=3=plane
         3         0         5
$   X0, Y0,   Z0,    NX,  NY, NZ   <=== card b-3: details on container =plane
  25.0,20.0, 0.0,   0.0,-1.0,0.0
$ filling action 2: AMMG 2 fills OUTSIDE (FILLOPT=1) shell S6 (inward normals);
$ CONTTYPE   FILLOPT     FAMMG  <== card a  : container #1; FILLOPT=1=fill tail
         1         1         2
$    SETID   SETTYPE   NORMDIR  <== card b-1: details on container #1
         6         1         0
$ filling action 3 = AMMG 3 fill all elms inside a CONICAL region
$ CONTTYPE   FILLOPT     FAMMG  CONTTYP = 4 = Container = conical region
         4         0         3
$       X1        Y1        Z1        X2        Y2        Z2        R1        R2
      25.0      75.0       0.0      25.0      75.0       1.0       8.0       8.0
$ filling action 4 = AMMG 4 fill all elms inside a BOX region
$ CONTTYPE   FILLOPT  FFLUIDID                  : CONTTYP=5 = "BOX"
         5         0         4
$     XMIN      YMIN      ZMIN      XMAX      YMAX      ZMAX
      65.0      35.0       0.0      85.0      65.0       1.0
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|....8
```

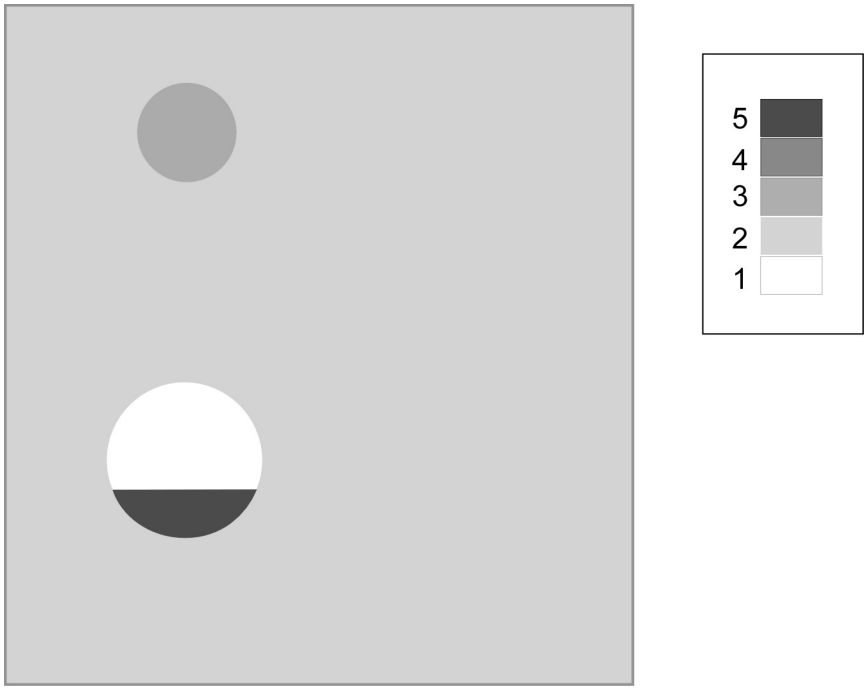
Before the 1st "filling action" the whole ALE mesh of part H1 is filled with AMMG 1 (white). After the 1st "filling action", AMMG 5 fills below the specified plane.



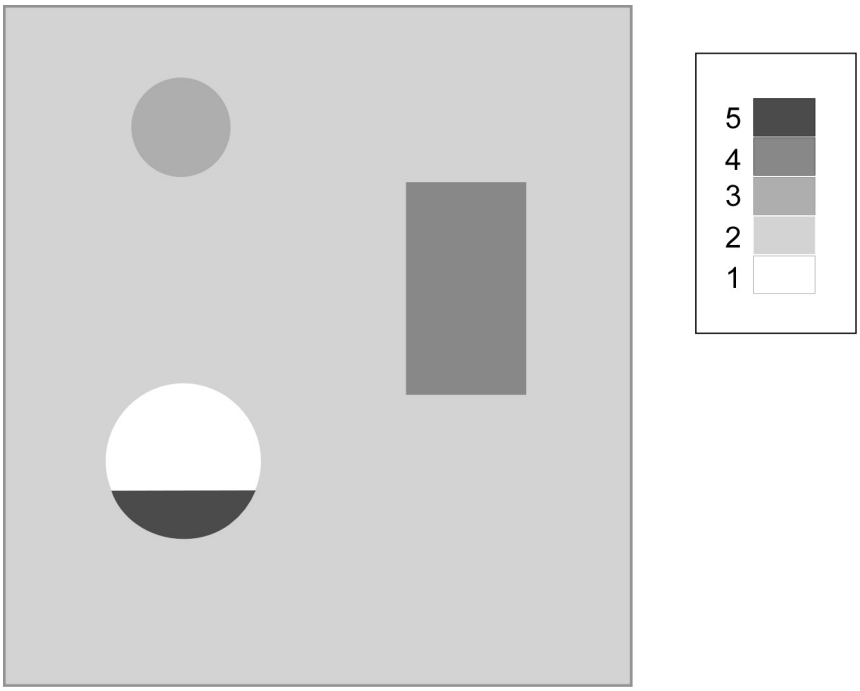
After the 1st and 2nd "filling actions", it fills outside the shell (S6) with AMMG 2.



After the 1st, 2nd and 3rd "filling actions", it fills in the analytical sphere with AMMG 3.



After the 1st, 2nd, 3rd and 4th "filling actions", it fills in the analytical box with AMMG 4.



*INTEGRATION

In this section the user defined integration rules for beam and shell elements are specified. IRID refers to integration rule identification number on *SECTION_BEAM and *SECTION_ SHELL cards respectively. Quadrature rules in the *SECTION_SHELL and *SECTION_BEAM cards need to be specified as a negative number. The absolute value of the negative number refers to user defined integration rule number. Positive rule numbers refer to the built in quadrature rules within LS-DYNA. The keyword cards in this section are:

***INTEGRATION_BEAM**

***INTEGRATION_SHELL**

***INTEGRATION_BEAM**

Purpose: Define user defined through the thickness integration rules for the beam element.

Card 1	1	2	3	4	5	6	7	8
Variable	IRID	NIP	RA	ICST	K			
Type	I	I	F	I	I			
Default	none	0	0.0	0	0			

Define the following card if and only if ICST>0.

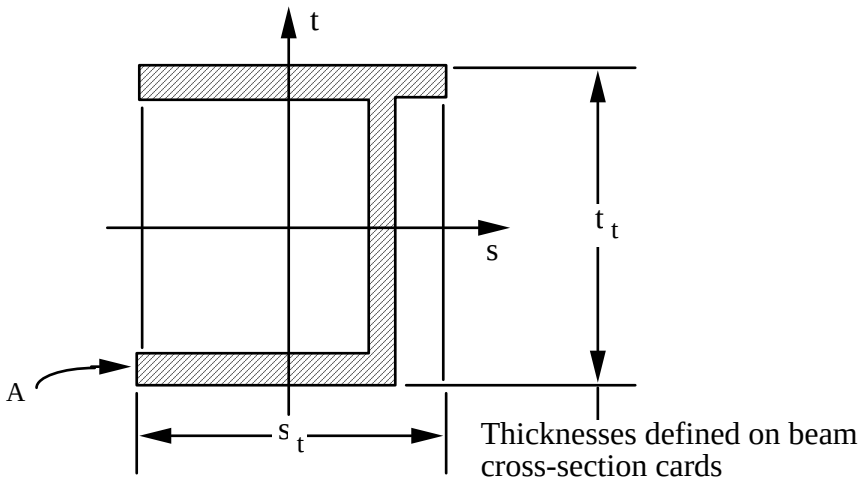
Card	1	2	3	4	5	6	7	8
Variable	D1	D2	D3	D4	SREF	TREF	D5	D6
Type	F	F	F	F	F	F	F	F
Default	none	none	none	none	1.0	0.0	none	none

Define NIP cards below (Skip if NIP=0).

Card	1	2	3	4	5	6	7	8
Variable	S	T	WF	PID				
Type	F	F	F	I				

VARIABLE	DESCRIPTION
IRID	Integration rule ID. IRID refers to IRID on *SECTION_BEAM card.
NIP	Number of integration points, see also ICST.
RA	Relative area of cross section, i.e., the actual cross-sectional area divided by the area defined by the product of the specified thickness in the s direction and the thickness in the t direction. See also ICST below and Figure 19.1.
ICST	Standard cross section type, ICST. If this type is nonzero then NIP and the relative area above should be input as zero. See the discussion following the input description Figure 19.3. EQ. 01: I-shape EQ. 02: Channel EQ. 03: L-shape EQ. 04: T-shape EQ. 05: Tubular box EQ. 06: Z-shape EQ. 07: Trapezoidal EQ. 08: Circular EQ. 09: Tubular EQ. 10: I-shape1 EQ. 11: Solid box EQ. 12: Cross EQ. 13: H-shape EQ. 14: T-shape1 EQ. 15: I-shape2 EQ. 16: Channel1 EQ. 17: Channel2 EQ. 18: T-shape2 EQ. 19: Box-shape1 EQ. 20: Hexagon EQ. 21: Hat-shape EQ. 22: Hat-shape1
K	Integration refinement parameter for standard cross section types. Select an integer ≥ 0. See Figure below.
D1-D6	Cross-section dimensions. See Figure below.
SREF	s_{ref}, location of reference surface normal to s, for the Hughes-Liu beam only. This option is only useful if the beam is connected to a shell or another beam on its outer surface. Overrides NSLOC in *SECTION_BEAM.
TREF	t_{ref}, location of reference surface normal to t, for the Hughes-Liu beam only. This option is only useful if the beam is connected to a shell or another beam on its outer surface. Overrides NTLOC in *SECTION_BEAM.
S	Normalized s coordinate of integration point, $-1 \leq s \leq 1$.
T	Normalized t coordinate of integration point, $-1 \leq t \leq 1$.
WF	Weighting factor, A_{ri}, i.e., the area associated with the integration point divided by actual cross sectional area $A_{ri} = A_i / A$, see Figure 19.2.

VARIABLE	DESCRIPTION
PID	Optional PID, used to identify material properties for this integration point. If zero, the "master" PID (referenced on *ELEMENT) will be used. This feature will be available in release 3 of version 971.



$$\text{Relative Area} = \frac{A}{s_t \cdot t_t}$$

Figure 19.1. Definition of relative area for user defined integration rule.

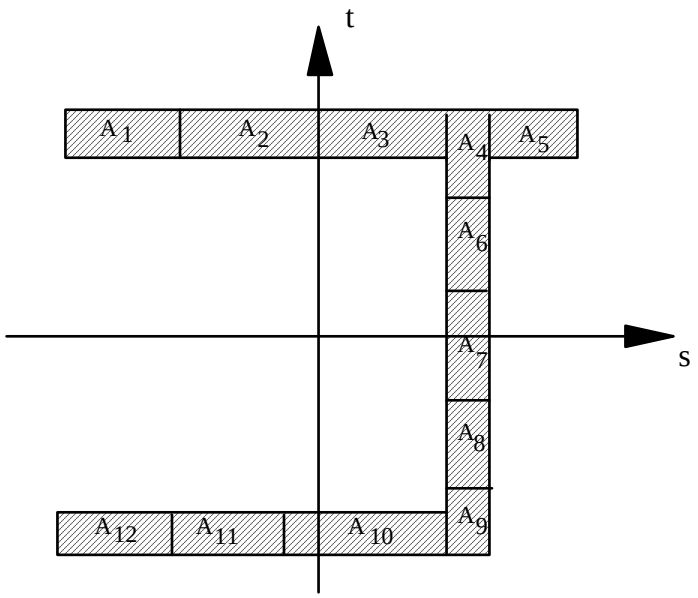


Figure 19.2. Definition of integration points for user defined integration rule.

Remarks:

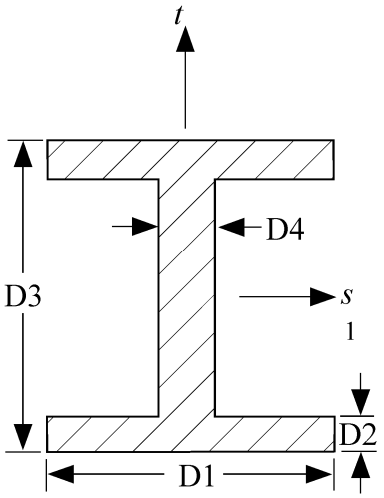
The input for standard beam section types is defined below. In following figures the dimensions are shown on the left and the location of the integration points are shown on the right. If a quantity is not defined in the sketch, then it should be set to zero in the input. The input quantities include:

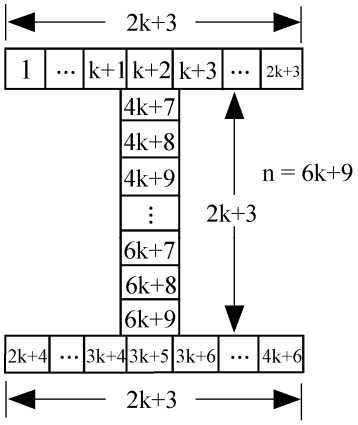
D1-D6 = Dimensions of section

k = Integration refinement parameter (an integer GE. 0)

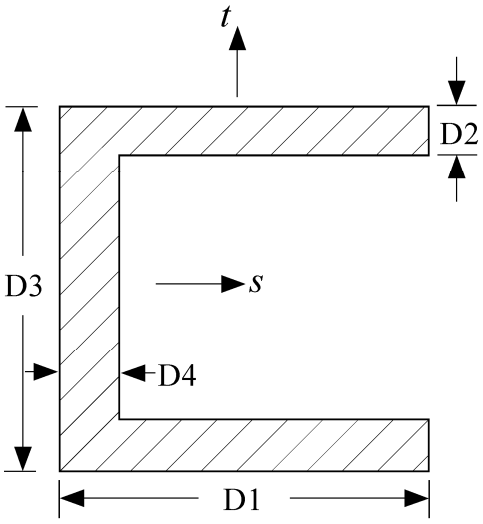
s_{ref} = location of reference surface normal to s, Hughes-Liu beam only

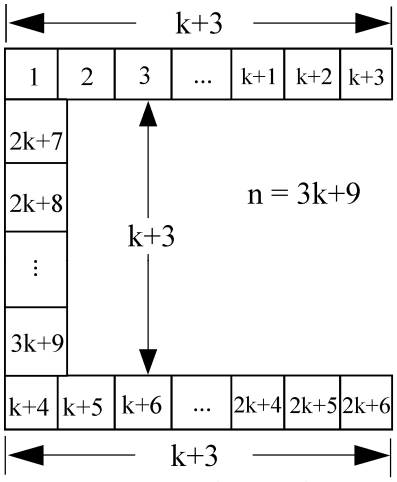
t_{ref} = location of reference surface normal to t, Hughes-Liu beam only



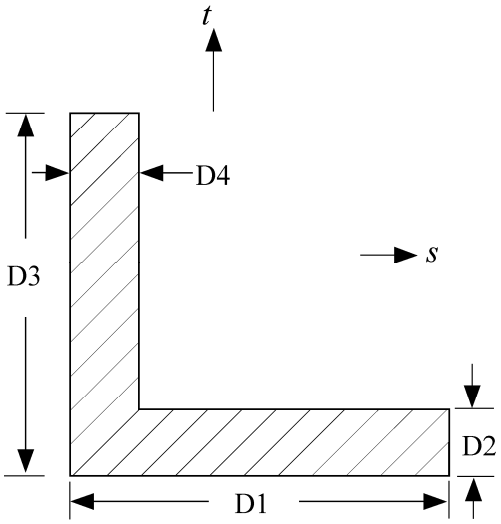


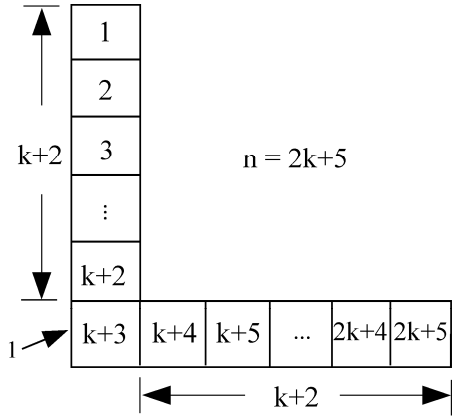
Type 1: I-shape



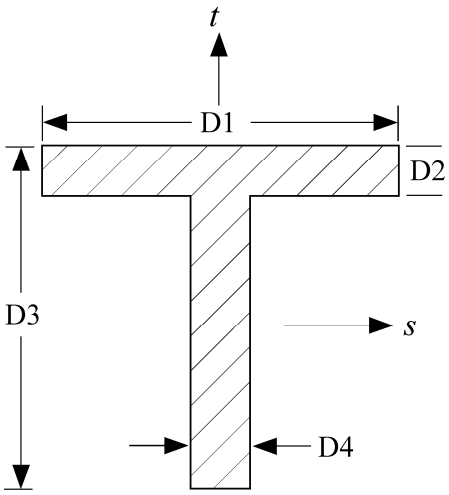


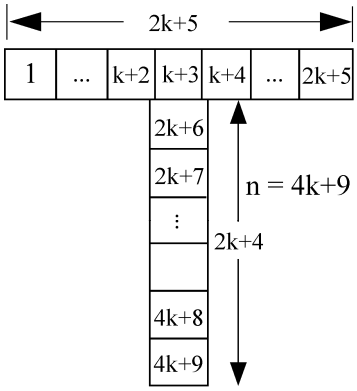
Type 2: Channel



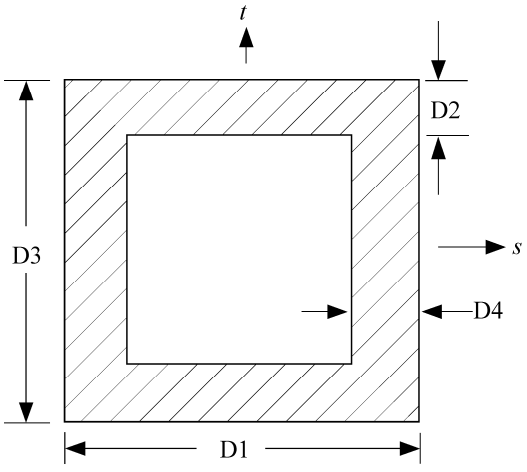


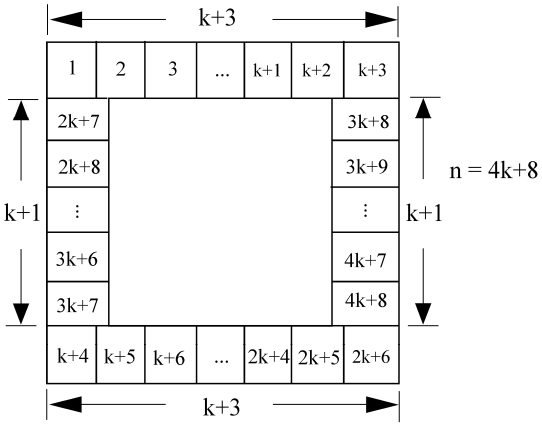
Type 3: L-shape



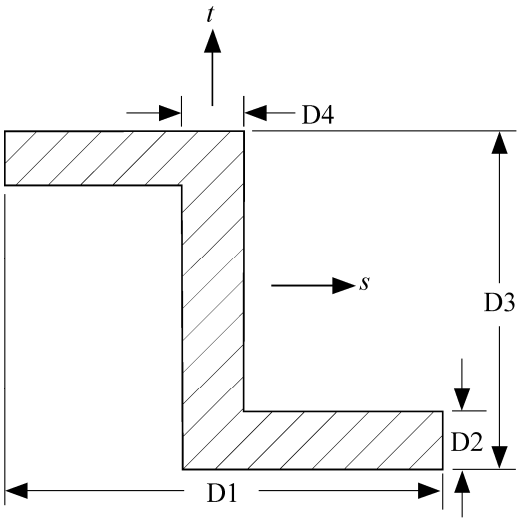


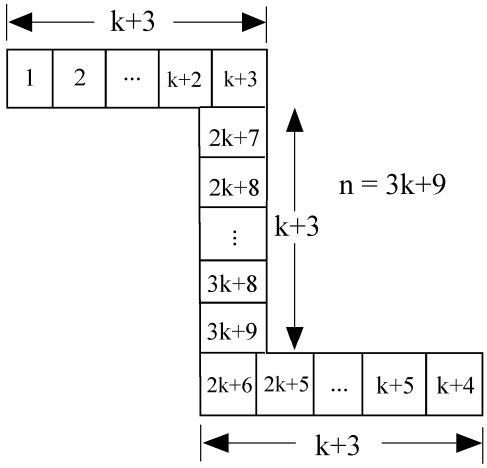
Type 4: T-shape



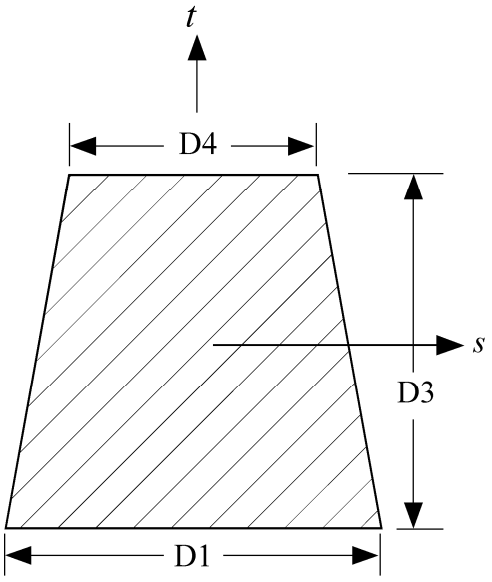


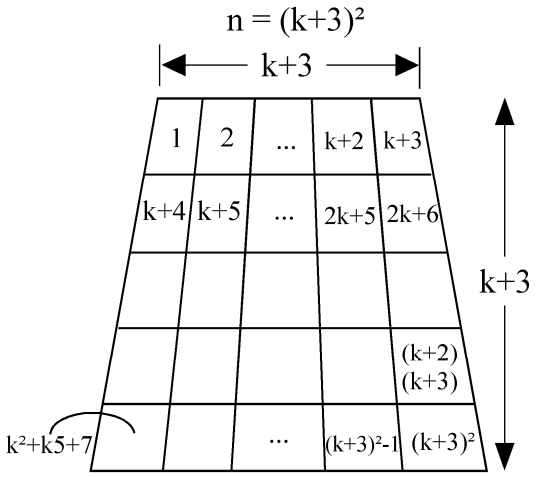
Type 5: Box-shape



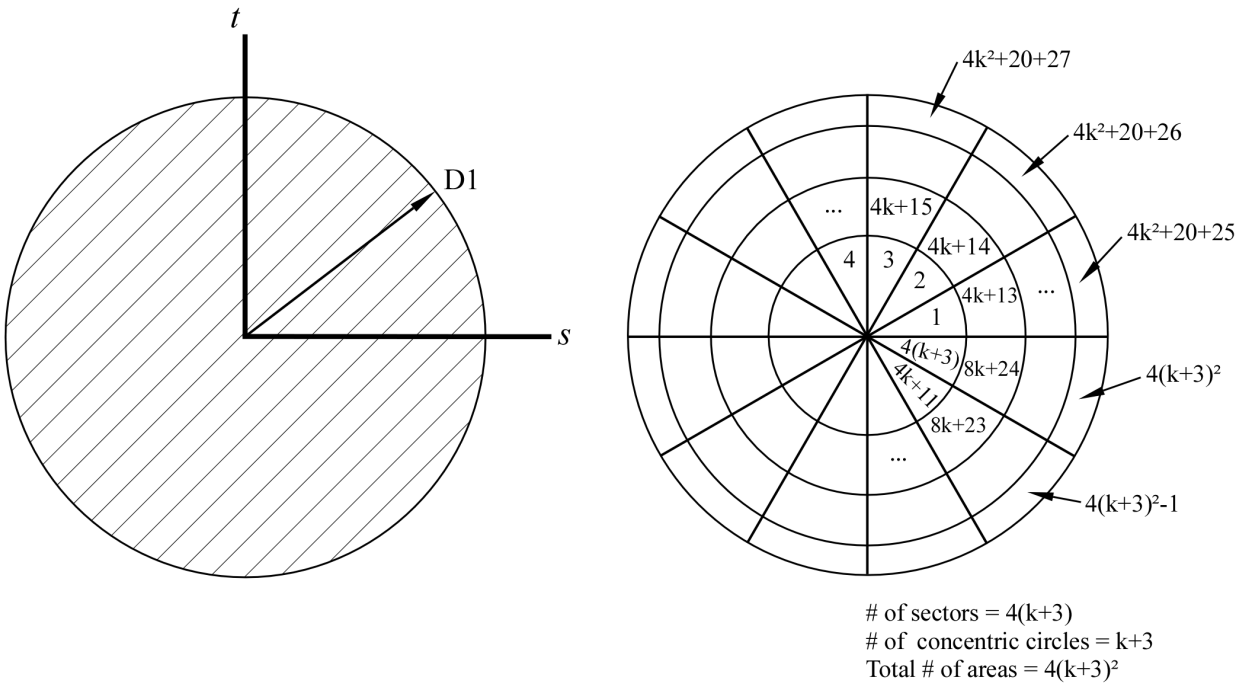


Type 6: Z-shape



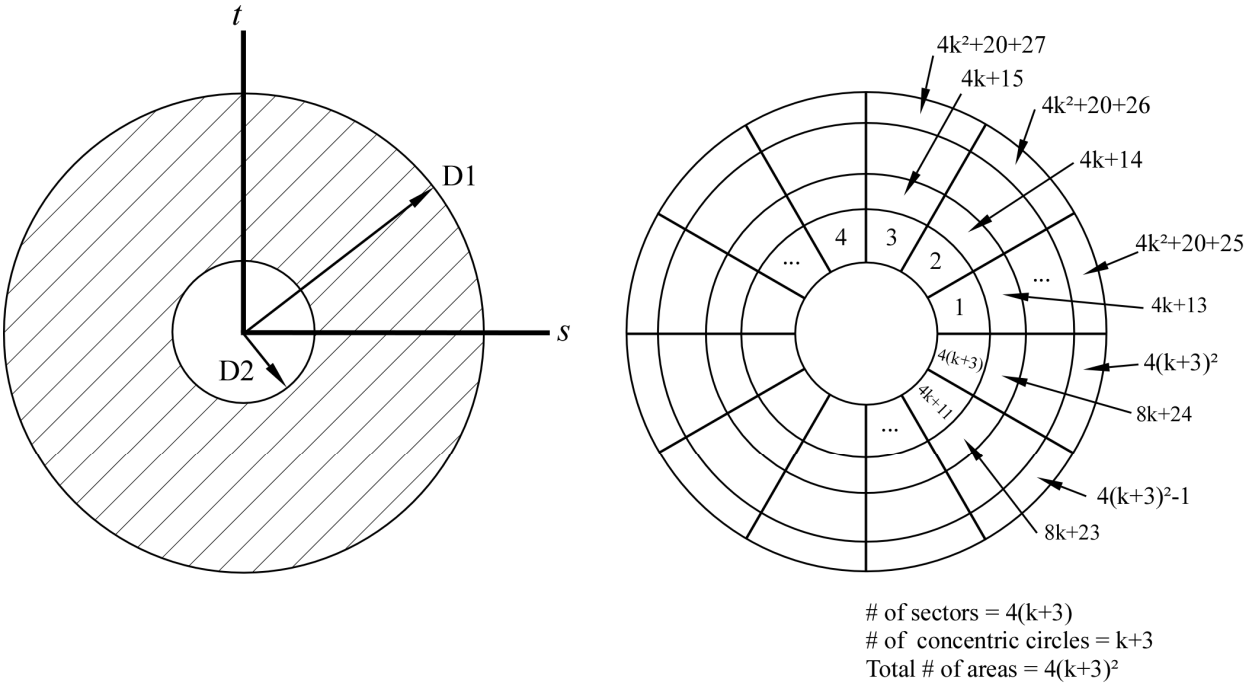


Type 7: Trapezoidal



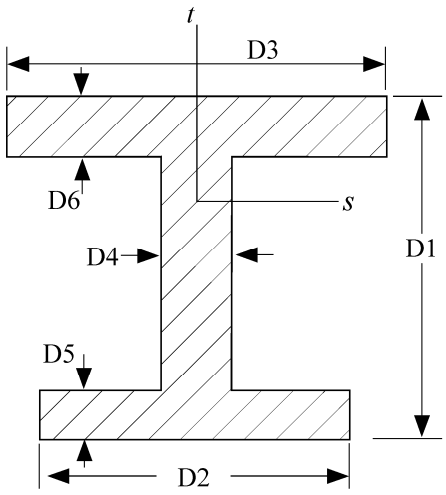
# of sectors = 4(k+3)
# of concentric circles = k+3
Total # of areas = 4(k+3)²

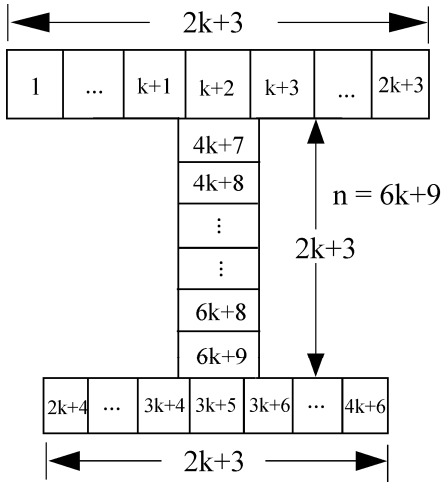
Type 8: Circular



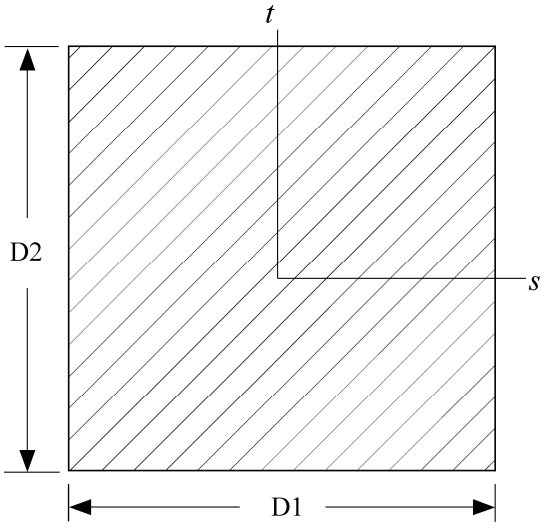
# of sectors = 4(k+3)
# of concentric circles = k+3
Total # of areas = 4(k+3)²

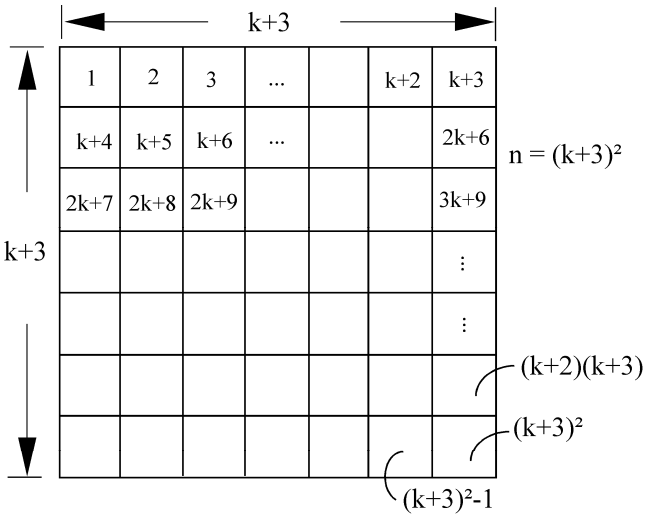
Type 9: Tubular



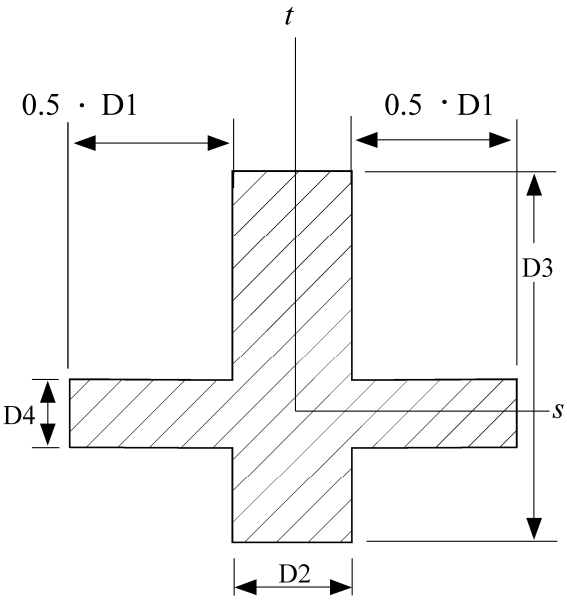


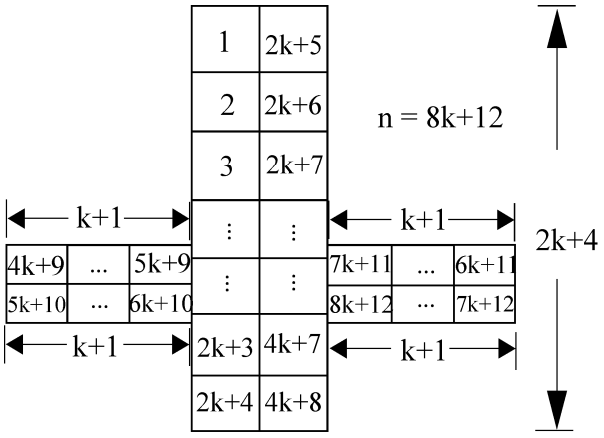
Type 10: I-shape1



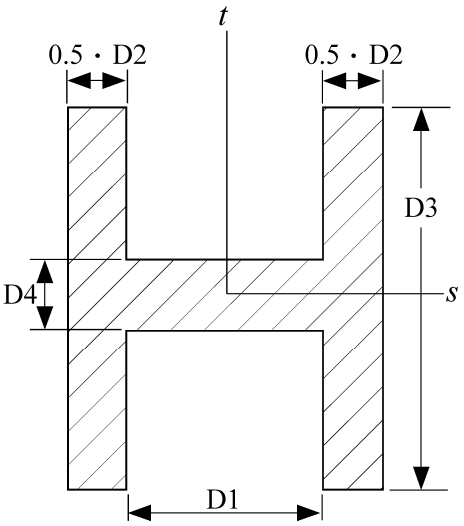


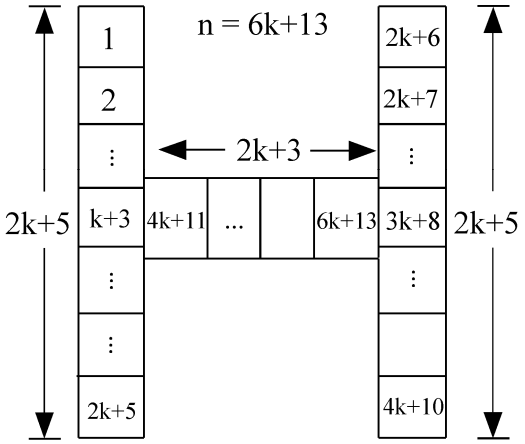
Type 11: Solid Box



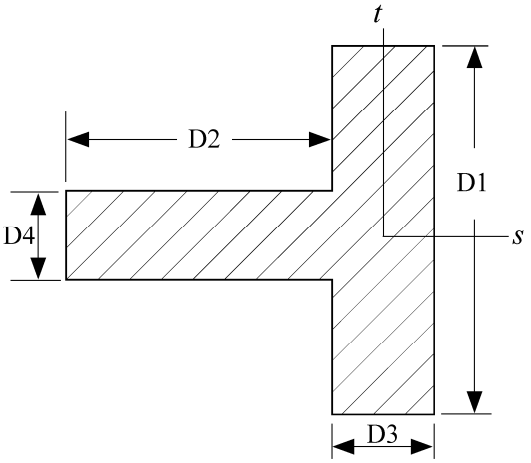


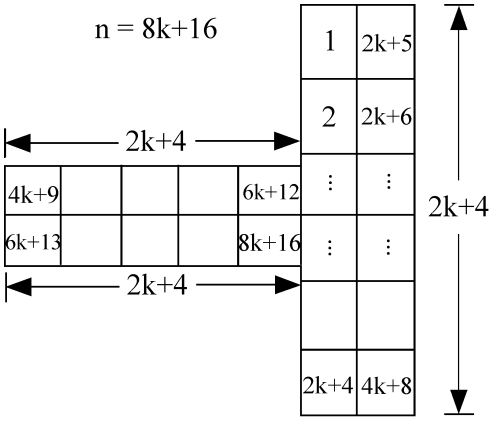
Type 12: Cross



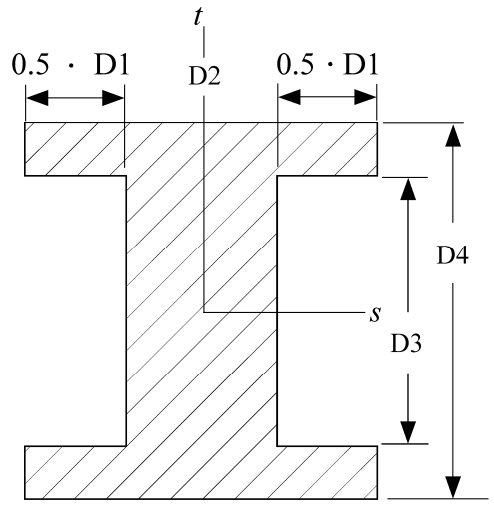


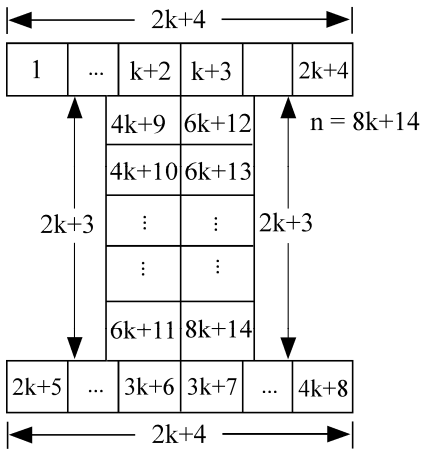
Type 13: H-shape



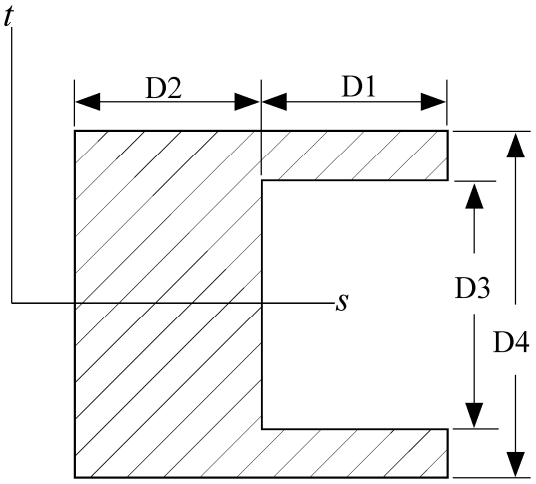


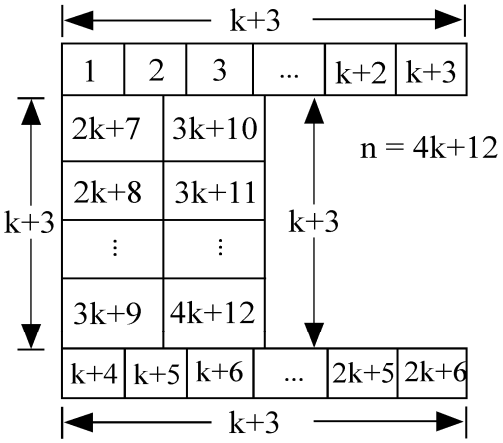
Type 14: T-shape1



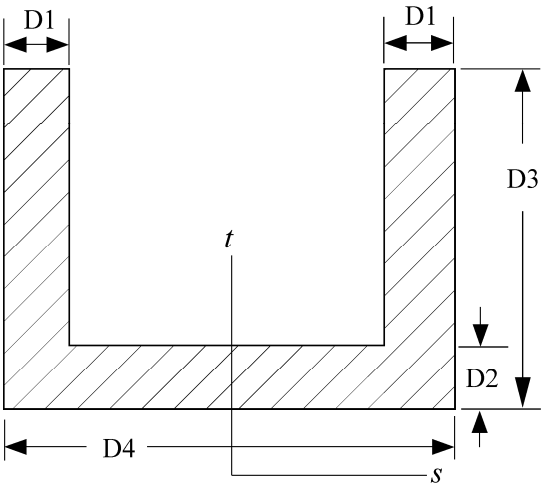


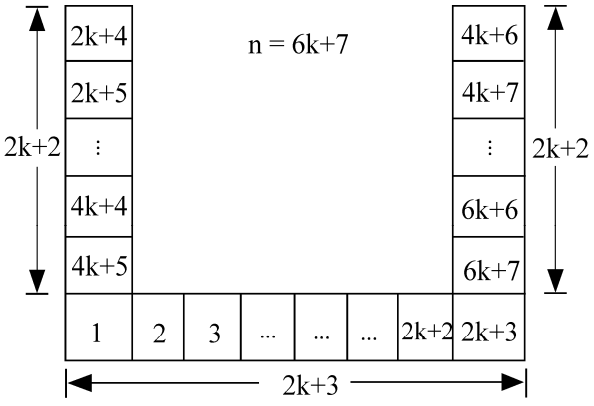
Type 15: I-shape2



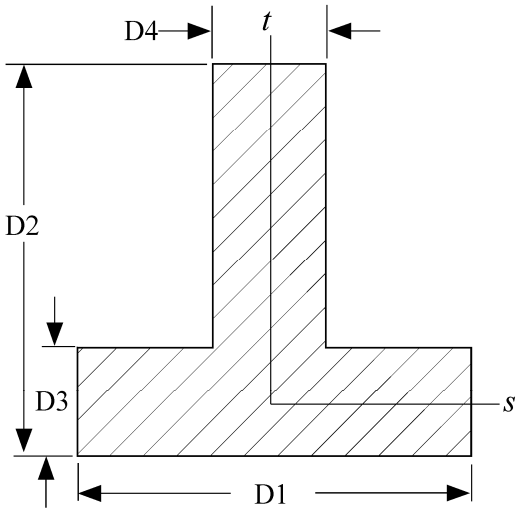


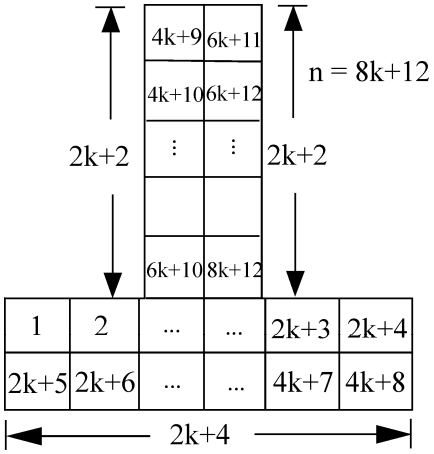
Type 16: Channel1



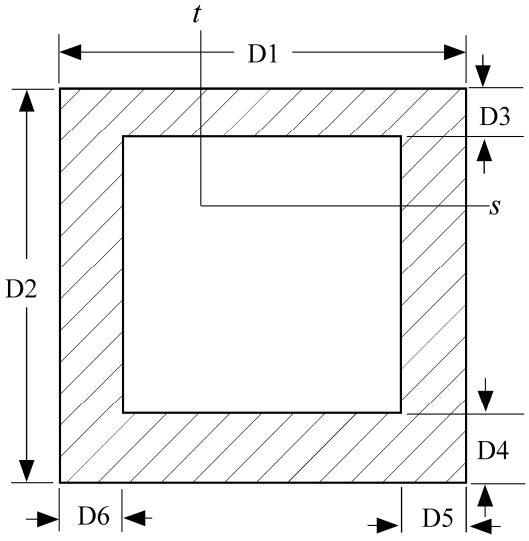


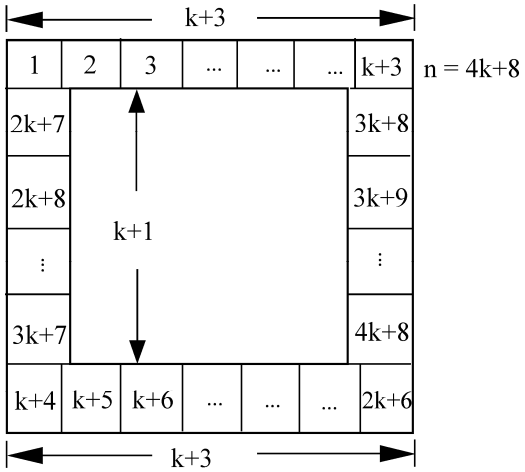
Type 17: Channel2



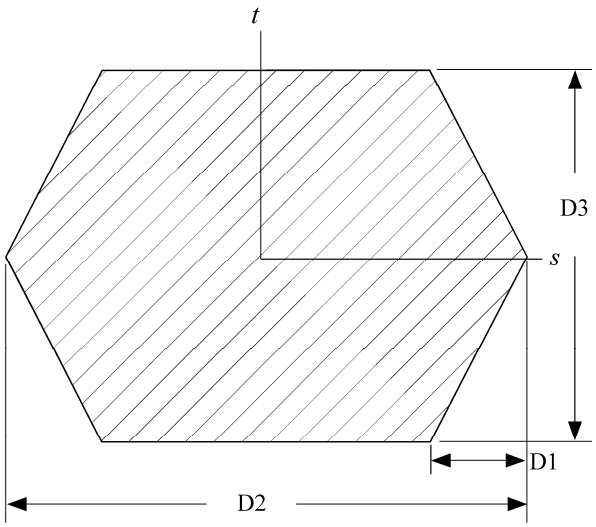


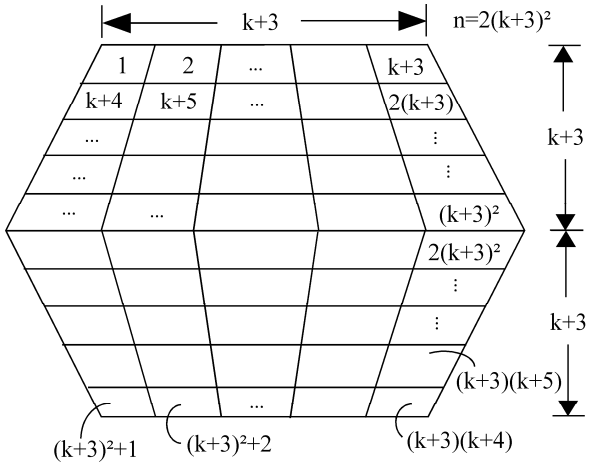
Type 18: T-shape2



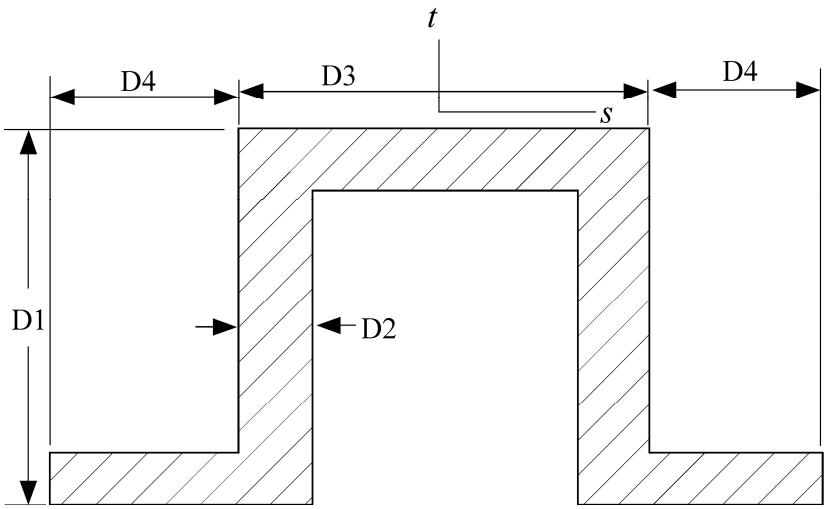


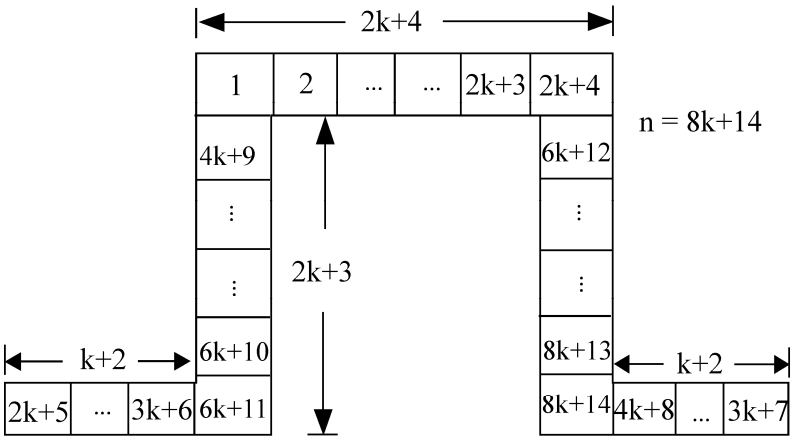
Type 19: Box-shape1



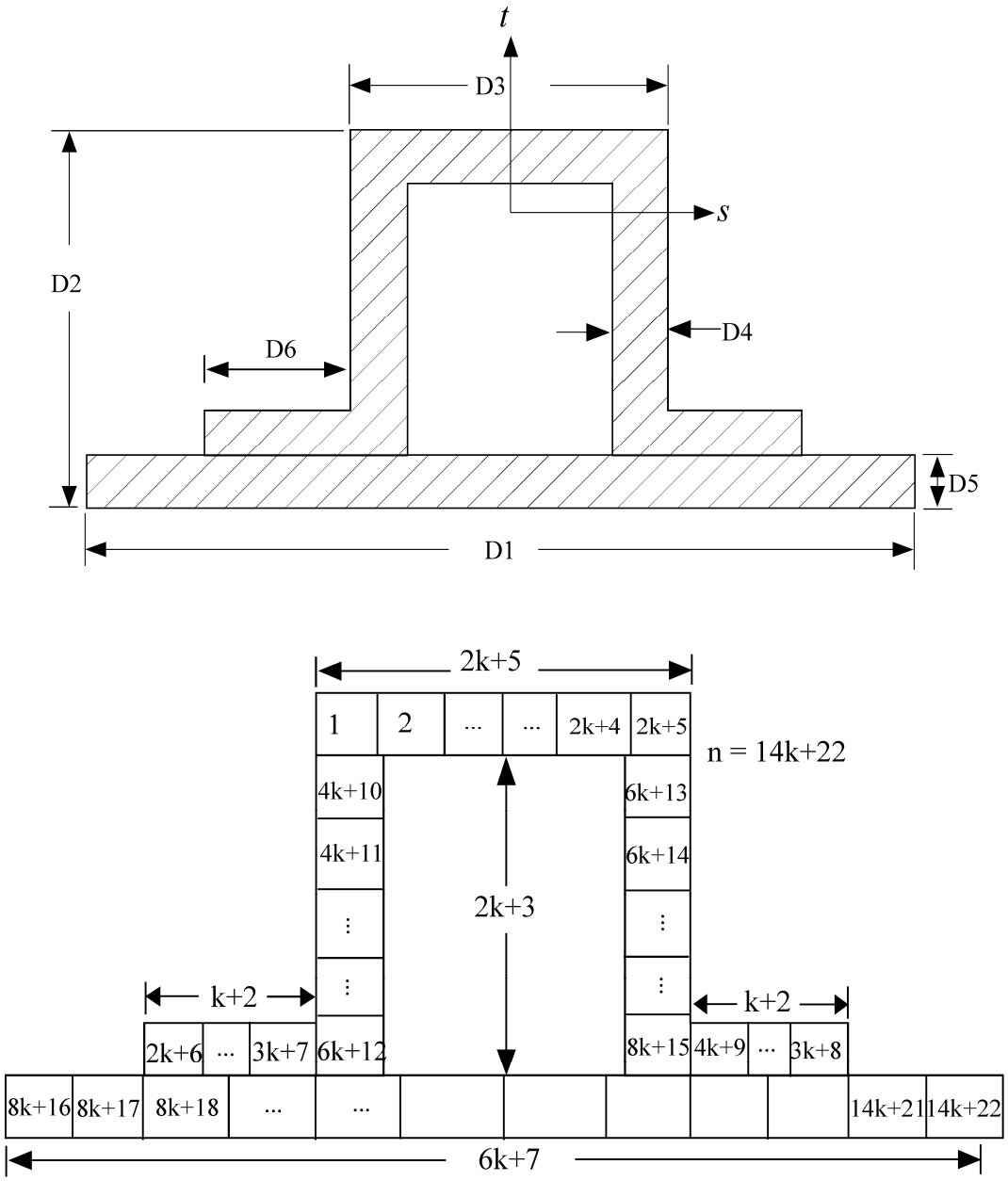


Type 20: Hexagon





Type 21: Hat-shape



Type 22: Hat-shape1

Figure 19.3. Standard beam cross sections.

*INTEGRATION_SHELL

Purpose: Define user defined through the thickness integration rules for the shell element. This option applies to three dimensional shell elements with three or four nodes (ELEMENT_SHELL types 1-11 and 16) and to the eight node thick shell (ELEMENT_TSHELL).

Card 1	1	2	3	4	5	6	7	8
Variable	IRID	NIP	ESOP	FAILOPT				
Type	I	I	I	I				

Define NIP cards below if ESOP = 0.

Card	1	2	3	4	5	6	7	8
Variable	S	WF	PID					
Type	F	F	I					

VARIABLE	DESCRIPTION
IRID	Integration rule ID (IRID refers to IRID on *SECTION_SHELL card).
NIP	Number of integration points
ESOP	Equal spacing of integration points option: EQ.0: integration points are defined below, EQ.1: integration points are equally spaced through thickness such that the shell is subdivided into NIP layers of equal thickness.
FAILOPT	Treatment of failure when mixing different constitutive types, which do and do not include failure models, through the shell thickness. For example, consider the case where a linear viscoelastic material model, which does not have a failure option, is mixed with a composite model, which does have a failure option. Note: If the failure option includes failure based on the time step size of the element, element deletion will occur regardless of the value of FAILOPT. EQ.0: Element is deleted when the layers which include failure, fail. EQ.1: Element failure cannot occur since some layers do not have a failure option.

VARIABLE	DESCRIPTION
S	Coordinate of integration point in range -1 to 1.
WF	Weighting factor. This is typically the thickness associated with the integration point divided by actual shell thickness, i.e., the weighting factor for the ith integration point = $\frac{\Delta t_i}{t}$ as seen in Figure 19.4.
PID	Optional part ID if different from the PID specified on the element card. The average mass density for the shell element is based on a weighted average of the density of each layer that is used through the thickness. When modifying the constitutive constants through the thickness, it is often necessary to defined unique part IDs without elements that are referenced only by the user integration rule. These additional part IDs only provide a density and constitutive constants with local material axes (if used) and orientation angles taken from the PID referenced on the element card. In defining a PID for an integration point, it is okay to reference a solid element PID. The material type through the thickness can change as long as the material type is not hyperelastic (rubber).

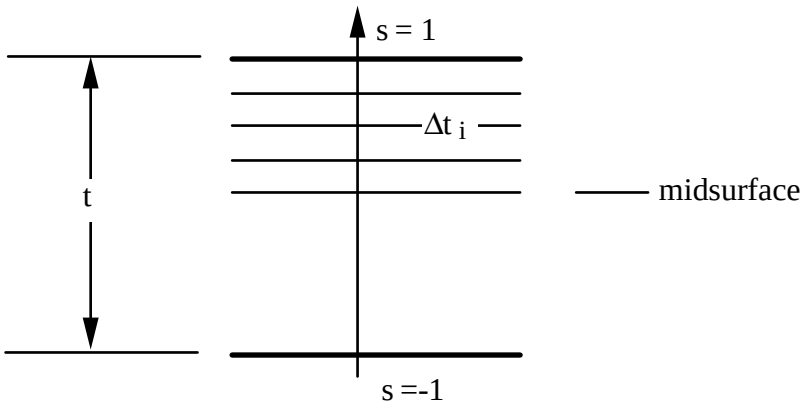


Figure 19.4. In the user defined shell integration rule the ordering of the integration points is arbitrary.

*INTERFACE

Interface definitions may be used to define surfaces, nodal lines, and nodal points for which the displacement and velocity time histories are saved at some user specified frequency. This data may then used in subsequent analyses as an interface ID in the *INTERFACE_LINKING_DISCRETE_NODE as master nodes, in *INTERFACE_LINKING_ SEGMENT as master segments and in *INTERFACE_LINKING_EDGE as the master edge for a series of nodes. This capability is especially useful for studying the detailed response of a small member in a large structure. For the first analysis, the member of interest need only be discretized sufficiently that the displacements and velocities on its boundaries are reasonably accurate. After the first analysis is completed, the member can be finely discretized in the region bounded by the interfaces. Finally, the second analysis is performed to obtain highly detailed information in the local region of interest. When beginning the first analysis, specify a name for the interface segment file using the Z=parameter on the LS-DYNA execution line. When starting the second analysis, the name of the interface segment file created in the first run should be specified using the L=parameter on the LS-DYNA command line. Following the above procedure, multiple levels of sub-modeling are easily accommodated. The interface file may contain a multitude of interface definitions so that a single run of a full model can provide enough interface data for many component analyses. The interface feature represents a powerful extension of LS-DYNA's analysis capabilities. The keyword cards for this purpose are:

***INTERFACE_COMPENSATION_NEW**

INTERFACE_COMPONENT_OPTION*

***INTERFACE_JOY**

INTERFACE_LINKING_DISCRETE_NODE_OPTION*

***INTERFACE_LINKING_EDGE**

***INTERFACE_LINKING_SEGMENT**

INTERFACE_SPRINGBACK_OPTION1_OPTION2*

Interface definitions may also be employed to define soil-structure interfaces in earthquake analysis involving non-linear soil-structure interaction where the structure may be non-linear but the soil outside the soil-structure interface is assumed to be linear. Free-field earthquake ground motions are required only at the soil-structure interface for such analysis. The keyword cards for this purpose are:

***INTERFACE_SSI**

***INTERFACE_SSI_AUX**

***INTERFACE_SSI_STATIC**

***INTERFACE_COMPENSATION_NEW**

Purpose: The current method is used to compensate springback behavior. The capabilities of this feature include: (1) To calculate the deviation of the part from its intended design of the stamped part, and automatically compensate the tool to minimize the deviation, (2) the trimming curve can also be modified after the die modification to make sure that the trimming curve follows the modified tool, and (3) automatically detect the undercut problem.

This method is a nonlinear method. If one iteration is not enough to bring down the deviation to less than the acceptable tolerance, it is always advised to use more iterations. Usually, it is found that 2-4 iterations are needed for most of the cases. In addition, this method provides a scale factor, which allows the user to decide the ratio of shape deviation the part is compensated.

Limitation of the current method involves deficiency in handling (eliminating) the undercut problem.

All the target and current geometry must be included by using the keyword: *INCLUDE_COMPENSATION_*OPTION*.

Card	1	2	3	4	5	6	7	8
Variable	METHOD	SL	SF	ELREF	PSIDm	UNDCT		
Type	I	F	F	I	F	F		
Default	6	5.0	0.75	1				

VARIABLE	DESCRIPTION
METHOD	There are seven extrapolation methods. See Remark 1.
SL	The smooth level parameter controls the smoothness of the modified surfaces. A large value makes the surface smoother. The commonly used value is between 5 and 10. If springback is large, the transition region is expected to be large. However, by using a smaller value of SL, the region of transition can be reduced.
SF	This scales how much of the shape deviation is compensated. For example, if 10 mm springback is predicted, and the scale factor is chosen as 0.75, then the compensation in the opposite direction will only be 7.5 mm.

VARIABLE	DESCRIPTION
	Through many parameter studies, it is found that the best scale factor is case dependent. For some cases, a scale factor of 0.75 is best, while for others, larger values are better. Sometimes, the best value can be larger than 1.1. Since it is impossible to choose the best value for each application it is suggested that for a new application, the initial trial is 0.75. If the springback cannot be effectively compensated, more iterations must be used to compensate the remaining shape deviation. For channel with twisting, the scale factor is more important. It was found that a small change of the tool shape might change the twisting mode. If this occurs, using a small value (<0.5) is suggested.
ELREF	EQ.1: special element refinement is used with the tool elements (default) EQ.2: special element refinement is turned off
PSIDm	Define the Part set ID for master parts. It is important to properly choose the parts for the master side. Usually, only one side (master side) of the tool will be chosen as the master side, and the modification of the other side (slave side) depends solely on the change, which occurs in the master side. In this way, the two sides are coupled and a constant gap between the two sides is maintained. If both sides are chosen as master side, the gap between the two sides might change and the gap might become inhomogeneous. The choice of Master side will have effect on the final result for method 7 for three-piece draw. At this time, when the punch and binder are chosen as the master side, the binder region will not be changed. Otherwise, when the die is chosen as Master side the binder will be changed, since the changes extend to the edges of the Master tool.
UNDCT	EQ.0: Default EQ.1: Check and fix undercut

Remarks:

1. After trimming, only a limited part of the tool has direct relationship with the springback of the blank part. The modification of the rigid tool outside the trimming curve has to rely on extrapolation. However, extrapolating is an unstable process, it is easy to have a non-smooth surface. To resolve this problem, seven smoothing algorithms have been proposed. The frequently used methods are: 7, 3, and 6. The others are used occasionally.

   Method 7: If the punch is chosen as the master side, the binder will not be changed. The only change occurring involves inside punch opening. Under this option, the smoothing

factor has little effect. The smoothness of the modified tool depends on the magnitude of the springback and the size of the addendum region.
Advantages: The binder will not be changed.
Disadvantages: The change will be limited inside the addendum region, and the modified surface may not be smooth if the springback magnitude is large and the transition is small. This is a non-linear method, and the iterative method is used.

Method 6: The smoothness and the transition region of the modified surface will depend on the springback magnitude and the smoothing factor. If the springback magnitude is large, the transition region will be increased automatically. On the other hand, the transition region will be smaller if the springback magnitude is small. At the same time, a larger smoothing factor will result in a smaller transition region.
Advantages: The smoothness of the modified surfaces can be controlled. This is a non-linear method and the iterative method is used.
Disadvantages: It is impossible to limit the transition region, and the binder surface could change if the springback is large.

Method 3: Similar to Method 6, however, it is a linear method and no iteration is necessary. The other options may be removed in the future; therefore they will not be discussed.

Method used to Prevent Undercut
When the wall is steep, it is very possible that undercut will happen. Since undercut is not accepted in real die manufacturing, it is necessary to prevent it from happening.

The code can automatically detect undercut and issue a warning message. In addition, it will save all the element information into a file so that the user can easily know which elements may be problematic.

If the undercut is only limited to a few elements, it is possible to fix the problem manually. The code provides one more option to handle undercut problem, i.e. to compensate the springback only in the punch moving direction (by using a negative scale factor). It is known that this method is not the premium method to handle undercut problem, and we are still working on it and try to find a better solution.

INTERFACE_COMPONENT_OPTION*

Available options include:

NODE

SEGMENT

Purpose: Define an interface for linking calculations. This card applies to the first analysis for storing interfaces in the file specified by Z=isf1 on the execution command line. The output interval used to write data to the interface file is controlled by OPIFS on *CONTROL_OUTPUT.

This capability allows the definition of interfaces that isolate critical components. A database is created that records the motion of the interfaces. In later calculations the isolated components can be reanalyzed with arbitrarily refined meshes with the motion of their boundaries specified by the database created by this input. The interfaces defined here become the masters in the tied interface options.

Each definition consists of a set of cards that define the interface. Interfaces may consists of a set of four node segments for moving interfaces of solid elements, a line of nodes for treating interfaces of shells, or a single node for treating beam and spring elements.

Card	1	2	3	4	5	6	7	8
Variable	SID							
Type	I							

VARIABLE	DESCRIPTION
SID	Set ID, see *SET_NODE or *SET_SEGMENT.

*INTERFACE_JOY

Purpose: Define an interface for linking calculations by moving a nodal interface.

Card	1	2	3	4	5	6	7	8
Variable	SID							
Type	I							

VARIABLE	DESCRIPTION
SID	Nodal set ID, see *SET_NODE.

INTERFACE_LINKING_DISCRETE_NODE_*OPTION

Available options include:

NODE

SET

Purpose: Define an interface for linking discrete nodes to an interface file. This link applies to all element types. With this option the nodes, if specified by a node set, must be give in the same order as they appear in the interface file. This restriction is removed by the more recent keyword *INTERFACE_LINKING_NODE_....

Card	1	2	3	4	5	6	7	8
Variable	NID/NSID	IFID						
Type	I	I						

VARIABLE	DESCRIPTION
NID	Node ID or Node set ID to be moved by interface file, see *NODE or *SET_NODE.
IFID	Interface ID in interface file.

*INTERFACE_LINKING_EDGE

Purpose: Define an interface for linking a series of nodes in shell structure to an interface file for the second analysis using L=isf2 on the execution command line. This link applies segments on shell elements only.

Card	1	2	3	4	5	6	7	8
Variable	NSID	IFID						
Type	I	I						

VARIABLE	DESCRIPTION
NSID	Node set ID to be moved by interface file.
IFID	Interface ID in interface file.

INTERFACE_LINKING _NODE_OPTION*

Available options include:

NODE

SET

Purpose: Define an interface for linking nodes to an interface file. This link applies to all element types.

Card	1	2	3	4	5	6	7	8
Variable	NID/NSID	IFID						
Type	I	I						

VARIABLE	DESCRIPTION
NID	Node ID or Node set ID to be moved by interface file, see *NODE or *SET_NODE.
IFID	Interface ID in interface file.

Remarks:

The difference between this keyword and *INTERFACE_LINKING_DISCRETE_... is that the constraint is done to the nearest node via a bucketsort during initialization. Nodes not found are reported and deleted from the list to tie. These nodes are treated just as if they were tied via the .._DISCRETE_NODE option, they are constrained based on their location rather than the input order.

*INTERFACE_LINKING_SEGMENT

Purpose: Define an interface for linking segments to an interface file for the second analysis using L=isf2 on the execution command line. This applies segments on shell and solid elements.

Card	1	2	3	4	5	6	7	8
Variable	SSID	IFID						
Type	I	I						

VARIABLE	DESCRIPTION
SSID	Segment set to be moved by interface file.
IFID	Interface ID in interface file.

INTERFACE_SPRINGBACK_OPTION1_OPTION2*

Available options included for *OPTION1* are:

LSDYNA

NASTRAN

SEAMLESS

and for *OPTION2:*

THICKNESS

NOTHICKNESS

See the remarks below.

Purpose: Define a material subset for an implicit springback calculation in LS-DYNA and any nodal constraints to eliminate rigid body degrees-of-freedom.

Card	1	2	3	4	5	6	7	8
Variable	PSID	NSHV	FTYPE		FTENSR	NTHHSV		
Type	I	I	I		I	I		

VARIABLE	DESCRIPTION
PSID	Part set ID for springback, see ∗SET_PART.
NSHV	Number of shell or solid history variables (beyond the six stresses and effective plastic strain) to be initialized in the interface file. For solids, one additional state variable (initial volume) is also written. If NSHV is nonzero, the element formulations, calculational units, and constitutive models should not change between runs. If NHSV exceeds the number of integration point history variables required by the constitutive model, only the number required is written; therefore, if in doubt, set NHSV to a large number.
FTYPE	File type: EQ.0: ASCII, EQ.1: binary EQ.2: both ASCII and binary. EQ.10: ASCII large format (see *INITIAL_STRESS_SHELL)

VARIABLE	DESCRIPTION
	EQ.11: binary large format EQ.12: both ASCII and binary large format
FTENSR	Flag for dumping tensor data from the element history variables into the dynain file. EQ.0: Don't dump tensor data from element history variables EQ.1: Dump any tensor data from element history variables into the dynain file in GLOBAL coordinate system Currently, only Material 190 supports this option.
NTHHSV	Number of thermal history variables.

Define a list of nodal points that are constrained for the springback. This section is terminated by an "*" indicating the next input section.

Card	1	2	3	4	5	6	7	8
Variable	NID	TC	RC					
Type	I	F	F					

VARIABLE	DESCRIPTION
NID	Node ID, see *NODE.
TC	Translational Constraint: EQ.0: no constraints, EQ.1: constrained x displacement, EQ.2: constrained y displacement, EQ.3: constrained z displacement, EQ.4: constrained x and y displacements, EQ.5: constrained y and z displacements, EQ.6: constrained z and x displacements. EQ.7: constrained x, y, and z displacements.
RC	Rotational constraint: EQ.0: no constraints, EQ.1: constrained x rotation, EQ.2: constrained y rotation, EQ.3: constrained z rotation, EQ.4: constrained x and y rotations, EQ.5: constrained y and z rotations, EQ.6: constrained z and x rotations, EQ.7: constrained x, y, and z rotations.

Remarks:

1. The NOTHICKNESS option is available for LS-DYNA and NASTRAN in which case the shell element thickness is not an output. The file name for LS-DYNA is "dynain" and for NASTRAN is "nastin." The *CONTROL_ADAPTIVITY is available for LS-DYNA.

2. Trimming is available for the adaptive mesh but it requires some steps. To trim an adaptive mesh use the following procedure:

   (1) Generate the file, "dynain" using the keyword *INTERFACE_SPRINGBACK_ LSDYNA.

   (2) Prepare a new input deck including the file "dynain."

   (3) Add the keyword *ELEMENT_TRIM to this new deck.

   (4) Add the keyword *DEFINE_CURVE_TRIM to this new deck.

   (5) Run this new input deck with i=input_file_name. The adaptive constraints are eliminated by remeshing and the trimming is performed.

   (6) In case this new trimmed mesh is needed, run a zero termination time job and output the file generated via the keyword, *INTERFACE_SPRINGBACK_ LSDYNA.

3. Temperature – The file new_temp_ic.inc will be created for a thermal solution and a coupled thermal-mechanical solution. The file new_temp_ic.inc is a KEYWORD include file which contains "new temperature initial conditions" for the nodes belonging to the PSID.

4. For thermal user materials it is possible to dump thermal history variables using NTHHSV.

Remarks for Seamless Springback:

In seamless springback LS-DYNA automatically and seamlessly switches from explicit or implicit dynamic to implicit static mode at the end of a forming simulation, and continues to run the static springback analysis. Seamless springback can be activated in the original LS-DYNA input deck, or later using a small restart input deck. In this way, the user can decide to continue a previous forming analysis by restarting to add the implicit springback phase. (Another alternative approach to springback simulation is to use the keyword *INTERFACE_SPRINGBACK_ LSDYNA to generate a "dynain" file after forming, and then perform a second simulation running LS-DYNA in fully implicit mode for springback. See Appendix P for a description of how to run an implicit analysis using LS-DYNA.

The implicit springback phase begins when the forming simulation termination time ENDTIM is reached, as specified with the keyword *CONTROL_TERMINATION. Since the springback phase is static, its termination time can be chosen arbitrarily (unless material rate

effects are included). The default choice is 2.0*ENDTIM, and can be changed using the *CONTROL_IMPLICIT_GENERAL keyword.

Since the springback analysis is a static simulation, a minimum number of essential boundary conditions or Single Point Constraints (SPC's) can be input to prohibit rigid body motion of the part. These boundary conditions can be added for the springback phase using the input option on the *INTERFACE_SPRINGBACK_SEAMLESS keyword above.

If no boundary conditions are added with the SEAMLESS option an eigenvalue computation is automatically performed using the Inertia Relief Option to find any rigid body modes and then automatically constrain them out of the springback simulation (see *CONTROL_IMPLICIT_INERTIA_RELIEF). This approach introduces no artificial deformation and is recommended for many simulations.

Several new *CONTROL_IMPLICIT keywords have been added to control the implicit springback phase. These keywords can also be added to a restart input deck. Generally, default settings can be used, so these keywords need not be included in the input deck.

To obtain accurate springback solutions, a nonlinear springback analysis must be performed. In many simulations, this iterative equilibrium search will converge without difficulty. If the springback simulation is particularly difficult, either due to nonlinear deformation, nonlinear material response, or numerical precision errors, a multi-step springback simulation will be automatically invoked. In this approach, the springback deformation is divided into several smaller, more manageable steps.

Two specialized features in LS-DYNA are used to perform multi-step springback analyses. The addition and gradual removal of artificial springs is performed by the artificial stabilization feature. Simultaneously, the automatic time step control is used to guide the solution to the termination time as quickly as possible, and to persistently retry steps where the equilibrium search has failed. By default, both of these features are active during a seamless springback simulation. However, the default method attempts to solve the springback problem in a single step. If this is successful, the solution will terminate normally. If the single step springback analysis fails to converge, the step size will be reduced, and artificial stabilization will become active. Defaults for these features can be changed using the following keywords: *CONTROL_IMPLICIT_GENERAL, *CONTROL_IMPLICIT_AUTO, and *CONTROL_IMPLICIT_STABILIZATION.

***INTERFACE_SSI_*{OPTION}*_ID**

Purpose: This card creates a tied-contact soil-structure interface for use in a transient analysis of a soil-structure system subjected to earthquake excitation. This card allows the analysis to start from a static state of the structure, as well as to read in ground motions recorded on the interface in an earlier analysis.

Available options are:

<**BLANK**>

OFFSET

CONSTRAINED_OFFSET

LS-DYNA implements the effective seismic input method [Bielak and Christiano (1984)] for modeling the interaction of a non-linear structure with a linear soil foundation subjected to earthquake excitation. Note that any non-linear portion of the soil near the structure may be incorporated with the structure into a larger generalized structure, but the soil is assumed to behave linearly beyond a certain distance from the structure.

The effective seismic input method couples the dynamic scattered motion in the soil — the difference between the motion in the presence of the structure and the free-field motion in its absence — with the total motion of the structure. This replaces the distant earthquake source with equivalent effective forces adjacent to the soil-structure interface and allows truncation of the large soil domain using a non-reflecting boundary (e.g. *MAT_PML_ELASTIC) to avoid unnecessary computation. These effective forces can be computed using the free-field ground motion at the soil-structure interface, thus avoiding deconvolution of the free-field motion down to depth.

Non-linear behavior of the structure may be modeled by first carrying out a static analysis of the soil-structure system, and then carrying out the transient analysis with only the structure initialized to its static state. Because the transient analysis employs the dynamic scattered motion in the soil, the soil cannot have any static loads only it — only the structure is subjected to static forces. Consequently, the structure must be supported by the static reactions at the soil-structure interface. Additionally, the soil nodes at the interface must be initialized to be compatible with the initial static displacement of the structure. LS-DYNA will do these automatically if the soil-structure interface is identified appropriately in the static analysis and reproduced in the transient analysis.

Thus, soil-structure interaction analysis under earthquake excitation may be carried out in LS-DYNA as follows:

0. Carry out a static analysis of the soil-structure system (e.g. using dynamic relaxation; see *CONTROL_DYNAMIC_RELAXATION), with the soil-structure interface identified using *INTERFACE_SSI_STATIC_ID

   Optionally, carry out a free-field analysis to record free-field motions on a free surface using *INTERFACE_SSI_AUX.

1. Carry out the transient analysis as a full-deck restart job (see *RESTART), with only the structure initialized to its static stress state (see *STRESS_INITIALIZATION), and the same soil-structure interface identified using *INTERFACE_SSI_ID with the same ID as in static analysis:

   a. The structure mesh must be identical to the one used for static analysis.

   b. The soil mesh is expected to be different from the one used for static analysis, especially because non-reflecting boundary models may be used for transient analysis.

   c. The meshes for the structure and the soil need not match at the interface.

   d. Only the structure must be subjected to static loads, via *LOAD_BODY_PARTS

   e. The earthquake ground motion is specified using *LOAD_SEISMIC_SSI, and/or read from motions recorded from a previous analysis using *INTERFACE_SSI_AUX

Card format (i10,a70)

Card 1 1 2-8

Variable	ID	HEADING
Type	I	A70

Card format (2i10)

Card 2 1 2 3 4 5 6 7 8

Variable	STRID	SOILID	SPR	MPR				
Type	I	I	I	I				
Default	none	none	0	0				

Optional card

Card format (i10,3e10.0,i10)

Card 3	1	2	3	4	5	6	7	8
Variable	GMSET	SF	BIRTH	DEATH	MEMGM			
Type	I	F	F	F	I			
Default	none	1.	0.	1.E+28	2500000			

VARIABLE	DESCRIPTION
ID	Soil-structure interface ID. This is required and must be unique amongst all the contact interface IDs in the model.
HEADING	A descriptor for the given ID.
STRID	Segment set ID of base of structure at soil-structure interface.
SOILID	Segment set ID of soil at soil-structure interface.
SPR	Include the slave side in the *DATABASE_NCFORC and the *DATABASE _BINARY_INTFOR interface force files: EQ.1: slave side forces included.
MPR	Include the master side in the *DATABASE_NCFORC and the *DATABASE_BINARY_INTFOR interface force files: EQ.1: master side forces included.
GMSET	Identifier for set of recorded motions from *INTERFACE_SSI_AUX
SF	Recorded motion scale factor. (default=1.0)
BIRTH	Time at which specified recorded motion is activated.
DEATH	Time at which specified recorded motion is removed: EQ.0.0: default set to 10^{28}
MEMGM	Size in words of buffer allocated to read in recorded motions

Remarks:

1. A tied contact interface (*CONTACT_TIED_SURFACE_TO_SURFACE) is created between the structure and the soil using the specified segment sets, with the soil segment set as the master segment set and the structure segment set as the slave. Naturally, the two segment sets should not have merged nodes and can be non-matching in general. However, the area covered by the two surfaces should match.

2. The options OFFSET and CONSTRAINED_OFFSET create the corresponding tied surface-to-surface contact interface.

3. The soil-structure interface ID is assigned as the ID of the generated contact interface.

4. It is assumed that the soil segment set is oriented toward the structure.

5. Multiple soil-structure interfaces are allowed, e.g. for bridge analysis.

6. The recorded motions are read in from a binary file named **gmbin** by default, but a different filename may be chosen using the option GMINP on the command line (see INTRODUCTION, Execution Syntax).

7. If the motions from *INTERFACE_SSI_AUX were recorded on a segment set, then the free-field motions on each node in the master segment set of the soil-structure interface are calculated from the nearest segment of the segment set used to record the motions.

   If however, the motions were recorded on a node set, then the motions on the master segment set nodes is found by interpolation as is done for *LOAD_SEISMIC_SSI.

***INTERFACE_SSI_AUX**_*{OPTION}*

Available options are:

<**BLANK**>

NODE

Purpose: This card records the motion at a free surface, or on a set of nodes on a free surface, for the purpose of using the recorded motion as a free-field motion in a subsequent interaction analysis using *INTERFACE_SSI. By default, this card records motions on a segment set defining a surface, but can record motions on a node set using the option NODE.

Card format (2i10)

Card 1	1	2	3	4	5	6	7	8
Variable	GMSET	SETID						
Type	I	I						
Default	none	none						

VARIABLE	DESCRIPTION
GMSET	Identifier for this set of recorded motions to be referred to in *INTERFACE_SSI. Must be unique.
SETID	Segment set or node set ID where motions are to be recorded.

Remarks:

1. The motions on the specified segment set or node set is recorded in a binary file named **gmbin** by default, but a different filename may be chosen using option GMOUT on the command line (see INTRODUCTION, Execution Syntax).

2. The output interval for the motions may be specified using the parameter GMDT on the *CONTROL_OUTPUT card, with the default value being 1/10-th of the output interval for D3PLOT states.

***INTERFACE_SSI_STATIC_*{OPTION}*_ID**

Purpose: This card creates a tied-contact soil-structure interface in order to record the static reactions at the base of the structure, which are to be used in a subsequent dynamic analysis of the soil-structure system subjected to earthquake excitation. This card is intended to be used with the initial static analysis of the structure subjected to gravity loads.

Available options are:

<**BLANK**>

OFFSET

CONSTRAINED_OFFSET

Card format (i10,a70)

Card 1	1	2-8
Variable	ID	HEADING
Type	I	A70

Card format (2i10)

Card 2	1	2	3	4	5	6	7	8
Variable	STRID	SOILID	SPR	MPR				
Type	I	I	I	I				
Default	none	none	0	0				

VARIABLE	DESCRIPTION
ID	Soil-structure interface ID. This is required and must be unique amongst all the contact interface IDs in the model.
HEADING	A descriptor for the given ID.
STRID	Segment set ID of base of structure at soil-structure interface.

VARIABLE	DESCRIPTION
SOILID	Segment set ID of soil at soil-structure interface.
SPR	Include the slave side in the *DATABASE_NCFORC and the *DATABASE _BINARY_INTFOR interface force files: EQ.1: slave side forces included.
MPR	Include the master side in the *DATABASE_NCFORC and the *DATABASE_BINARY_INTFOR interface force files: EQ.1: master side forces included.

Remarks: See *INTERFACE_SSI_ID. The ID used for a particular interface in the static analysis must also be used for the same interface identified using *INTERFACE_SSI_ID during dynamic analysis.

*KEYWORD

***KEYWORD_**{*OPTION*} {*MEMORY*} {*NCPU=n*}

Available options include:

<BLANK>

ID

Purpose: The keyword, *KEYWORD, flags LS-DYNA that the input deck is a keyword deck rather than the structured format, which has a strictly defined format. This must be the first card in the input file. Alternatively, by typing "keyword" on the execution line, keyword input formats are assumed and this beginning "*KEYWORD" line is not required.

There are 2 optional parameters that can be specified with the *KEYWORD which can be listed in any order. If a number *{MEMORY}* is specified after the word *KEYWORD, it defines the memory size to be used in words. The memory size can also be set on the command line. Note that the memory size specified on the *KEYWORD card is overridden by the memory specified on the execution line. If the parameter {NCPU=n} is specified it defines the number of CPUs "n" to be used during the analysis. This only applies to the Shared Memory Parallel (SMP) version of LS-DYNA. For the Distributed Memory Version (MPP) the number of CPUs is always defined with the "mpirun" command. Defining the number of CPUs on the execution line overrides what is specified with the *KEYWORD command and both override the number of CPUs specified by *CONTROL_PARALLEL.

One optional card is available for *KEYWORD for naming file names with a prefix. This option allows for multiple simulations in a directory, since the prefix is attached to all output and scratch filenames, i.e., not the input filenames.

Optional if the ID option is active.

Card 1 1 2 3 4 5 6 7 8

Variable	PROJECT	NUM	STAGE
Type	A	A	A
Default	none	none	none
Remarks	see below		

*KEYWORD

VARIABLE	DESCRIPTION
PROJECT	First part of the file name prefix.
NUM	Second part of the file name prefix.
STAGE	Third part of the file name prefix.

Remarks:

1. An example, the prefix with the file name given on the optional card for file D3PLOT would be the following:

   PROJECT_NUM_STAGE.D3PLOT

   The prefix can also be assigned by the option, jobid on the execution line. The execution line would look like the following:

   ls971 i=input.k jobid= PROJECT_NUM_STAGE

2. An example of the {MEMORY} and {NCPU=n} options would be the follows:

   *KEYWORD 12000000 NCPU=2

   The *KEYWORD command is requesting 12 million words of memory and 2 CPUs to be used for the analysis.

*LOAD

The keyword *LOAD provides a way of defining applied forces. The keyword control cards in this section are defined in alphabetical order:

***LOAD_ALE_CONVECTION_**{*OPTION*}

LOAD_BEAM_OPTION*

***LOAD_BLAST**

LOAD_BODY_OPTION*

***LOAD_BODY_GENERALIZED**

***LOAD_BODY_POROUS**

***LOAD_BRODE**

***LOAD_DENSITY_DEPTH**

***LOAD_GRAVITY_PART**

***LOAD_HEAT_CONTROLLER**

LOAD_HEAT_GENERATION_OPTION*

***LOAD_MASK**

***LOAD_MOTION_NODE**

***LOAD_MOVING_PRESSURE**

LOAD_NODE_OPTION*

***LOAD_REMOVE_PART**

***LOAD_RIGID_BODY**

***LOAD_SEGMENT_{***OPTION***}**

***LOAD_SEGMENT_NONUNIFORM_{***OPTION***}**

***LOAD_SEGMENT_SET_{***OPTION***}**

***LOAD_SEGMENT_SET_ANGLE**

***LOAD_SEGMENT_SET_NONUNIFORM_{***OPTION***}**

***LOAD_SHELL_**{*OPTION1*}_{*OPTION2*}

***LOAD_SSA**

***LOAD_STIFFEN_PART**

***LOAD_SUPERPLASTIC_FORMING**

LOAD_SURFACE_STRESS_OPTION*

LOAD_THERMAL_OPTION*

***LOAD_THERMAL_CONSTANT**

***LOAD_THERMAL_CONSTANT_ELEMENT**

***LOAD_THERMAL_CONSTANT_NODE**

***LOAD_THERMAL_LOAD_CURVE**

***LOAD_THERMAL_TOPAZ**

***LOAD_THERMAL_VARIABLE**

***LOAD_THERMAL_VARIABLE_ELEMENT**

***LOAD_THERMAL_VARIABLE_NODE**

***LOAD_THERMAL_VARIABLE_SHELL_**{*OPTION*}

***LOAD_VOLUME_LOSS**

***LOAD_ALE_CONVECTION_{***OPTION***}**

Purpose: This card is used to define the convection thermal energy transfer from a hot ALE fluid to the surrounding Lagrangian structure (remark 1). It is associated with a corresponding coupling card defining the interaction between the ALE fluid and the Lagrangian structure. It is only used when thermal energy transfer from the ALE fluid to the surrounding Lagrangian structure is significant. This is designed specifically for airbag deployment application where the heat transfer from the inflator gas to the inflator compartment can significantly affect the inflation potential of the inflator.

Available options include:

<BLANK>

ID

To define an ID number for each convection heat transfer computation in an optional card preceding all other cards for this command. This ID number can be used to output the part temperature and temperature change as functions of time in the *DATABASE_FSI card. To do this, set the CONVID parameter in the *DATABASE_FSI card equal to this ID.

Card	1	2	3	4	5	6	7	8
Variable	LAGPID	LAGT	LAGCP	H	LAGMAS			
Type	I	F	F	F	F			
Default	none	none	none	none	none			
Remarks								

VARIABLE	DESCRIPTION
LAGPID	Lagrangian PID (slave PID) from a corresponding coupling card which receives the thermal energy in the convection heat transfer.
LAGT	Initial temperature of this Lagrangian slave part.
LAGCP	Constant-pressure heat capacity of this Lagrangian slave part. It has a per-mass unit (for example, J/[Kg*K]).

VARIABLE	DESCRIPTION
H	Convection heat transfer coefficient on this Lagrangian slave part surface. It is the amount of energy (J) transferred per unit area, per time, and per temperature difference. For example, its units may be J/[m^2*s*K]
LAGMAS	The mass of the Lagrangian slave part receiving the thermal energy. This is in absolute mass unit (for example, Kg).

Remarks:

1. The only application of this card so far has been for the transfer of thermal energy from the ALE hot inflator gas to the surrounding Lagrangian structure (inflator canister and airbag-containing compartment) in an airbag deployment model.

2. The heat transferred is taken out of the inflator gas thermal energy thus reducing its inflating potential.

3. This is not a precise heat transfer modeling attempt. It is simply one mechanism for taking out excessive energy from the inflating potential of the hot inflator gas.

4. The heat transfer formulation may roughly be represented as following. Some representative units are shown just for clarity.

$$\dot{Q} = H * A * \Delta T \sim Watt \sim \frac{J}{s} \sim \left[\frac{J}{m^2 * s * K} \right] * m^2 * K$$

$$\dot{Q} * dt = \dot{M} * C_p * \left[T_{Lag\_new} - T_{Lag\_orig} \right] \sim \frac{kg}{s} * \left[\frac{J}{kg * K} \right] * K \sim \frac{J}{s}$$

LOAD_BEAM_OPTION*

Available options include:

ELEMENT

SET

Purpose: Apply the distributed traction load along any local axis of beam or a set of beams. The local axes are defined in Figure 22.1, see also *ELEMENT_BEAM.

Card	1	2	3	4	5	6	7	8
Variable	EID/ESID	DAL	LCID	SF				
Type	I	I	I	F				
Default	none	none	none	1.				
Remarks								

VARIABLE	DESCRIPTION
EID/ESID	Beam ID (EID) or beam set ID (ESID), see *ELEMENT_BEAM or *SET_ BEAM.
DAL	Direction of applied load: EQ.1: parallel to r-axis of beam, EQ.2: parallel to s-axis of beam, EQ.3: parallel to t-axis of beam.
LCID	Load curve ID (see *DEFINE_CURVE) or function ID (see *DEFINE_FUNCTION).
SF	Load curve scale factor. This is for a simple modification of the function values of the load curve.

Remarks:

1. The function defined by LCID has 7 arguments: time, the 3 current coordinates, and the 3 reference coordinates. A function that applies a force proportional to the distance from the initial coordinates would be `f(t,x,y,z,x0,y0,z0)= -10.*sqrt ((x-x0)*(x-x0)+(y-y0)*(y-y0)+(z-z0)*(z-z0))`.

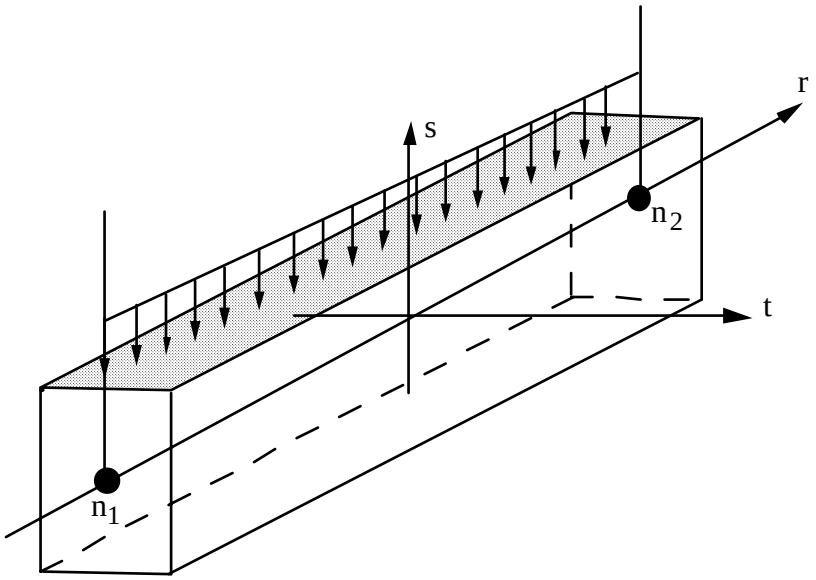


Figure 22.1. Applied traction loads are given in force per unit length. The s and t directions are defined on the *ELEMENT_BEAM keyword.

*LOAD_BLAST

Purpose: Define an airblast function for the application of pressure loads due to explosives in conventional weapons. The implementation is based on a report by Randers-Pehrson and Bannister [1997] where it is mentioned that this model is adequate for use in engineering studies of vehicle responses due to the blast from land mines. This option determines the pressure values when used in conjunction with the keywords: *LOAD_SEGMENT, *LOAD_SEGMENT_SET, or *LOAD_SHELL.

Card 1

Variable	WGT	XBO	YBO	ZBO	TBO	IUNIT	ISURF	
Type	F	F	F	F	F	I	I	
Default	none	0.0	0.0	0.0	0.0	2	2	

Card 2

Variable	CFM	CFL	CFT	CFP				
Type	F	F	F	F				
Default	0.0	0.0	0.0	0.0				

VARIABLE	DESCRIPTION
WGT	Equivalent mass of TNT.
XBO	x-coordinate of point of explosion.
YBO	y-coordinate of point of explosion.
ZBO	z-coordinate of point of explosion.
TBO	Time-zero of explosion.

VARIABLE	DESCRIPTION
IUNIT	Unit conversion flag. EQ.1: feet, pound-mass, seconds, psi EQ.2: meters, kilograms, seconds, Pascals (default) EQ.3: inch, dozens of slugs, seconds, psi EQ.4: centimeters, grams, microseconds, Megabars EQ.5: user conversions will be supplied (see Card 2)
ISURF	Type of burst. EQ.1: surface burst - is located on or very near the ground surface (see Remark 5) EQ.2: air burst - spherical charge (default)
CFM	Conversion factor - pounds per LS-DYNA mass unit.
CFL	Conversion factor - feet per LS-DYNA length units.
CFT	Conversion factor - milliseconds per LS-DYNA time unit.
CFP	Conversion factor - psi per LS-DYNA pressure unit.

Remarks:

1. A minimum of two load curves, even if unreferenced, must be present in the model.

2. Segment normals should point toward the charge.

3. Several methods can be used to approximate the equivalent mass of TNT for a given explosive. The simplest involves scaling the mass by the ratio of the Chapman-Jouguet detonation velocities given the by relationship.

$$M_{TNT} = M \frac{DCJ^2}{DCJ_{TNT}^2}$$

where M_{TNT} is the equivalent TNT mass and DCJ_{TNT} is the Chapman-Jouguet detonation velocity of TNT. M and DCJ are, respectively, the mass and C-J velocity of the explosive under consideration. "Standard" TNT is considered to be cast with a density of 1.57 gm/cm^3 and DCJ_{TNT}=0.693 cm/microsecond.

4. The empirical equations underlying this feature are valid for the scaled distance 0.136 ft/lbm$^{1/3}$ < Z < 100 ft/lbm$^{1/3}$ where Z=R/M$^{1/3}$, R is the distance in feet from the charge center to the target and M is the TNT equivalent mass of the charge in lbm.

5. When a charge is located on or very near the the ground surface it is considered to be a surface burst. Under this circumstance the initial blast wave is immediately reflected and reinforced by the nearly unyielding ground to produce a reflected wave that moves out hemispherically from the point of burst. This reflected wave merged with the initial

incident wave produces overpressures which are greater than those produced by the initial wave alone. In LS-DYNA this wave moves out spherically from the burst point so no distinction of the ground orientation is made. Target points equidistant from the burst point are loaded identically with the surface burst option.

*LOAD_BLAST_ENHANCED

Purpose: Define an airblast function for the application of pressure loads due the explosion of conventional charge. While similar to *LOAD_BLAST this feature includes enhancements for treating reflected waves, moving warheads and multiple blast sources. The loads are applied to facets defined with the keyword *LOAD_BLAST_SEGMENT. A database containing blast pressure history is also available (see *DATABASE_BINARY_BLSTFOR).

Card 1

Variable	BID	M	XBO	YBO	ZBO	TBO	UNIT	BLAST
Type	I	F	F	F	F	F	I	I
Default	none	0.0	0.0	0.0	0.0	0.0	2	2
Remarks		1						

Card 2

Variable	CFM	CFL	CFT	CFP	NIDBO	DEATH	NEGPHS	
Type	F	F	F	F	I	F	I	
Default	0.0	0.0	0.0	0.0	none	1.e+20	0	

VARIABLE	DESCRIPTION
BID	Blast ID. A unique number must be defined for each blast source (charge). Multiple charges may be defined, however, interaction of the waves in air is not considered.
M	Equivalent mass of TNT (see Remark 1).
XBO	x-coordinate of charge center.
YBO	y-coordinate of charge center.
ZBO	z-coordinate of charge center.
TBO	Time of detonation.

VARIABLE	DESCRIPTION
UNIT	Unit conversion flag. EQ.1: feet, pound-mass, seconds, psi EQ.2: meters, kilograms, seconds, Pascals (default) EQ.3: inch, dozens of slugs, seconds, psi EQ.4: centimeters, grams, microseconds, Megabars EQ.5: user conversions will be supplied (see Card 2)
BLAST	Type of blast source. EQ.1: hemispherical surface burst – charge is located on or very near the ground surface (see Remark 5) EQ.2: spherical free-air burst (default) – no amplification of the initial shock wave due to interaction with the ground surface EQ.3: air burst – moving non-sperhical warhead EQ.4: air burst with ground reflection – initial shock wave impinges on the ground surface and is reinforced by the reflected wave to produce a Mach front.
CFM	Conversion factor - pounds per LS-DYNA mass unit.
CFL	Conversion factor - feet per LS-DYNA length units.
CFT	Conversion factor - milliseconds per LS-DYNA time unit.
CFP	Conversion factor - psi per LS-DYNA pressure unit.
NIDBO	Optional node ID representing the charge center. If non-zero then XBO, YBO and XBO are ignored.
NEGPHS	Treament of negative phase. EQ.0: negative dictated by the Friedlander equation. EQ.1: negative phase ignored as in ConWep.
DEATH	Death time. Blast pressures are deactivated at this time.

No further input is required for BLAST=1 or 2. Additional input is required for BLAST=3 and 4.

This Card 3 is mandatory for BLAST=3 (moving non-spherical warhead).

Card 3	1	2	3	4	5	6	7	8
Variable	VEL	TEMP	RATIO	VID				
Type	F	F	F	F				

Default	0.0	70.0	1.0	none				

VARIABLE	DESCRIPTION
VEL	Speed of warhead.
TEMP	Ambient air temperature, Fahrenheit.
RATIO	Aspect ratio of the oblate spheroidal shape fo the blast front. This is the short axis radius divided by the long axis raidus and thus should be a positive value less than unity. Shaped charge and EFP warheads typically have significant lateral blast and its effect can be captured with this option.
VID	Vector representing the symmetry axis of the warhead (see *DEFINE_VECTOR). This vector is parallel to the short axis of the oblate spheroid described above. It is also parallel to the velocity vector when a non-zero velocity VEL is defined.

This Card 3 is mandatory for BLAST=4 (spherical air burst with ground reflection).

Card 3	1	2	3	4	5	6	7	8
Variable	GNID	GVID						
Type	I	I						
Default	none	none						

VARIABLE	DESCRIPTION
GNID	ID of node residing on the ground surface.
GVID	ID of vector representing the vertically upward direction, i.e., normal to the ground surface (see *DEFINE_VECTOR).

Remarks:

1. Several methods can be used to approximate the equivalent mass of TNT for a given explosive. The simplest involves scaling the mass by the ratio of the Chapman-Jouguet detonation velocities given the by relationship.

$$M_{TNT} = M \frac{DCJ^2}{DCJ_{TNT}^2}$$

where M_{TNT} is the equivalent TNT mass and DCJ_{TNT} is the Chapman-Jouguet detonation velocity of TNT. M and DCJ are, respectively, the mass and C-J velocity of the explosive under consideration. "Standard" TNT is considered to be cast with a density of 1.57 gm/cm^3 and DCJ_{TNT}=0.693 cm/microsecond.

2. Segment normals should point toward the charge.

3. The empirical equations underlying this feature are valid for the scaled distance 0.136 ft/lbm$^{1/3}$ < Z < 100 ft/lbm$^{1/3}$ where Z=R/M$^{1/3}$, R is the distance in feet from the charge center to the target and M is the TNT equivalent mass of the charge in lbm. Unexpected results may occur if the scaled distance lies outside this range.

5. When a charge is located on or very near the the ground surface it is considered to be a surface burst. Under this circumstance the initial blast wave is immediately reflected and reinforced by the nearly unyielding ground to produce a reflected wave that moves out hemispherically from the point of burst. This reflected wave merged with the initial incident wave produces overpressures which are greater than those produced by the initial wave alone. In LS-DYNA this wave moves out spherically from the burst point so no distinction of the ground orientation is made. Target points equidistant from the burst point are loaded identically with the surface burst option.

***LOAD_BLAST_SEGMENT**

Purpose: Apply blast pressure loading over one triangular or quadrilateral segment defined by for nodes (see *LOAD_BLAST_ENHANCED).

Card 1	1	2	3	4	5	6	7	8
Variable	BID	N1	N2	N3	N4	ALEPID	SFNRB	
Type	I	I	I	I	I	I	F	
Default	none	none	none	none	none	none	0.	

VARIABLE	DESCRIPTION
BID	Blast source ID (see *LOAD_BLAST_ENHANCED).
N1	Node ID.
N2	Node ID.
N3	Node ID.
N4	Node ID.
ALEPID	Part ID of ALE ambient part underlying this segment to be loaded by this blast (see *PART and *SECTION_SOLID, AET=5).
SFNRB	Scale factor for the ambient element non-reflecting boundary condition. Shocks waves reflected back to the ambient elements can be attenuated with this feature. A value of 1.0 works well for most situations. The feature is disabled when a value of zero is specified.

Remarks:

1. Triangular segments are deifined by repeating the third node.

*LOAD_BLAST_SEGMENT_SET

Purpose: Apply blast pressure loading over each segment in a segment set (see *LOAD_BLAST_ENHANCED).

Card 1	1	2	3	4	5	6	7	8
Variable	BID	SSID	ALEPID	SFNRB				
Type	I	I	I	F				
Default	none	none	none	0.				

VARIABLE	DESCRIPTION
BID	Blast source ID (see *LOAD_BLAST_ENHANCED).
SSID	Segment set ID (see *SET_SEGMENT).
ALEPID	Part ID of ALE ambient part underlying this segment to be loaded by this blast (see *PART and *SECTION_SOLID, AET=5).
SFNRB	Scale factor for the ambient element non-reflecting boundary condition. Shocks waves reflected back to the ambient elements can be attenuated with this feature. A value of 1.0 works well for most situations.

LOAD_BODY_OPTION*

Available options include for base accelerations:

X

Y

Z

for angular velocities:

RX

RY

RZ

and to specify a part set:

PARTS

Purpose: Define body force loads due to a prescribed base acceleration or angular velocity using global axes directions. This data applies to all nodes in the complete problem unless a part subset is specified via the *LOAD_BODY_PARTS keyword. If a part subset is defined then all nodal points belonging to the subset will have body forces applied. The parts specified via the *LOAD_ BODY_PARTS keyword apply to the options X, Y, Z, RX, RY, and RZ above, i.e., different part sets may not apply to different options. Only one part set is expected. **Note: This option applies nodal forces, i.e., it cannot be used to prescribe translational or rotational motion.** Two keyword definitions are needed to apply body loads on a subset of parts: *LOAD_BODY_X and *LOAD_BODY_PARTS.

For options: X, Y, Z, RX, RY, and RZ.

Card	1	2	3	4	5	6	7	8
Variable	LCID	SF	LCIDDR	XC	YC	ZC	CID	
Type	I	F	I	F	F	F	I	
Default	none	1.	0	0.	0.	0.	0	

For option: PARTS.

Card	1	2	3	4	5	6	7	8
Variable	PSID							
Type	I							
Default	none							

VARIABLE	DESCRIPTION
LCID	Load curve ID, see *DEFINE_CURVE.
SF	Load curve scale factor
LCIDDR	Load curve ID for dynamic relaxation phase (optional). This is only needed if dynamic relaxation is defined and a different load curve to LCID is required during the dynamic relaxation phase. Note if LCID is set to zero then no body load will be applied during dynamic relaxation regardless of the value LCIDDR is set to. See *CONTROL_ DYNAMIC_RELAXATION
XC	X-center of rotation, define for angular velocities.
YC	Y-center of rotation, define for angular velocities.
ZC	Z-center of rotation, define for angular velocities.
CID	Coordinate system ID to define acceleration in local coordinate system. The accelerations (LCID) are with respect to CID. EQ.0: global
PSID	Part set ID.

Remarks:

1. Translational base accelerations allow body forces loads to be imposed on a structure. Conceptually, base acceleration may be thought of as accelerating the coordinate system in the direction specified, and, thus, the inertial loads acting on the model are of opposite sign. For example, if a cylinder were fixed to the y-z plan and extended in the positive x-direction, then a positive x-direction base acceleration would tend to shorten the cylinder, i.e., create forces acting in the negative x-direction.

2. Base accelerations are frequently used to impose gravitational loads during dynamic relaxation to initialize the stresses and displacements. During the analysis, in this latter

case, the body forces loads are held constant to simulate gravitational loads. When imposing loads during dynamic relaxation, it is recommended that the load curve slowly ramp up to avoid the excitation of a high frequency response.

3. Body force loads due to the angular velocity about an axis are calculated with respect to the deformed configuration and act radially outward from the axis of rotation. Torsional effects which arise from changes in angular velocity are neglected with this option. The angular velocity is assumed to have the units of radians per unit time.

4. The body force density is given at a point P of the body by:

$$b = \rho(\omega \times \omega \times r)$$

where ρ is the mass density, ω is the angular velocity vector, and r is a position vector from the origin to point P. Although the angular velocity may vary with time, the effects of angular acceleration are not included.

5. Angular velocities are useful for studying transient deformation of spinning three-dimensional objects. Typical applications have included stress initialization during dynamic relaxation where the initial rotational velocities are assigned at the completion of the initialization, and this option ceases to be active.

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$   *LOAD_BODY_Z
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Add gravity such that it acts in the negative Z-direction.
$  Use units of mm/ms2.  Since gravity is constant, the load
$  curve is set as a constant equal to 1.  If the simulation
$  is to exceed 1000 ms, then the load curve needs to be
$  extended.
$
$$$  Note: Positive body load acts in the negative direction.
$
*LOAD_BODY_Z
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$     lcid        sf    lciddr        xc        yc        zc
         5   0.00981
$
$
*DEFINE_CURVE
$     lcid      sidr      scla      sclo      offa      offo
         5
$
$           abscissa            ordinate
                0.00               1.000
             1000.00               1.000
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

LOAD_BODY_GENERALIZED_OPTION*

Available options include:

SET_NODE

SET_PART

Purpose: Define body force loads due to a prescribed base acceleration or a prescribed angular velocity over a subset of the complete problem. The subset is defined by using nodes. Warning: Nodes, which belong to rigid bodies, should not be specified. Rigid bodies must be included within the part sets definitions.

Card 1

Variable	N1/SID	N2/0	LCID	DRLCID	XC	YC	ZC	
Type	I	I	I	I	F	F	F	
Default	none	none	none	0	0.	0.	0.	
Remarks								

Card 2

Variable	AX	AY	AZ	OMX	OMY	OMZ	CID	
Type	F	F	F	F	F	F	I	
Default	0.	0.	0.	0.	0.	0.	0	
Remarks	1, 2	1, 2	1, 2	3, 4, 5	3, 4, 5	3, 4, 5	optional	

VARIABLE	DESCRIPTION
N1/SID	Beginning node ID for body force load or the node or part set ID.
N2	Ending node ID for body force load. Set to zero if a set ID is defined.

VARIABLE	DESCRIPTION
LCID	Load curve ID, see *DEFINE_CURVE.
DRLCID	Load curve ID for dynamic relaxation phase. Only necessary if dynamic relaxation is defined. See *CONTROL_DYNAMIC_RELAXATION.
XC	X-center of rotation. Define only for angular velocity.
YC	Y-center of rotation. Define only for angular velocity.
ZC	Z-center of rotation. Define only for angular velocity.
AX	Scale factor for acceleration in x-direction
AY	Scale factor for acceleration in y-direction
AZ	Scale factor for acceleration in z-direction
OMX	Scale factor for x-angular velocity
OMY	Scale factor for y-angular velocity
OMZ	Scale factor for z-angular velocity
CID	Coordinate system ID to define acceleration in the local coordinate system. The coordinate (XC, YC, ZC) is defined with respect to the local coordinate system if CID is nonzero. The accelerations, LCID and their scale factors are with respect to CID. EQ.0: global

Remarks:

1. Translational base accelerations allow body forces loads to be imposed on a structure. Conceptually, base acceleration may be thought of as accelerating the coordinate system in the direction specified, and, thus, the inertial loads acting on the model are of opposite sign. For example, if a cylinder were fixed to the y-z plane and extended in the positive x-direction, then a positive x-direction base acceleration would tend to shorten the cylinder, i.e., create forces acting in the negative x-direction.

2. Base accelerations are frequently used to impose gravitational loads during dynamic relaxation to initialize the stresses and displacements. During the analysis, in this latter case, the body forces loads are held constant to simulate gravitational loads. When imposing loads during dynamic relaxation, it is recommended that the load curve slowly ramp up to avoid the excitation of a high frequency response.

3. Body force loads due to the angular velocity about an axis are calculated with respect to the deformed configuration and act radially outward from the axis of rotation. Torsional effects which arise from changes in angular velocity are neglected with this option. The angular velocity is assumed to have the units of radians per unit time.

4. The body force density is given at a point P of the body by:

$$b = \rho(\omega \times \omega \times r)$$

where ρ is the mass density, ω is the angular velocity vector, and r is a position vector from the origin to point P. Although the angular velocity may vary with time, the effects of angular acceleration are included.

5. Angular velocities are useful for studying transient deformation of spinning three-dimensional objects. Typical applications have included stress initialization during dynamic relaxation where the initial rotational velocities are assigned at the completion of the initialization, and this option ceases to be active.

*LOAD_BODY_POROUS

Purpose: Define the effects of porosity on the flow with body-force-like loads applied to the ALE element nodes. Ergun porous flow assumptions are used. This only applies to non-deformable (constant-porosity), fully saturated porous media. This model only works with a non-zero and constant viscosity fluid defined via either *MAT_NULL or *MAT_ALE_VISCOUS card.

For options: X, Y, Z, RX, RY, and RZ.

	1	2	3	4	5	6	7	8
Variable	SID	SIDTYP	AX	AY	AZ	BX	BY	BZ
Type	I	I	F	F	F	F	F	F
Default	0	0	0.0	0.0	0.0	0.0	0.0	0.0

VARIABLE	DESCRIPTION
SID	Set ID of the ALE fluid part subjected to porous flow condition.
SIDTYP	Set ID type of the SID above. If SIDTYP=0 (default), then the SID=PSID (part set ID). If SIDTYP=1, then SID=PID (part ID).
AX,AY,AZ	Permeability coefficients for viscous terms in global X, Y and Z directions (please see equation below). If $\{[A_x \neq 0] \& [A_y = 0] \& [A_z = 0]\}$, then an isotropic viscous permeability condition is assumed for the porous medium.
BX,BY,BZ	Passability coefficients for inertia terms in global X, Y and Z directions (please see equation below). If $\{[B_x \neq 0] \& [B_y = 0] \& [B_z = 0]\}$, then an isotropic inertial permeability condition is assumed for the porous medium.

Remarks:

1. Consider the basic general Ergun equation for porous flow in one direction:

$$\frac{\Delta P}{\Delta L} = \frac{\mu}{k_1} V_s + \frac{\rho}{k_2} V_s^2$$

Where

ρ = Fluid density.

μ = Fluid dynamic vicosity.

$V_s = \frac{4Q}{\pi D^2}$ = Superficial fluid velocity.

Q = Overall volume flow rate (m^3/s).

D = Porous channel characteristic width (perpendicular to ΔL).

$k_1 = \frac{\varepsilon^3 d_p{}^2}{150(1-\varepsilon)^2}$ = Viscous parameter

$k_2 = \frac{\varepsilon^3 d_p}{1.75(1-\varepsilon)}$ = Inertial parameter

ε = Porosity = Pore volume / Total media volume
d_p = Particle diameter

2. The above equation can be generalized into 3 dimensional flows where each component may be written as

$$-\frac{dP}{dx_i} = A_i \mu V_i + B_i \rho |V_i| V_i$$

where $i = 1,2,3$ refers to the global coordinate directions (no summation intended for repeated indices), μ is the constant dynamic viscosity, ρ is the fluid density, V_i is the fluid velocity components, A_i is analogous to k_1 above, and B_i is analogous to k_2 above.

3. If $B_i = 0$, the equation is reduced to simple Darcy Law for porous flow (may be good for sand-like flow). For coarse grain (rocks) media, the inertia term will be important and the user needs to input these coefficients.

*LOAD_BRODE

Purpose: Define Brode function for application of pressure loads due to explosion, see Brode [1970], also see *LOAD_SEGMENT, *LOAD_SEGMENT_SET, or *LOAD_SHELL.

Card 1

Variable	YLD	BHT	XBO	YBO	ZBO	TBO	TALC	SFLC
Type	F	F	F	F	F	F	I	I
Default	0.0	0.0	0.0	0.0	0.0	0.0	0	0
Remarks							1	1

Card 2

Variable	CFL	CFT	CFP					
Type	F	F	F					
Default	0.0	0.0	0.0					

VARIABLE	DESCRIPTION
YLD	Yield (Kt, equivalent tons of TNT).
BHT	Height of burst.
XBO	x-coordinates of Brode origin.
YBO	y-coordinates of Brode origin.
ZBO	z-coordinates of Brode origin.
TBO	Time offset of Brode origin.
TALC	Load curve number giving time of arrival versus range relative to Brode origin (space, time), see *DEFINE_CURVE and remark below.

VARIABLE	DESCRIPTION
SFLC	Load curve number giving yield scaling versus scaled time (time relative to Brode origin divided by [yield(**1/3)]) origin (space, time), see *DEFINE_ CURVE and remark below.
CFL	Conversion factor - kft to LS-DYNA length units.
CFT	Conversion factor - milliseconds to LS-DYNA time units.
CFP	Conversion factor - psi to LS-DYNA pressure units.

Remarks:

1. If these curves are defined a variable yield is assumed. Both load curves must be specified for the variable yield option. If this option is used, the shock time of arrival is found from the time of arrival curve. The yield used in the Brode formulas is computed by taking the value from the yield scaling curve at the current time/[yield(**1/3)] and multiplying that value by yield.

*LOAD_DENSITY_DEPTH

Purpose: Define density versus depth for gravity loading. This option has been occasionally used for analyzing underground and submerged structures where the gravitational preload is important. The purpose of this option is to initialize the hydrostatic pressure field at the integration points in the element.

This card should be only defined once in the input deck.

Card	1	2	3	4	5	6	7	8
Variable	PSID	GC	DIR	LCID				
Type	I	F	I	I				
Default	0	0.0	1	none				
Remarks	1,2			3				

VARIABLE	DESCRIPTION
PSID	Part set ID, see *SET_PART. If a PSID of zero is defined then all parts are initialized.
GC	Gravitational acceleration value.
DIR	Direction of loading: EQ.1: global x, EQ.2: global y, EQ.3: global z.
LCID	Load curve ID defining density versus depth, see *DEFINE_CURVE.

Remarks:

1. Density versus depth curves are used to initialize hydrostatic pressure due to gravity acting on an overburden material. The hydrostatic pressure acting at a material point at depth, d, is given by:

$$p = -\int_{d}^{d_{surface}} p(z) g dz$$

where p is pressure, $d_{surface}$, is the depth of the surface of the material to be initialized (usually zero), $\rho(z)$ is the mass density at depth z, and g is the acceleration of gravity. This integral is evaluated for each integration point. Depth may be measured along any of the global coordinate axes, and the sign convention of the global coordinate system should be respected. The sign convention of gravity also follows that of the global coordinate system. For example, if the positive z axis points "up", then gravitational acceleration should be input as a negative number.

2. For this option there is a limit of 12 parts that can be defined by PSID, unless all parts are initialized.

3. Depth is the ordinate of the curve and is input as a descending x, y, or z coordinate value. Density is the abscissa of the curve and must vary (increase) with depth, i.e., an infinite slope is not allowed.

*LOAD_ERODING_PART_SET

Purpose: Apply a pressure load to the exposed surface composed of solid elements that may erode. .

Card	1	2	3	4	5	6	7	8
Variable	ID	LCID	SF	AT	PSID	BOXID	MEM	ALPHA
Type	I	I	F	F	I	I	I	F
Default	none	none	1	0.0	none	0	50	80
Card 2								
Variable	IFLAG	X	Y	Z	BETA			
Type	I	F	F	F	F			
Default	0	0.0	0.0	0.0	90			

VARIABLE	DESCRIPTION
ID	ID number.
LCID	Load curve ID defining pressure as a function of time, see *DEFINE_CURVE.
SF	Scale factor.
AT	Arrival time.
PSID	Part set ID, see *SET_PART.
BOXID	Box ID, see *DEFINE_BOX.
MEM	Extra memory, in percent, to be allocated above the initial memory for storing the new load segments exposed by the erosion.
ALPHA	The maximum angle (in degrees) permitted between the normal of a segment at its centroid and the average normal at its nodes. This angle is used to eliminate interior segments.

IFLAG	Flag for choosing a subset of the exposed surface that is oriented towards a blast or other loading source. The vector from the center of the element to the source location must be within an angle of BETA of the surface normal. If IFLAG>0, then the subset is chosen, otherwise if IFLAG=0, the entire surface is loaded.
X,Y,Z	Optional source location.
BETA	Maximum permitted angle (in degrees) between the surface normal and the vector to the source. The exposed segment is not loaded if the calculated angle is greater than BETA.

Remarks:

1. If LCID is input as -1, then the Brode function is used to determine the pressure for the segments, see *LOAD_BRODE.

2. If LCID is input as -2, then the ConWep function is used to determine the pressure for the segments, see *LOAD_BLAST.

3. The load curve multipliers may be used to increase or decrease the pressure. The time value is not scaled.

4. The activation time, AT, is the time during the solution that the pressure begins to act. Until this time, the pressure is ignored. The function value of the load curves will be evaluated at the offset time given by the difference of the solution time and AT i.e., (solution time-AT). Relative displacements that occur prior to reaching AT are ignored. Only relative displacements that occur after AT are prescribed.

*LOAD_GRAVITY_PART

Purpose: Define gravity for individual parts. This feature is intended for use with *LOAD_STIFFEN_PART to simulate staged construction. Available for solids and shells, and also beam element types 1, 2, 6, and 9.

Note: This keyword card will be available starting in release 3 of version 971.

Card 1

Variable	PID	DOF	LC	ACCEL	LCDR	STGA	STGR	
Type	I	I	I	F	I	I	I	
Default	none	none	none	0	none	0	0	

VARIABLE	DESCRIPTION
PID	Part ID for application of gravity load
DOF	Direction: enter 1, 2 or 3 for X, Y or Z
LC	Load curve defining factor vs. time (or zero if STGA, STGR are defined)
ACCEL	Acceleration (will be multiplied by factor from curve
LCDR	Load curve defining factor vs. time during dynamic relaxation
STGA	Construction stage at which part is added (optional)
STGR	Construction stage at which part is removed (optional)

Remarks:

There are 3 options for defining how the gravity load on a part varies with time.

1. Curve LC gives factor vs time. This overrides the other methods if LC is non-zero.

2. STGA, STGR refer to stages at which part is added and removed – the stages are defined in *DEFINE_CONSTRUCTION_STAGES. If STGA is zero, the gravity load starts at time zero. If not, it ramps up from the small factor FACT (on *CONTROL_STAGED_CONSTRUCTION) up to full value over the ramp time

at the start of stage STGA. If STGR is zero, the gravity load continues until the end of the analysis. If not, it ramps down from full value to FACT over the ramp time at the start of stage STGR.

3. *DEFINE_STAGED_CONSTRUCTION_PART can be used instead of *LOAD_GRAVITY_PART to define this loading. During initialization, a LOAD_GRAVITY_PART card will be created and the effect is the same as using the STGA, STGR method described above; ACCEL is then taken from *CONTROL_STAGED_CONSTRUCTION.

*LOAD_HEAT_CONTROLLER

Purpose: Used to define a thermostat control function. The thermostat controls the heat generation within a material by monitoring a remote nodal temperature. Control can be specified as on-off, proportional, integral, or proportional + integral.

Card 1	1	2	3	4	5	6	7	8
Variable	NODE	PID	LOAD	TSET	TYPE	GP	GI	
Type	I	I	F	F	I	F	F	
Default	none	none	none	none	none	none	none	

VARIABLE	DESCRIPTION
NODE	Sensor is located at this node number.
PID	Part ID assigned to the elements modeling the heater or cooler being controlled.
LOAD	Heater output (q_0). [typical units: W/m^3]
TSET	Controller set point temperature at the location identified by NODE.
TYPE	Type of control function. EQ.1: on-off EQ.2: proportional + integral
GP	Proportional gain.
GI	Integral gain.

Remarks:

The thermostat control function is

$$\dot{q}''' = \dot{q}_0''' + G_P\left(T_{set} - T_{node}\right) + G_I \int_{t=0}^{t} \left(T_{set} - T_{node}\right) dt$$

LOAD_HEAT_GENERATION_*OPTION

Available options include:

SET

SOLID

Purpose: Define solid elements or solid element set with heat generation.

Card	1	2	3	4	5	6	7	8
Variable	SID	LCID	CMULT	WBLCID	CBLCID	TBLCID		
Type	I	I	F	I	I	I		
Default	none	none	0.	0	0	0		

VARIABLE	DESCRIPTION
SID	Solid element set ID or solid element ID, see *SET_SOLID or *ELEMENT_SOLID, respectively.
LCID	Load curve ID for volumetric heat generation rate, $\dot{q}'''$: GT.0: function versus time, EQ.0: use multiplier value CMULT only, LT.0: function versus temperature.
CMULT	Curve multiplier for $\dot{q}'''$. Depending on the definition of LCID this value is either used for scaling or for constant heat generation.
WBLCID	Load curve ID defining the blood perfusion rate [e.g., kg/m^3 sec] as a function of time.
CBLCID	Load curve ID defining the blood heat capacity [e.g., J/kg C] as a function of the blood temperature.
TBLCID	Load curve ID defining the blood temperature [e.g., C] as a function of time.

Remarks:

1. Heat Generation can be defined by:

   a) LCID – load curve id

b) FID – function id

c) a constant baseline value of CMULT

2. Rate of heat transfer from blood to tissue = $W_b C_b \left(T_b - T\right)$ [units: J/m^3 sec]

*LOAD_MASK

Purpose: Apply a distributed pressure load over a three-dimensional shell part. The pressure is applied to a subset of elements that are within a fixed global box and lie either outside or inside of a closed curve in space which is projected onto the surface.

Card	1	2	3	4	5	6	7	8
Variable	PID	LCID	VID1	OFF	BOXID	LCIDM	VID2	INOUT
Type	I	I	F	F	I	I	I	I
Default	none	none	1.	0.	0	0	none	0
Remarks	1		2					

Card

Variable	ICYCLE							
Type	I							
Default	200							
Remarks								

VARIABLE	DESCRIPTION
PID	Part ID (PID). This part must consist of 3D shell elements. To use this option with solid element the surface of the solid elements must be covered with null shells. See *MAT_NULL.
LCID	Curve ID defining the pressure time history, see *DEFINE_CURVE.
VID1	Vector ID normal to the surface on which the applied pressure acts. Positive pressure acts in a direction that is in the opposite direction. This vector may be used if the surface on which the pressure acts is relatively flat. If zero, the pressure load depends on the orientation of the shell elements as shown in Figure 22.3.

VARIABLE	DESCRIPTION
OFF	Pressure loads will be discontinued if $\lvert VID1 \cdot n_{shell} \rvert < OFF$ where n_{shell} is the normal vector to the shell element.
BOXID	Only elements inside the box with part ID, PID, are considered. If no ID is given all elements of part ID, PID, are included. When the active list of elements are updated, elements outside the box will no longer have pressure applied, i.e., the current configuration is always used.
LCIDM	Curve ID defining the mask. This curve gives (x,y) pairs of points in a local coordinate system defined by the vector ID, VID2. Generally, the curve should form a closed loop, i.e., the first point is identical to the last point, and the curve should be flagged as a DATTYP=1 curve in the *DEFINE_CURVE section. If no curve ID is given, all elements of part ID, PID, are included with the exception of those deleted by the box. The mask works like the trimming option, i.e., see DEFINE_CURVE_TRIM and Figure 11.5.
VID2	Vector ID used to project the masking curve onto the surface of part ID, PID. The origin of this vector determines the origin of the local system that the coordinates of the PID are transformed into prior to determining the pressure distribution in the local system. This curve must be defined if LCIDM is nonzero. See Figure 11.5.
INOUT	If 0, elements whose center falls inside the projected curve are considered. If 1, elements whose center falls outside the projected curve are considered.
ICYCLE	Number of time steps between updating the list of active elements (default=200). The list update can be quite expensive and should be done at a reasonable interval. The default is not be appropriate for all problems.

Remarks:

1. The part ID must consist of 3D shell elements.

*LOAD_MOTION_NODE

Purpose: Apply a concentrated nodal force or moment to a node based on the motion of another node.

Card	1	2	3	4	5	6	7	8
Variable	NODE1	DOF1	LCID	SF	CID1	NODE2	DOF2	CID2
Type	I	I	I	F	I	I	I	I
Default	none	none	none	1.	0	0	0	0
Remarks					1			1

VARIABLE	DESCRIPTION
NODE1	Node ID for the concentrated force.
DOF1	Applicable degrees-of-freedom: EQ.1: x-direction of load action, EQ.2: y-direction of load action, EQ.3: z-direction of load action, EQ.4: moment about the x-axis, EQ.5: moment about the y-axis, EQ.6: moment about the z-axis.
LCID	Load curve ID (see *DEFINE_CURVE) or function ID (see *DEFINE_FUNCTION). The applied force is a function of the applicable degree-of-freedom of NODE2.
SF	Load curve scale factor.
CID1	Coordinate system ID (optional), see remark 1 on next page.
NODE2	Node ID for calculating the force.
DOF2	Applicable degrees-of-freedom: EQ. 1: x-coordinate EQ. 2: y-coordinate, EQ. 3: z-coordinate, EQ. 4: x-translational displacement, EQ. 5: y-translational displacement,

VARIABLE	DESCRIPTION
	EQ. 6: z-translational displacement, EQ. 7: rotational displacement about the x-axis, EQ. 8: rotational displacement about the y-axis, EQ. 9: rotational displacement about the z-axis. EQ.10: x-translational velocity, EQ.11: y-translational velocity, EQ.12: z-translational velocity, EQ.13: rotational velocity about the x-axis, EQ.14: rotational velocity about the y-axis, EQ.15: rotational velocity about the z-axis.
CID2	Coordinate system ID (optional), see Remark 1.

Remarks:

1. The global coordinate system is the default. The local coordinate system ID's are defined in the *DEFINE_COORDINATE_SYSTEM section.

*LOAD_MOVING_PRESSURE

Purpose: Apply moving pressure loads to a nondisjoint surface. The pressure loads approximate a jet of high velocity fluid impinging on the surface. Multiple surfaces may be defined each acted on by a set of nozzles.

Card 1	1	2	3	4	5	6	7	8
Variable	LOADID							
Type	I							
Default	none							

Define the following cards for each nozzle. Include one card for each nozzle

Cards 2,..,n	1	2	3	4	5	6	7	8
Variable	NODE1	NODE2	LCID	CUTOFF	LCIDT	LCIDD		
Type	I	I	I	F	I	I		
Default	none	none	none	none	0	0		

The following card defines the surface where the nozzles act.

Card n+1	1	2	3	4	5	6	7	8
Variable	ID	IDTYPE	NIP					
Type	I	I	I					
Default	none	none	3x3					

VARIABLE	DESCRIPTION
LOADID	Loading ID.
NODE1	Node located at the origin of the nozzle.
NODE2	Node located at the head of the nozzle
LCID	Load curve or function (see *DEFINE_FUNCTION) ID defining pressure versus radial distance from the center of the jet.
CUTOFF	Outer radius of jet. The pressure acting outside this radius is set to zero.
LCIDT	Load curve or function (see *DEFINE_FUNCTION) ID, which scales the pressure as a function of time. If a load curve isn't specified, the scale factor defaults to 1.0.
LCIDD	Load curve or function (see *DEFINE_FUNCTION) ID, which scales the pressure as a function of distance from the nozzle. If a load curve isn't specified, the scale factor defaults to 1.0.
ID	Segment set ID, shell element set ID, part set ID, or part ID. See IDT below.
IDT	Slave segment or node set type. The type must correlate with the number specified for SSID: EQ.0: segment set ID for surface-to-surface contact, EQ.1: shell element set ID for surface-to-surface contact, EQ.2: part set ID, EQ.3: part ID,
NIP	Number of integration in segment used to compute pressure loads.

LOAD_NODE_OPTION*

Available options include:

POINT

SET

Purpose: Apply a concentrated nodal force to a node or a set of nodes.

Card	1	2	3	4	5	6	7	8
Variable	NODE/NSID	DOF	LCID	SF	CID	M1	M2	M3
Type	I	I	I	F	I	I	I	I
Default	none	none	none	1.	0	0	0	0
Remarks					1	2		

VARIABLE	DESCRIPTION
NODE/NSID	Node ID or nodal set ID (NSID), see *SET_NODE_*OPTION*.
DOF	Applicable degrees-of-freedom: EQ.1: x-direction of load action, EQ.2: y-direction of load action, EQ.3: z-direction of load action, EQ.4: follower force, see remark 2 on next page, EQ.5: moment about the x-axis, EQ.6: moment about the y-axis, EQ.7: moment about the z-axis. EQ.8: follower moment
LCID	Load curve ID (see *DEFINE_CURVE) or function ID (see *DEFINE_FUNCTION).
SF	Load curve scale factor.
CID	Coordinate system ID (optional), see remark 1 on next page.

VARIABLE	DESCRIPTION
M1	Node 1 ID. Only necessary if DOF.EQ.4 or 8, see remark 2 below.
M2	Node 2 ID. Only necessary if DOF.EQ.4 or 8, see remark 2 below.
M3	Node 3 ID. Only necessary if DOF.EQ.4 or 8, see remark 2 below.

Remarks:

1. The global coordinate system is the default. The local coordinate system ID's are defined in the *DEFINE_COORDINATE_SYSTEM section.

2. Nodes M_1, M_2, M_3 must be defined for a follower force. A positive follower force acts normal to the plane defined by these nodes, and a positive follower moment puts a counterclockwise torque about the t-axis. These actions are depicted in Figure 22.2.

3. For shell formulations 14 and 15, the axisymmetric solid elements with area and volume weighting, respectively, the specified nodal load is per unit length (type14) and per radian (type 15).

4. The function defined by LCID has 7 arguments: time, the 3 current coordinates, and the 3 reference coordinates. A function that applies a force proportional to the distance from the initial coordinates would be `f(t,x,y,z,x0,y0,z0)= -10.*sqrt ((x-x0)*(x-x0)+(y-y0)*(y-y0)+(z-z0)*(z-z0))`.

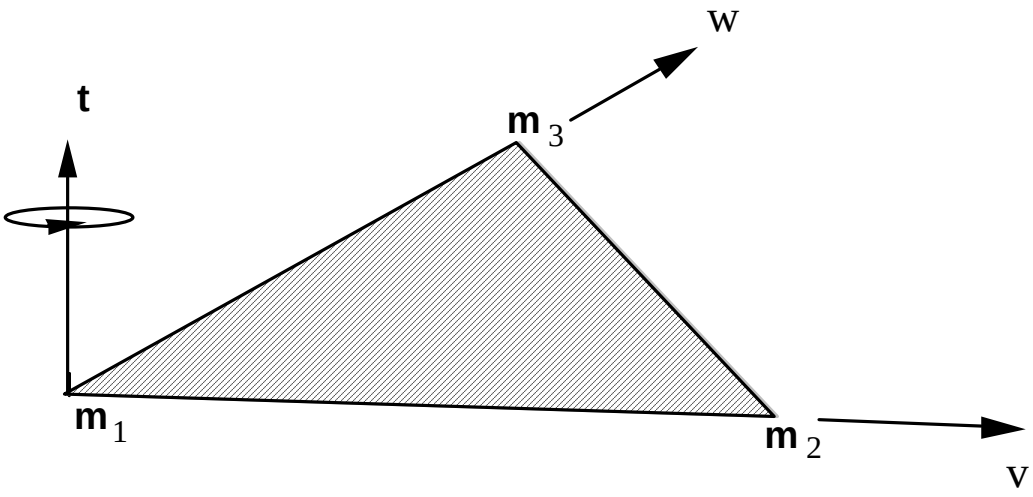


Figure 22.2. Follower force and moment acting on a plane defined by nodes m_1, m_2, and m_3. In this case, the load is applied to node m_1; i.e., m=m_1. A positive force acts in the positive t-direction, and a positive moment puts a counterclockwise torque about the normal vector. The positive t-direction is found by the cross product $t = v \times w$ where v and w are vectors as shown.

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$   *LOAD_NODE_SET
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  A cantilever beam (made from shells) is loaded on the two end nodes
$   (nodes 21 & 22).  The load is applied in the y-direction (dof=2).
$  Load curve number 1 defines the load, but is scaled by sf=0.5 in the
$  *LOAD_NODE_SET definition.
$
*LOAD_NODE_SET
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$     nsid       dof      lcid        sf       cid        m1        m2        m3
        14         2         1       0.5
$
$
*SET_NODE_LIST
$      sid
        14
$
$     nid1      nid2      nid3      nid4      nid5      nid6      nid7      nid8
        21        22
$
$
*DEFINE_CURVE
$     lcid      sidr      scla      sclo      offa      offo
         1
$
$           abscissa           ordinate
                 0.0                0.0
                10.0              100.0
                20.0                0.0
$
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

*LOAD_REMOVE_PART

Purpose: Delete the elements of a part in a staged construction simulation. Shock effects are prevented by gradually reducing the stresses prior to deletion. Available only for solid and shell elements.

Note: This keyword card will be available starting in release 3 of version 971.

Card	1	2	3	4	5	6	7	8
Variable	PID	TIME0	TIME1	STGR				
Type	I	F	F	I				
Default	none	0	0	0				

VARIABLE	DESCRIPTION
PID	Part ID for deletion
TIME0	Time at which stress reduction starts
TIME1	Time at which stresses become zero and elements are deleted
STGR	Construction stage at which part is removed (optional)

Remarks:

There are 3 methods of defining the part removal time:

1. TIME0, TIME1 override all the other methods if non-zero

2. STGR refers to the stage at which the part is removed – the stages are defined in *DEFINE_CONSTRUCTION_STAGES. This is equivalent to setting TIME0 and TIME1 equal to the start and end of the ramp time at the beginning of stage STGR.

3. *DEFINE_STAGED_CONSTRUCTION_PART can be used instead of *LOAD_REMOVE_PART to define this loading. During initialization, a STIFFEN_PART card will be created and the effect is the same as using the STGA, STGR method described above.

*LOAD_RIGID_BODY

Purpose: Apply a concentrated nodal force to a rigid body. The force is applied at the center of mass or a moment is applied around a global axis. As an option, local axes can be defined for force or moment directions.

Card	1	2	3	4	5	6	7	8
Variable	PID	DOF	LCID	SF	CID	M1	M2	M3
Type	I	I	I	F	I	I	I	I
Default	none	none	none	1.	0	0	0	0
Remark					1	2		

VARIABLE	DESCRIPTION
PID	Part ID of the rigid body, see *PART_*OPTION*.
DOF	Applicable degrees-of-freedom: EQ.1: x-direction of load action, EQ.2: y-direction of load action, EQ.3: z-direction of load action, EQ.4: follower force, see Remark 2, EQ.5: moment about the x-axis, EQ.6: moment about the y-axis, EQ.7: moment about the z-axis. EQ.8: follower moment, see Remark 2.
LCID	Load curve ID (see *DEFINE_CURVE) or function ID (see *DEFINE_FUNCTION). GT.0: force as a function of time, LT.0: force as a function of the absolute value of the rigid body displacement. This option only applies to load curves.
SF	Load curve scale factor
CID	Coordinate system ID
M1	Node 1 ID. Only necessary if DOF.EQ.4 or 8, see Remark 2.
M2	Node 2 ID. Only necessary if DOF.EQ.4 or 8, see Remark 2.

M3 Node 3 ID. Only necessary if DOF.EQ.4 or 8, see Remark 2.

Remarks:

1. The global coordinate system is the default. The local coordinate system ID's are defined in the *DEFINE_COORDINATE_SYSTEM section. This local axis is fixed in inertial space, i.e., it does not move with the rigid body.

2. Nodes M_1, M_2, M_3 must be defined for a follower force or moment. The follower force acts normal to the plane defined by these nodes as depicted in Figure 22.2. The positive t-direction is found by the cross product $t = v \times w$ where v and w are vectors as shown. The follower force is applied at the center of mass. A positive follower moment puts a counterclockwise torque about the t-axis.

3. When LCID defines a function, the function has seven arguments: time, the 3 current coordinates for the center of mass, and the 3 reference coordinates. A function that applies a force proportional to the distance from the initial coordinates would be `f(t,x,y,z,x0,y0,z0)= -10.*sqrt ((x-x0)*(x-x0)+(y-y0)*(y-y0)+(z-z0)*(z-z0)).`

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$  *LOAD_RIGID_BODY
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  From a sheet metal forming example.  A blank is hit by a punch, a binder is
$  used to hold the blank on its sides.  The rigid holder (part 27) is held
$  against the blank using a load applied to the cg of the holder.
$
$  The direction of the load is in the y-direction (dof=2) but is scaled
$  by sf = -1 so that the load is in the correct direction.  The load
$  is defined by load curve 12.
$
*LOAD_RIGID_BODY
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$      pid       dof      lcid        sf       cid        m1        m2        m3
        27         2        12      -1.0
$
$
*DEFINE_CURVE
$     lcid      sidr      scla      sclo      offa      offo
        12
$
$           abscissa            ordinate
           0.000E+00           8.000E-05
           1.000E+04           8.000E-05
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

***LOAD_SEGMENT_{*OPTION*}**

To define an ID for the segment loading the following option is available:

ID

If the ID is defined an additional card is required.

Purpose: Apply the distributed pressure load over one triangular or quadrilateral segment defined by four, six, or eight nodes. The pressure and node numbering convention follows Figure 22.3.

ID Card - Required if the option _ID is active on the keyword card.

Optional 1 2-8

Variable	ID	HEADING
Type	I	A70

Card 1 1 2 3 4 5 6 7 8

Variable	LCID	SF	AT	N1	N2	N3	N4	N5
Type	I	F	F	I	I	I	I	I
Default	none	1.	0.	none	none	none	none	none
Remarks	1	2	3	4				

Extra card if N5 is not zero

Card	1	2	3	4	5	6	7	8
Variable	N6	N7	N8					
Type	I	I	I					
Default	none	none	none					
Remarks								

VARIABLE	DESCRIPTION
ID	Loading ID
HEADING	A description of the loading.
LCID	Load curve ID (see *DEFINE_CURVE) or function ID (see *DEFINE_FUNCTION).
SF	Load curve scale factor
AT	Arrival time for pressure or birth time of pressure.
N1	Node ID
N2	Node ID
N3	Node ID. Repeat N2 for two-dimensional geometries.
N4	Node ID. Repeat N2 for two-dimensional geometries.
N5	Optional mid-side node ID located between nodes 1 and 2.
N6	Optional mid-side node ID located between nodes 2 and 3.
N7	Optional mid-side node ID located between nodes 3 and 4.
N8	Optional mid-side node ID located between nodes 4 and 1. Do not define for six node quadratic surface segments.

Remarks:

1. If LCID is input as -1, then the Brode function is used to determine the pressure for the segments, see *LOAD_BRODE.

2. If LCID is input as -2, then the ConWep function is used to determine the pressure for the segments, see *LOAD_BLAST.

3. The load curve multipliers may be used to increase or decrease the pressure. The time value is not scaled.

4. The activation time, AT, is the time during the solution that the pressure begins to act. Until this time, the pressure is ignored. The function value of the load curves will be evaluated at the offset time given by the difference of the solution time and AT i.e., (solution time-AT). Relative displacements that occur prior to reaching AT are ignored. Only relative displacements that occur after AT are prescribed.

5. Triangular segments are defined by repeating the third node.

6. The function defined by LCID has 7 arguments: time, the 3 current coordinates, and the 3 reference coordinates. A function that applies a pressure proportional to the distance from the initial coordinates would be `f(t,x,y,z,x0,y0,z0)= sqrt ((x-x0)*(x-x0)+(y-y0)*(y-y0)+(z-z0)*(z-z0))`.

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$   *LOAD_SEGMENT
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  A block of solid elements is pressed down onto a plane as it moves along
$  that plane.  This pressure is defined using the *LOAD_SEGMENT keyword.
$
$  The pressure is applied to the top of the block.  This top is defined
$  by the faces on top of the appropriate solid elements.  The faces are
$  defined with segments.  For example, nodes 97, 106, 107 & 98 define
$  a top face on one of the solids (and thus, one of the faces to apply the
$  pressure too).  This "face" is referred to as a single segment.
$
$  The load is defined with load curve number 1.  The curve starts at zero,
$  ramps to 100 in 0.01 time units and then remains constant.  However,
$  the curve is then scaled by sclo = 2.5.  Thus, raising the load to 250.
$  Note that the load is NOT scaled in the *LOAD_SEGMENT keyword, but
$  could be using the sf variable.
$
*LOAD_SEGMENT
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$     lcid        sf        at        n1        n2        n3        n4
         1      1.00       0.0        97       106       107        98
         1      1.00       0.0       106       115       116       107
         1      1.00       0.0        98       107       108        99
         1      1.00       0.0       107       116       117       108
$
$
*DEFINE_CURVE
$
$     lcid      sidr      scla      sclo      offa      offo
         1         0       0.0       2.5
$
$           abscissa           ordinate
              0.000               0.0
              0.010             100.0
              0.020             100.0
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

***LOAD_SEGMENT_NONUNIFORM_{*OPTION*}**

To define an ID for the non-uniform segment loading the following option is available:

ID

If the ID is defined an additional card is required.

Purpose: Apply a distributed load over one triangular or quadrilateral segment defined by three, four, six, or eight nodes. The loading and node numbering convention follows Figure 22.3.

ID Card - Required if the option _ID is active on the keyword card.

Optional 1 2-8

Variable	ID	HEADING
Type	I	A70

Card 1 1 2 3 4 5 6 7 8

Variable	LCID	SF	AT	DT	CID	V1	V2	V3
Type	I	F	F	F	I	F	F	F
Default	none	1.	0.	1.E+16	0	none	none	none
Remarks	1	2	3	3	4			

Card 2

Variable	N1	N2	N3	N4	N5	N6	N7	N8
Type	I	I	I	I	I	I	I	I
Default	none	none	none	none	none	none	none	none

Card 3	1	2	3	4	5	6	7	8
Variable	P1	P2	P3	P4	P5	P6	P7	P8
Type	F	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	none	none

VARIABLE	DESCRIPTION
ID	Loading ID
HEADING	A description of the loading.
LCID	Load curve ID (see *DEFINE_CURVE) or function ID (see *DEFINE_FUNCTION). The seven arguments for the function are current time minus the birth time, the current *x*, *y*, and *z* coordinates, and the initial *x*, *y*, and *z* coordinates.
SF	Load curve scale factor
AT	Arrival/birth time for the traction load.
DT	Death time for the traction load.
CID	Coordinate system ID
V1,V2,V3	Vector direction cosines referenced to coordinate system CID to define the direction of the traction loading.
N1	Node ID
N2	Node ID
N3	Node ID. Repeat N2 for two-dimensional geometries.
N4	Node ID. Repeat N2 for two-dimensional geometries or repeat N3 for triangular segments.
N5	Optional mid-side node ID located between nodes 1 and 2.
N6	Optional mid-side node ID located between nodes 2 and 3.
N7	Optional mid-side node ID located between nodes 3 and 4.

VARIABLE	DESCRIPTION
N8	Optional mid-side node ID located between nodes 4 and 1. Do not define for six node quadratic surface segments.
P1	Scale factor at node ID, N1.
P2	Scale factor at node ID, N2.
P3	Scale factor at node ID, N3.
P4	Scale factor at node ID, N4.
P5	Scale factor at node ID, N5.
P6	Scale factor at node ID, N6.
P7	Scale factor at node ID, N7.
P8	Scale factor at node ID, N8.

*LOAD_SEGMENT_SET_{*OPTION*}

To define an ID for the segment loading the following option is available:

ID

If the ID is defined an additional card is required.

Purpose: Apply the distributed pressure load over each segment in a segment set. The pressure convention follows Figure 22.3.

ID Card - Required if the option _ID is active on the keyword card.

Optional 1 2-8

Variable	ID	HEADING
Type	I	A70

Card 1 1 2 3 4 5 6 7 8

Variable	SSID	LCID	SF	AT	DT			
Type	I	I	F	F				
Default	none	none	1.	0.				
Remarks		1	2	3				

VARIABLE	DESCRIPTION
SSID	Segment set ID, see *SET_SEGMENT.
LCID	Load curve ID (see *DEFINE_CURVE) or function ID (see *DEFINE_FUNCTION). The seven arguments for the function are current time minus the birth time, the current *x*, *y*, and *z* coordinates, and the initial *x*, *y*, and *z* coordinates.
SF	Load curve scale factor
AT	Arrival time for pressure or birth time of pressure.

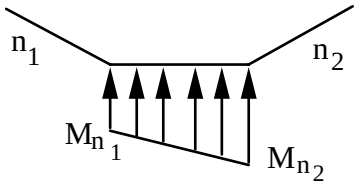
Remarks:

1. If LCID is input as -1, then the Brode function is used to determine pressure for the segment set, also see *LOAD_BRODE.

2. If LCID is input as -2, then the ConWep function is used to determine the pressure for the segments, see *LOAD_BLAST.

3. The load curve multipliers may be used to increase or decrease the pressure. The time value is not scaled.

4. The activation time, AT, is the time during the solution that the pressure begins to act. Until this time, the pressure is ignored. The function value of the load curves will be evaluated at the offset time given by the difference of the solution time and AT i.e., (solution time-AT). Relative displacements that occur prior to reaching AT are ignored. Only relative displacements that occur after AT are prescribed.



2-Dimensional Definition for axisymmetic, plane stress, and plane strain geometries

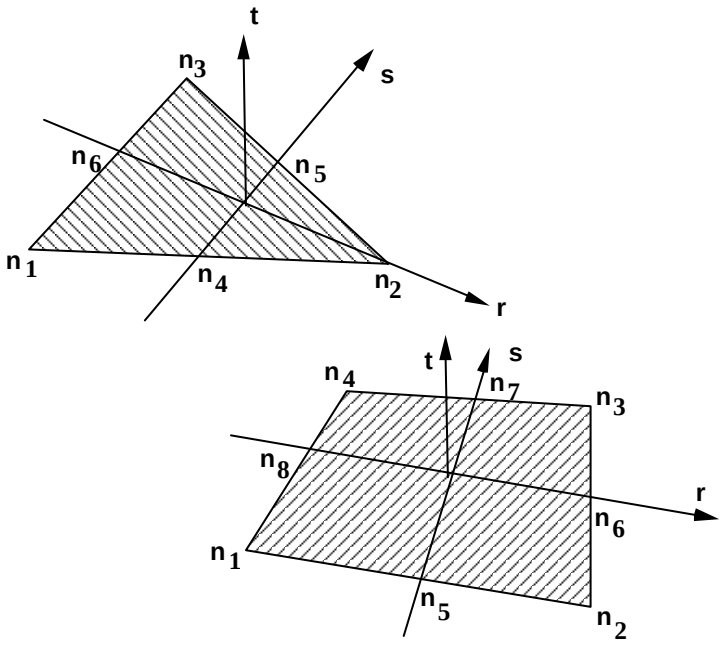


Figure 22.3. Nodal numbering for pressure cards. Positive pressure acts in the negative t-direction. For two-dimensional problems repeat the second node for the third and fourth nodes in the segment definitions.

*LOAD_SEGMENT_SET_ANGLE

Purpose: Apply the traction load over a segment set that is dependent on the orientation of a vector. An example application is applying a pressure to a cylinder as a function of the crank angle in an automobile engine. The pressure and node numbering convention follows Figure 22.3.

Card 1

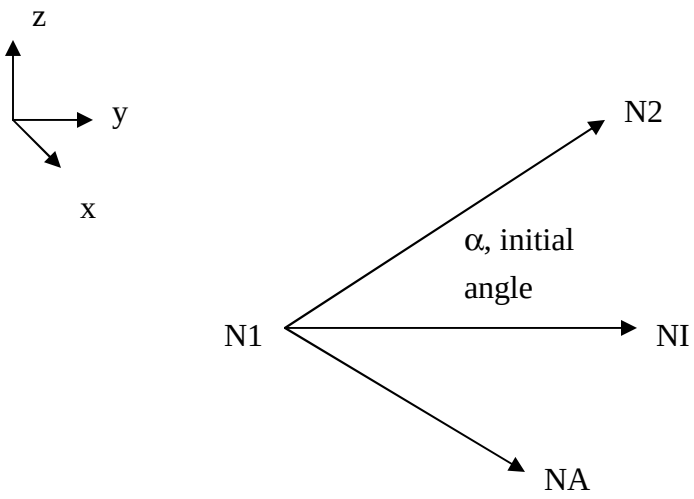
Variable	ID	IDSS	LCID	SCALE	IOPTP	IOPTD		
Type	I	I	I	F	I	I		
Default	none	none	none	1.	0	0		

Card 2

Variable	N1	N2	NA	NI				
Type	I	I	I	I				
Default	none	none	none	none				

VARIABLE	DESCRIPTION
ID	Loading ID
IDSS	Segment set ID.
LCID	Load curve or function ID defining the traction as a function of the angle. If IOPT=0 below, define the abscissa between 0 and 2π radians or 0 and 360 degrees if IOPD=1.
SCALE	Scale factor on value of the load curve or function.
IOPTP	Flag for periodicity. The default (IOPTP=0) requires the load curve to be defined between 0 and 2π. This is useful, for example, for modeling an engine that is running at a steady state since each rotation will experience the same loading. To model a transient response, IOPTP=1 uses a load curve defined over the full range of angles, permitting a different response on the second and subsequent revolutions.

IOPTD	Flag for specifying if the load curve or function argument is in radians (IOPTD=0, the default) or degrees (IOPTD=1).
N1	The node specifying the tail of the rotating vector.
N2	The node specifying the head of the rotating vector.
NA	The node specifying the head of the vector defining the axis of rotation. The node N1 specifies the tail.
NI	The node specifying the orientation of the vector at an angle of zero. If the initial angle is zero, NI should be equal to N2.



*LOAD_SEGMENT_SET_NONUNIFORM_{*OPTION*}

To define an ID for the non-uniform segment loading the following option is available:

ID

If the ID is defined an additional card is required.

Purpose: Apply the traction load over one triangular or quadrilateral segment defined by three or four nodes. The pressure and node numbering convention follows Figure 22.3.

ID Card - Required if the option _ID is active on the keyword card.

Optional 1 2-8

Variable	ID	HEADING
Type	I	A70

Card 1 1 2 3 4 5 6 7 8

Variable	SSID	LCID	SF	AT	DT			
Type	I	I	F	F	F			
Default	none	none	1.	0.	1.E+16			
Remarks	1	1	2	3	3			

Card

Variable	CID	V1	V2	V3				
Type	I	F	F	F				
Default	0	none	none	none				
Remarks								

VARIABLE	DESCRIPTION
ID	Loading ID
HEADING	A description of the loading.
SSID	Segment set ID.
LCID	Load curve ID (see *DEFINE_CURVE) or function ID (see *DEFINE_FUNCTION). The seven arguments for the function are current time minus the birth time, the current *x*, *y*, and *z* coordinates, and the initial *x*, *y*, and *z* coordinates.
SF	Load curve scale factor
AT	Arrival/birth time for pressure.
DT	Death time for pressure.
CID	Coordinate system ID
V1,V2,V3	Vector direction cosines referenced to coordinate system CID to define the direction of the traction loading.

***LOAD_SEISMIC_SSI**_*OPTION1*_{*OPTION2*}

Available options for *OPTION1* include:

NODE

SET

POINT

OPTION2 allows an optional ID to be given:

ID

Purpose: Apply earthquake load due to free-field earthquake ground motion at certain locations — defined by either nodes or coordinates — on a soil-structure interface, for use in earthquake soil-structure interaction analysis. The specified motions are used to compute a set of effective forces in the soil elements adjacent to the soil-structure interface, according to the effective seismic input–domain reduction method [Bielak and Christiano (1984)].

The following card is read if and only if the ID option is specified. The second card is required.

Optional 1 2-8

Variable	ID	HEADING
Type	I	A70

Card 1 for *OPTION1*=NODE or SET:

Card 1 1 2 3 4 5 6 7 8

Variable	SSID	*type*ID	GMX	GMY	GMZ			
Type	I	I	I	I	I			
Default	none	none	none	none	none			

Card 1 for *OPTION1*=POINT:

Card format (i8,3e16.0,3i8)

Card 1	1	2	3	4	5	6	7	8	9	10
Variable	SSID	XP		YP		ZP		GMX	GMY	GMZ
Type	I	F		F		F		I	I	I
Default	none	0.		0.		0.		none	none	None

Card 2:

Card format (e10.0,i10,2e10.0,2i10)

Card 2	1	2	3	4	5	6	7	8
Variable	SF	CID	BIRTH	DEATH	ISG	IGM		
Type	F	I	F	F	I	I		
Default	1.	0	0.	1.E+28	0	0		

VARIABLE	DESCRIPTION
ID	Optional ID. This ID does not need to be unique.
HEADING	An optional descriptor for the given ID.
SSID	Soil-structure interface ID.
*type*ID	Node ID (NID in *NODE) or nodal set ID (SID in *SET_NODE).

VARIABLE	DESCRIPTION
XP	x coordinate of ground motion location on soil-structure interface.
YP	y coordinate of ground motion location on soil-structure interface.
ZP	z coordinate of ground motion location on soil-structure interface.
GMX	Acceleration load curve or ground motion ID for motion in the (local) x-direction.
GMY	Acceleration load curve or ground motion ID for motion in the (local) y-direction.
GMZ	Acceleration load curve or ground motion ID for motion in the (local) z-direction.
SF	Ground motion scale factor. (default=1.0)
CID	Coordinate system ID, see *DEFINE_COORDINATE_SYSTEM.
BIRTH	Time at which specified ground motion is activated.
DEATH	Time at which specified ground motion is removed: EQ.0.0: default set to 10^{28}
ISG	Definition of soil-structure interface: EQ.0: SSID is ID for soil-structure interface defined by *INTERFACE_SSI_ID for non-matching mesh between soil and structure. EQ.1: SSID is segment set ID identifying soil-structure interface for merged meshes between soil and structure.
IGM	Specification of ground motions GMX, GMY, GMZ: EQ.0: ground motions are specified as acceleration load curves. See *DEFINE_CURVE EQ.1: Both ground accelerations and velocities specified using *DEFINE_GROUND_MOTION.

Remarks:

1. The ground motion at any node on a soil-structure interface is computed as follows:

   (a) If the node coincides with a location where ground motion is specified, that ground motion is used for that node.

   (b) If the node does not coincide with a location where ground motion is specified, the ground motion at that node along a particular degree-of-freedom is taken as a weighted average of all the ground motions specified on the interface along that

degree-of-freedom, where the weights are inversely proportional to the distance of the node from the ground motion location.

2. Multiple ground motions specified at the same location are added together to obtain the resultant ground motion at that location.

3. Spatially-uniform ground motion may be specified on a soil-structure interface by specifying the ground motion at only one location on that interface. Specifying the ground motion at more than one point on a soil-structure interface results in spatially-varying ground motion on that interface.

LOAD_SHELL_OPTION1*_{*OPTION2*}

Available options for *OPTION1* include:

ELEMENT

SET

Available options for *OPTION2* include:

ID

To define an ID for the segment loading.

If the ID is defined an additional card is required.

Purpose: Apply the distributed pressure load over one shell element or shell element set. The numbering of the shell nodal connectivities must follow the right hand rule with positive pressure acting in the negative t-direction. See Figure 22.3. This option applies to the three-dimensional shell elements only.

ID Card - Required if the option ID is active on the keyword card.

Optional 1 2-8

Variable	ID	HEADING
Type	I	A70

Card 1 1 2 3 4 5 6 7 8

Variable	EID/ESID	LCID	SF	AT				
Type	I	I	F	F				
Default	none	none	1.	0.				
Remarks	1	1	2					

VARIABLE	DESCRIPTION
EID/ESID	Shell ID (SID) or shell set ID (SSID), see *ELEMENT_SHELL or *SET_SHELL.

VARIABLE	DESCRIPTION
LCID	Load curve ID, see *DEFINE_CURVE.
SF	Load curve scale factor
AT	Arrival time for pressure or birth time of pressure.

Remarks:

1. If LCID is input as -1, then the Brode function is used to determine the pressure for the segments, see also *LOAD_BRODE.

2. If LCID is input as -2, then the ConWep function is used to determine the pressure for the segments, see *LOAD_BLAST.

3. The load curve multipliers may be used to increase or decrease the pressure. The time value is not scaled.

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$   *LOAD_SHELL_ELEMENT
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  From a sheet metal forming example.  A blank is hit by a punch, a holder is
$  used to hold the blank on its sides.  All shells on the holder are given a
$  pressure boundary condition to clamp down on the blank.  The pressure
$  follows load curve 3, but is scaled by -1 so that it applies the load in the
$  correct direction.  The load starts at zero, but quickly rises to 5 MPa
$  after 0.001 sec.  (Units of this model are in: ton, mm, s, N, MPa, N-mm)
$
*LOAD_SHELL_ELEMENT
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$      eid      lcid        sf        at
     30001         3 -1.00E+00       0.0
     30002         3 -1.00E+00       0.0
     30003         3 -1.00E+00       0.0
     30004         3 -1.00E+00       0.0
     30005         3 -1.00E+00       0.0
     30006         3 -1.00E+00       0.0
     30007         3 -1.00E+00       0.0
$
$  Note: Just a subset of all the shell elements of the holder is shown above,
$        in practice this list contained 448 shell element id's.
$
$
*DEFINE_CURVE
$     lcid      sidr      scla      sclo      offa      offo
         3
$
$           abscissa           ordinate
              0.000                0.0
              0.001                5.0
              0.150                5.0
$
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

***LOAD_SSA**

Purpose: The Sub-Sea Analysis capability allows a simple way of loading the structure to account for the effects of the primary explosion and the subsequent bubble oscillations.

Define one card.

Card 1	1	2	3	4	5	6	7	8
Variable	VS	DS	REFL	ZB	ZSURF	FPSID	PSID	
Type	F	F	F	F	F	I	I	
Default	none	none	0.	0.	0.	0	0	

Define two cards for each explosive charge. This input is terminated by the next "*" keyword card.

Card 1	1	2	3	4	5	6	7	8
Variable	A	ALPHA	GAMMA	KTHETA	KAPPA			
Type	F	F	F	F	F			
Default	none	none	none	none	none			

Card 2

Variable	XS	YS	ZS	W	TDELY	RAD	CZ	
Type	F	F	F	F	F	F	F	
Default	none	none	none	none	none	none	none	

VARIABLE	DESCRIPTION
VS	Sound speed in fluid
DS	Density of fluid
REFL	Consider reflections from sea floor. EQ.0: off EQ.1: on
ZB	Z coordinate of sea floor if REFL=1, otherwise, not used.
ZSURF	Z coordinate of sea surface
FPSID	Part set ID of parts subject to flood control. Use the *PART_SET_COLUMN option where the parameters A1 and A2 must be defined as follows: Parameter A1: Flooding status: EQ.1.0: Fluid on both sides. EQ.2.0: Fluid outside, air inside. EQ.3.0: Air outside, fluid inside. EQ.4.0: Material or part is ignored. Parameter A2: Tubular outer diameter of beam elements. For shell elements this input must be greater than zero for loading.
PSID	Part IDs of parts defining the wet surface. The elements defining these parts must have their outward normals pointing into the fluid. See Figure 22.4. EQ.0: all parts are included. GT.0: define NPIDS part ID's below.
A	Shock pressure parameter
ALPHA	α, shock pressure parameter
GAMMA	γ, time constant parameter
KTHETA	K_θ, time constant parameter
KAPPA	κ, ratio of specific heat capacities
XS	X coordinate of charge
YS	Y coordinate of charge
ZS	Z coordinate of charge

VARIABLE	DESCRIPTION
W	Weight of charge
TDELY	Time delay before charge detonates
RAD	Charge radius
CZ	Water depth

Remarks:

The pressure history of the primary shockwave at a point in space through which a detonation wave passes is given as**:**

$$P(t) = P_m e^{-\frac{t}{\theta}}$$

where P_m and the time constant θ below are functions of the type and weight W of the explosive charge and the distance Q from the charge.

$$P_{peak} = A\left[\frac{W^{1/3}}{Q}\right]^{\alpha}$$

$$\theta = K_\theta W^{1/3}\left[\frac{W^{1/3}}{Q}\right]^{\gamma}$$

where A, α, γ, and K_θ are constants for the explosive being used.

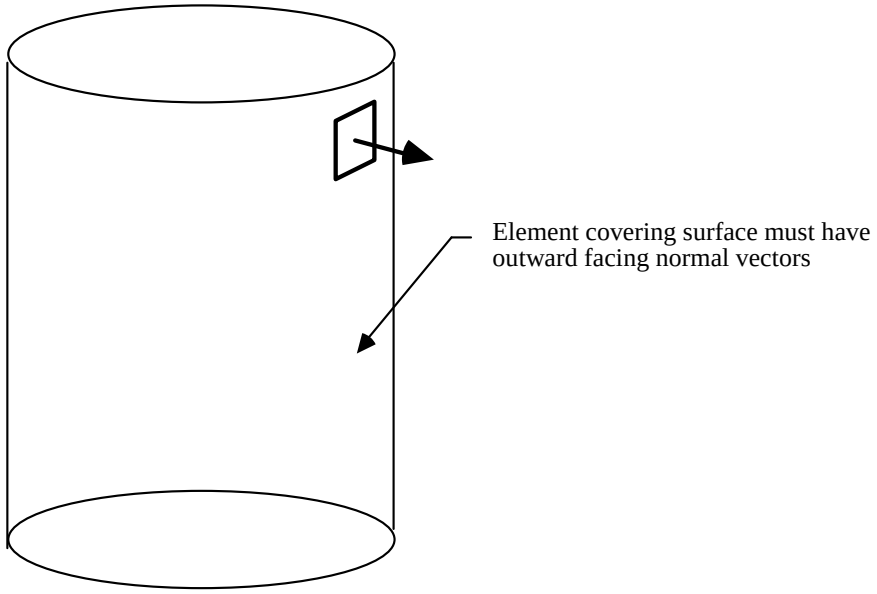


Figure 22.4. The shell elements interacting with the fluid must be numbered such that their outward normal vector points into the fluid media.

*LOAD_STEADY_STATE_ROLLING

Steady state rolling analysis is a generalization of *LOAD_BODY, allowing the user to apply body loads to part sets due to translational and rotational accelerations in a manner that is more general than the *LOAD_BODY capability. The *LOAD_STEAD_STATE_ROLLING may be invoked an arbitrary number of times in the problem as long as no part has the option applied more than once and they can be applied to arbitrary meshes (i.e., axisymmetric spun meshes aren't required).

Card 1	1	2	3	4	5	6	7	8
Variable	ID	PSID						
Type	I	I						
Default	none	none						
Card 2	1	2	3	4	5	6	7	8
Variable	N1	N2	LCD1	LCD1R				
Type	I	I	I	I				
Default	0	0	0	0				
Card 3	1	2	3	4	5	6	7	8
Variable	N3	N4	LCD2	LCD2R				
Type 4	I	I	I	I				
Default	0	0	0	0				

Card 4	1	2	3	4	5	6	7	8
Variable	N5	N6	LCD3	LCD3R				
Type	I	I	I	I				
Default	0	0	0	0				

VARIABLE	DESCRIPTION
ID	ID
PSID	Part set ID
N1	Node 1 defining rotational axis
N2	Node 2 defining rotational axis
LCD1	Load curve defining angular velocity around rotational axis.
LCD1R	Optional load curve defining angular velocity around rotational axis for dynamic relaxation. LCD1 is used during dynamic relaxation if LCD1R isn't defined.
N3	Node 3 defining turning axis
N4	Node 4 defining turning axis
LCD2	Load curve defining angular velocity around turning axis.
LCD2R	Optional load curve defining angular velocity around turning axis for dynamic relaxation. LCD2 is used during dynamic relaxation if LCD2R isn't defined.
N5	Node 5 defining translational direction
N6	Node 6 defining translational direction
LCD3	Load curve defining translational velocity in translational direction.
LCD3R	Optional load curve defining translational velocity in translational direction. LCD3 is used during dynamic relaxation if LCD3R isn't defined.

Remarks:

The steady state rolling capability adds inertial body loads in terms of a moving reference defined by the user input. The current coordinates are defined in terms of the displacement, *u*, and the moving reference frame, *Y*,

$$x_{SSR} = u + Y \quad \dot{x}_{SSR} = \dot{u} + \dot{Y} \quad \ddot{x}_{SSR} = \ddot{u} + \ddot{Y}$$

$$Y = R(\omega_2 t)\left[R(\omega_1 t)(X - X_O) - X_C \right] + Y_T(t)$$

where *R* is the rotation matrix obtained by integrating the appropriate angular velocity, the magnitude of the angular velocities ω_1 and ω_2 are defined by load curves LCD1 and LCD2 respectively, and the directions are defined by the current coordinates of the node pairs N1-N2 and N3-N4 (see Figure XX). The velocity corresponding to the translational term, $Y_T(t)$, is defined in magnitude by LCD3 and in direction by the node pair N5-N6. The initial coordinates of the nodes are *X*, X_O is the initial coordinate vector of node N1 and X_C is the initial coordinate vector of node N3. If data defining an angular velocity is not specified, the velocity is defaulted to zero, and *R* is the identity matrix. In a similar manner, if the translational velocity isn't specified, it is defaulted to zero.

This capability is useful for initializing the stresses and velocity of tires during dynamic relaxation, and rolling processes in manufacturing. It is available for implicit and explicit simulations and is invoked for dynamic relaxation by specifying that the load curves are used during dynamic relaxation. At the end of the dynamic relaxation, the velocities of the parts are set to $\dot{x}_{SSR}$ and the remaining parts are initialized according to the input file.

Users must ensure that the appropriate load curves are turned on during the relaxation process, and if implicit dynamic relaxation is used, that sufficient constraints are applied during the initialization to remove any rigid body motion and that they are removed at the end of the dynamic relaxation. The implicit iteration convergence rate is often improved by adding the geometric stiffness matrix using *CONTROL_IMPLICIT_GENERAL. A consistent tangent matrix is available by using *CONTROL_IMPLICIT_GENERAL, and while it improves the convergence rate with problems with small strains, it is often unstable for problems with large strains. The *CONTROL_STEADY_STATE_ROLLING options should be used to ramp up the frictional forces to obtain smooth solutions and good convergence rates.

To obtain the free-rolling angular velocity, the tire should be first inflated, then brought into contact with the road while the frictional force is ramped up with a load curve and a large value of SCL_K specified in *CONTROL_STEADY_STATE_ROLLING. The angular velocity of the tire is then slowly varied over a range that covers the free rolling velocity. The free rolling velocity is obtained when either the frictional force in the direction of rolling or the moment about the tire axis is near zero. For a tire with an initial radius of *R* and a translational velocity of *V*, the approximate value for the free rolling value of the rolling velocity is $\omega = \frac{V}{(1+\varepsilon)R}$, where ε is the hoop strain of the rolling tire. For a first guess, the hoop strain can

be set to 0.0, and the rolling velocity will be within 10% of the actual value. After the first calculation, a smaller range bracketing the free rolling velocity should be used in a second calculation to refine the free rolling velocity. An accurate value of the free rolling velocity is necessary for subsequent analyses, such as varying the slip angle of the tire.

A time varying slip angle can be specified by moving one of the nodes defining the direction vector of the translational velocity. To check that the stiffness scale factor in *CONTROL_STEADY_STATE_ROLLING is high enough, a complete cycle from a zero slip angle to a maximum value, then back to zero, should be performed. If the loading and unloading values are reasonably close, then the stiffness scale factor is adequate.

***LOAD_STIFFEN_PART**

Purpose: Staged construction. Available for solid, shell, and beam elements.

Note: This keyword card is available starting in release 3 of version 971.

Card 1	1	2	3	4	5	6	7	8
Variable	PID	LC	(blank)	STGA	STGR			
Type	I	I		I	I			
Default	none	0		0	0			

VARIABLE	DESCRIPTION
PID	Part ID
LC	Load curve defining factor vs. time
STGA	Construction stage at which part is added (optional)
STGR	Construction stage at which part is removed (optional)

Remarks:

1. In many cases it is more convenient to use *DEFINE_STAGED_CONSTRUCTION_PART – this card creates *LOAD_STIFFEN_PART data automatically.

2. For parts that are initially present but are excavated (removed) during the analysis, the stiffness factor starts at 1.0. During the excavation time, it ramps down to a small value such as 1.0E-6. The excavation time should be sufficiently long to avoid introducing shock or dynamic effects. For parts that are introduced during the construction, e.g. retaining walls, the elements are initially present in the model but the factor is set to a low value such as 1.0e-6. During the construction time the factor should be ramped up to 1.0. The construction time should be sufficiently long to avoid shock or dynamic effects. A factor that ramps up from 1.0E-6 to 1.0, then reduces back to 1.0E-6, can be used for temporary retaining walls, props, etc.

3. When the factor is increasing, it applies only to the stiffness and strength of the material in response to subsequent strain increments, not to any existing stresses.

4. When the factor is decreasing, it applies also to existing stresses as well as to the stiffness and strength.

5. This feature works with all material models when used only to reduce the stiffness (e.g. parts that are excavated, not parts that are added during construction). It works for most material types in all other cases, except those few materials that re-calculate stresses each time step from total strains (elastic, SOIL_BRICK, rubber models, orthotropic elastic, fabric, etc). There is no error check at present to detect STIFFEN_PART being used with an inappropriate material model. Symptoms of resulting problems would include non-physical large stresses when a part stiffens, due to the accumulated strains in the "dormant" material since the start of the analysis.

6. This feature is generally used with *LOAD_GRAVITY_PART. The same curve is often used for the stiffness factor and the gravity factor.

7. There are 3 methods of defining the factor-versus-time:

    1. LC overrides all the other methods if non-zero
    2. STGA, STGR refer to stages at which the part is added and removed – the stages are defined in *DEFINE_CONSTRUCTION_STAGES. If STGA is zero, the part has full stiffness at time zero. If not, it ramps up from the small factor FACT (on *CONTROL_STAGED_CONSTRUCTION) up to 1.0 over the ramp time at the start of stage STGA. If STGR is zero, the stiffness factor continues at 1.0 until the end of the analysis. If not, it ramps down from 1.0 to FACT over the ramp time at the start of stage STGR.
    3. *DEFINE_STAGED_CONSTRUCTION_PART can be used instead of *LOAD_STIFFEN_PART to define this loading. During initialization, a *LOAD_STIFFEN_PART card will be created and the effect is the same as using the STGA, STGR method described above.

***LOAD_SUPERPLASTIC_FORMING**

Purpose: Perform superplastic forming (SPF) analyses. This option can be applied to both solid and shell elements. The pressure loading controlled by the load curve ID given below is scaled to maintain a constant maximum strain rate.

This option must be used with material model 64, *MAT_RATE_SENSITIVE_POWERLAW_PLASTICITY, for strain rate sensitive, powerlaw plasticity. For the output of data, see *DATABASE_SUPERPLASTIC_FORMING. Mass scaling is recommended in SPF applications.

New options to compute the target strain rate value with various averaging techniques and autojump options to control the simulation are implemented. Strain-rate speed up is also available. See Remarks 5-7 for details.

Card 1

Variable	LCP1	CSP1	NCP1	LCP2	CSP2	NCP2	PCTS1	PCTS2
Type	I	I	F	I	I	F	F	F
Default	none	none	none.	none	none	none	100.0	100.0
Remarks				1	1	1		

Card 2

Variable	ERATE	SCMIN	SCMAX	NCYL	(Not Used)	LEVEL	TSRCH	AT
Type	F	F	F	I		I	F	F
Default	none	none	none.	0		0	none	0.0
Remarks				2		5		

Card 3

Variable	TPEAK	TNEG	TOSC	POSC	PDROP	RILIM	RDLIM	STR
Type	F	F	F	F	F	F	F	F
Default	10.0	5.0	10.0	1.0	2.0	1.0	1.0	0.0
Remarks								6

Card 4

Variable	THRES	LOWER	UPPER	TFACT	NTFCT			
Type	F	F	F	F	I			
Default	5.0	90.0	99.0	1.0	10			
Remarks				7	7			

VARIABLE	DESCRIPTION
LCP1	Load curve number for Phase I pressure loading, (see *DEFINE_ CURVE).
CSP1	Contact surface number to determine completion of Phase 1.
NCP1	Percent of nodes in contact to terminate Phase I, (see *CONTACT_ *OPTION*).
LCP2	Load curve number for Phase II pressure loading (reverse), (see *DEFINE_CURVE).
CSP2	Contact surface to determine completion of Phase II, (see *CONTACT_ *OPTION*).
NCP2	Percent of nodes in contact to terminate Phase II.

VARIABLE	DESCRIPTION
PCTS1	Percentage of nodes-in-contact to active autojump in Phase I forming.
PCTS2	Percentage of nodes-in-contact to active autojump in Phase II forming.
ERATE	Desired strain rate. This is the time derivative of the logarithmic strain.
SCMIN	Minimum allowable value for load curve scale factor. To maintain a constant strain rate the pressure curve is scaled. In the case of a snap through buckling the pressure may be removed completely. By putting a value here the pressure will continue to act but at a value given by this scale factor multiplying the pressure curve.
SCMAX	Maximum allowable value for load curve scale factor. Generally, it is a good idea to put a value here to keep the pressure from going to unreasonable values after full contact has been attained. When full contact is achieved the strain rates will approach zero and pressure will go to infinity unless it is limited or the calculation terminates.
NCYL	Number of cycles for monotonic pressure after reversal.
LEVEL	Criterion to compute averaged maximum of controlling variable: 0: no average used. >=1: averaging over neighbors of element with peak value of controlling variable. This parameter determines the level of neighbors search. -1: averaging over elements within selective range of peak controlling variable.
TSRCH	Time interval to conduct neighbors search.
AT	Time when SPF Phase I simulation starts.
TPEAK	Additional run time to terminate simulation when maximum pressure is reached.
TNEG	Additional run time to terminate simulation when percentage change of nodes-in-contact is zero or negative.
TOSC	Additional run time to terminate simulation when percentage change of nodes-in-contact oscillates within a specific value.
POSC	Percentage change to define the oscillation of percentage of nodes-in-contact.

VARIABLE	DESCRIPTION
PDROP	Drop in percentage of nodes-in-contact from the maximum to terminate simulation after the specified termination percentage has been reached.
STR	Autojump option or strike-through time (period of time without autojump check): 0: no autojump -1: autojump controlled by peak pressure -2: autojump controlled by percentage of nodes in contact -3: autojump controlled by both above >0: strike-through time, then same as STR=-3
THRES	Threshold percentage that gives the threshold value above which elements are considered for average.
LOWER	Lower percentile of elements above the threshold value to be included for average.
UPPER	Upper percentile of elements above the threshold value to be included for average.
RILIM	Maximum percentage change for pressure increment.
RDLIM	Maximum percentage change for pressure decrement.
TFACT	Strain rate peed up factor
NTFCT	Number of computing cycles to ramp up speedup

Remarks:

1. Optionally, a second phase can be defined. In this second phase a unique set of pressure segments must be defined whose pressure is controlled by load curve 2. During the first phase, the pressure segments of load curve 2 are inactive, and likewise, during the second phase the pressure segments of the first phase are inactive. When shell elements are used the complete set of pressure segments can be repeated in the input with a sign reversal used on the load curve. When solid elements are used the pressure segments for each phase will, in general, be unique.

2. This is an ad hoc parameter which should probably not be used.

3. Data in the output files “pressure”, “curve1”, and “curve2”, may be plotted using ASCII > superpl in LS-PREPOST. The file “curve2” is created only if the second phase is active. See *DATABASE_SUPERPLASTIC_FORMING.

4. The constraint method contact, *CONTACT_CONSTRAINT_NODES_TO_SURFACE, is recommended for superplastic forming simulations since the penalty methods are not as reliable when mass scaling is applied. Generally, in superplastic simulations mass scaling is used to enable the calculation to be carried out in real time.

5. In order to reduce the oscillation in pressure, the maximum strain rate used to adjust the pressure load is calculated by special averaging algorithm. There are two options available:
   Averaging over neighbors of element with maximum strain rate: In this method, the element that has the maximum strain rate is stored in each cycle of the computation. The elements close to the element with the maximum strain rate are searched and stored in an array. The averaged maximum strain rate is computed over the neighboring elements. The user can input an integer number to control the level of neighbors search, which will affect the total number of elements for average. Because the neighbors search is time consuming, the user can input a time interval to limit the occurrence of searching. The neighbors search is conducted only when the simulation time reaches the specified time or the element with maximum strain rate falls out of the array of neighbors.
   Averaging over elements within selective range of strain rate: In this method, all elements that have strain rate above a threshold value (a threshold percentage of maximum strain rate) are sorted according to their strain rate and the elements between the user specified lower percentile and upper percentile are selected to average the strain rate.

6. The SPF simulation can be controlled by various autojump options. When autojump conditions are met, the SPF simulation will be either terminated or continued from phase I to phase II simulation. The autojump check can be held inactive by setting a strikethrough time. In this case the SPF simulation will continue for that period of time and only be interrupted when the percentage of nodes-in-contact reaches 100% for a specified time. The available autojump conditions are:
   Peak pressure is reached and stays for certain time: The peak pressure is determined by the maximum allowable scale factor and the load curve. The simulation will continue for a user specified time before termination.
   User specified percentage of nodes-in-contact is reached: The simulation will be terminated or continued to Phase II automatically if one of the following conditions is met:
   1) If the change of the percentage of nodes-in-contact is zero or negative for a specified time.
   2) If the percentage of nodes-in-contact oscillates in a specified range for a specified time.
   3) If the percentage of nodes-in-contact drops more than a specified value from the maximum value recorded.
   4) If the percentage of nodes-in-contact reaches a user specified stop value.

7. In order to speed up the simulation of the superplastic forming process, we scale down the computation time. By doing this we increase the strain rate allowed in the SPF

process, resulting in reduced simulation time. However, caution should be utilized with this speedup as it may affect the accuracy of the results. We recommend no or small strain rate speed up for simulations with complex geometry or tight angles.

***LOAD_SURFACE_STRESS_**{*OPTION*}

Available options include:

<BLANK>

SET

Purpose: Store segment pressures from contact and applied pressure loads on the upper and lower surfaces of the shell surface. Applicable only when the THERMAL_FRICTION option of *CONTACT is invoked.

Card 1

Variable	PID/PSID							
Type	I							

Card 2

Variable	LSCID1	LSCID2	LSCID3	LSCID4	LSCID5	LSCID6	LSCID7	LSCID8
Type	I	I	I	I	I	I	I	I

Card 3

Variable	USCID1	USCID2	USCID3	USCID4	USCID5	USCID6	USCID7	USCID8
Type	I	I	I	I	I	I	I	I

VARIABLE	DESCRIPTION
PID/PSID	Part ID or if option set is active, part set ID.
LSCID*n*	Lower surface contact ID's. Up to eight ID's can be defined. These contacts contribute to the pressure acting on the lower surface of the shell. If the pressure on the lower surface is due to applied pressure loads, specify a -1 instead of a contact ID. Only one, -1, may exist in the set of 8.
USCID*n*	Upper surface contact ID's. Up to eight ID's can be defined. These contacts contribute to the pressure acting on the upper surface of the shell. . If the pressure on the upper surface is due to applied pressure loads, specify a -1 instead of a contact ID. Only one, -1, may exist in the set of 8.

LOAD_THERMAL_OPTION*

Available options include:

CONSTANT

CONSTANT_NODE

LOAD_CURVE

TOPAZ

VARIABLE

VARIABLE_NODE

Purpose: To define nodal temperatures that thermally load the structure. Nodal temperatures defined by the *LOAD_THERMAL_*OPTION* method are all applied in a structural only analysis. They are ignored in a thermal only or coupled thermal/structural analysis, see *CONTROL_THERMAL_*OPTION*.

All the *LOAD_THERMAL options cannot be used in conjunction with each other. Only those of the same thermal load type, as defined below in column 2, may be used together.

*LOAD_THERMAL_CONSTANT	- Thermal load type 1
*LOAD_THERMAL_CONSTANT_NODE	- Thermal load type 1
*LOAD_THERMAL_LOAD_CURVE	- Thermal load type 2
*LOAD_THERMAL_TOPAZ	- Thermal load type 3
*LOAD_THERMAL_VARIABLE	- Thermal load type 4
*LOAD_THERMAL_VARIABLE_NODE	- Thermal load type 4

*LOAD_THERMAL_CONSTANT

Purpose: Define nodal sets giving the temperature that remains constant for the duration of the calculation. The reference temperature state is assumed to be a null state with this option. A nodal temperature state, read in above and held constant throughout the analysis, dynamically loads the structure. Thus, the temperature defined can also be seen as a relative temperature to a surrounding or initial temperature.

Card 1

Variable	NSID	NSIDEX	BOXID					
Type	I	I	I					
Default	none	0.	0.					

Card 2

Variable	T	TE						
Type	F	F						
Default	0.	0.						

VARIABLE	DESCRIPTION
NSID	Nodal set ID containing nodes for initial temperature (see *SET_NODES): EQ.0: all nodes are included:
NSIDEX	Nodal set ID containing nodes that are exempted from the imposed temperature (optional).
BOXID	All nodes in box which belong to NSID are initialized. Others are excluded (optional).
T	Temperature
TE	Temperature of exempted nodes (optional)

LOAD_THERMAL_CONSTANT_ELEMENT_OPTION*

Available options include:

BEAM

SHELL

SOLID

TSHELL

Purpose: Define a uniform element temperature that remains constant for the duration of the calculation. The reference temperature state is assumed to be a null state. An element temperature, read in above and held constant throughout the analysis, dynamically loads the structure. The defined temperature can also be seen as a relative temperature to a surrounding or initial temperature.

Card	1	2	3	4	5	6	7	8
Variable	EID	T						
Type	I	F						
Default	none	0.						

VARIABLE	DESCRIPTION
EID	Element ID
T	Temperature, see remark below.

Remarks:

1. The temperature range for the constitutive constants in the thermal materials must include the reference temperature of zero. If not termination will occur with a temperature out-of-range error immediately after the execution phase is entered.

*LOAD_THERMAL_CONSTANT_NODE

Purpose: Define nodal temperature that remains constant for the duration of the calculation. The reference temperature state is assumed to be a null state with this option. A nodal temperature state, read in above and held constant throughout the analysis, dynamically loads the structure. Thus, the temperature defined can also be seen as a relative temperature to a surrounding or initial temperature.

Card	1	2	3	4	5	6	7	8
Variable	NID	T						
Type	I	F						
Default	none	0.						

VARIABLE	DESCRIPTION
NID	Node ID
T	Temperature, see remark below.

Remarks:

1. The temperature range for the constitutive constants in the thermal materials must include the reference temperature of zero. If not termination will occur with a temperature out-of-range error immediately after the execution phase is entered.

*LOAD_THERMAL_LOAD_CURVE

Purpose: Nodal temperatures will be uniform throughout the model and will vary according to a load curve. The temperature at time=0 becomes the reference temperature for the thermal material. The reference temperature is obtained from the optional curve for dynamic relaxation if this curve is used. The load curve option for dynamic relaxation is useful for initializing preloads.

Card	1	2	3	4	5	6	7	8
Variable	LCID	LCIDDR						
Type	I	I						
Default	none	0						

VARIABLE	DESCRIPTION
LCID	Load curve ID, see *DEFINE_CURVE, to define temperature versus time.
LCIDDR	An optional load curve ID, see *DEFINE_CURVE, to define temperature versus time during the dynamic relaxation phase.

***LOAD_THERMAL_TOPAZ**

Purpose: Nodal temperatures will be read in from the TOPAZ3D database. This file is defined in the EXECUTION SYNTAX, see GETTING STARTED.

*LOAD_THERMAL_VARIABLE

Purpose: Define nodal sets giving the temperature that is variable in the duration of the calculation. The reference temperature state is assumed to be a null state with this option. A nodal temperature state, read in above and varied according to the load curve, dynamically loads the structure. Thus, the defined temperatures are relative temperatures to an initial reference temperature.

Card 1

Variable	NSID	NSIDEX	BOXID					
Type	I	I	I					
Default	none	0.	0.					

Card 2

Variable	TS	TB	LCID	TSE	TBE	LCIDE		
Type	F	F	I	F	F	I		
Default	0.	0.	none	0.	0.	none		
Remark	1	1	1	1	1			

VARIABLE	DESCRIPTION
NSID	Nodal set ID containing nodes (see *SET_NODE_*OPTION*): EQ.0: all nodes are included.
NSIDEX	Nodal set ID containing nodes that are exempted (optional), (see *SET_NODE_*OPTION*).
BOXID	All nodes in box which belong to NSID are initialized. Others are excluded.
TS	Scaled temperature.

VARIABLE	DESCRIPTION
TB	Base temperature.
LCID	Load curve ID that multiplies the scaled temperature, (see *DEFINE_ CURVE).
TSE	Scaled temperature of the exempted nodes (optional).
TBE	Base temperature of the exempted nodes (optional).
LCIDE	Load curve ID that multiplies the scaled temperature of the exempted nodes (optional), (see *DEFINE_CURVE).

Remarks:

1. The temperature is defined as

   $T = T_{base} + T_{scale}\, f(t)$

   where f(t) is the current value of the load curve, T_{scale} is the scaled temperature, and T_{base} is the base temperature.

LOAD_THERMAL_VARIABLE_ELEMENT_OPTION*

Available options include:

BEAM

SHELL

SOLID

TSHELL

Purpose: Define element temperature that is variable during the calculation. The reference temperature state is assumed to be the temperature at time=0.0 with this option.

Card	1	2	3	4	5	6	7	8
Variable	EID	TS	TB	LCID				
Type	I	F	F	I				
Default	none	0.	0.	none				

VARIABLE	DESCRIPTION
NID	Element ID
TS	Scaled temperature
TB	Base temperature
LCID	Load curve ID defining a scale factor that multiplies the scaled temperature as a function of time, (see *DEFINE_ CURVE).

Remarks:

1. The temperature is defined as:

$$T = T_{base} + T_{scale}\, f(t)$$

where f(t) is the current value of the load curve, T_{scale} is the scaled temperature, and T_{base} is the base temperature

*LOAD_THERMAL_VARIABLE_NODE

Purpose: Define nodal temperature that is variable during the calculation. The reference temperature state is assumed to be a null state with this option. A nodal temperature state read in and varied according to the load curve dynamically loads the structure. Thus, the defined temperatures are relative temperatures to an initial reference temperature.

Card	1	2	3	4	5	6	7	8
Variable	NID	TS	TB	LCID				
Type	I	F	F	I				
Default	none	0.	0.	none				

VARIABLE	DESCRIPTION
NID	Node ID
TS	Scaled temperature
TB	Base temperature
LCID	Load curve ID that multiplies the scaled temperature, (see *DEFINE_ CURVE).

Remarks:

1. The temperature is defined as:

   $T = T_{base} + T_{scale}\, f(t)$

   where f(t) is the current value of the load curve, T_{scale} is the scaled temperature, and T_{base} is the base temperature

***LOAD_THERMAL_VARIABLE_SHELL_**{*OPTION*}

Available options include:

<BLANK>

SET

Purpose: Define a known temperature time history as a function of the through-thickness coordinate for the shell elements.

Card 1

1	2	3	4	5	6	7	8
Variable	ID	EID/SID						
Type	I	I						
Default	none	2						

Card 2, 3, 4, etc. Input is terminated when a "*" card is found.

Card 2…

1	2	3	4	5	6	7	8
Variable	TBASE	TSCALE	TCURVE	TCURDR	ZCO			
Type	F	F	I	I	F			
Default	0	1.0	constant	TCURVE	-1/+1			

VARIABLE	DESCRIPTION
ID	Load case ID
EID/SID	Shell/Shell set ID.
TBASE	Base temperature
TSCALE	Scale factor on temperature from load curve
TCURVE	Load curve ID for temperature vs time

VARIABLE	DESCRIPTION
TCURDR	Load curve ID used during dynamic relaxation
ZCO	Relative coordinate through-thickness (-1.0 to +1.0)

Remarks:

1. The temperature is defined as:

$$T = T_{base} + T_{scale}\, f(t)$$

   where f(t) is the current value of the load curve, T_{scale} is the scaled temperature, and T_{base} is the base temperature.

2. If a load curve ID is undefined, unity is used instead of the value from the curve.

3. Through-thickness points must be defined in order of increasing ZCO (-1.0 to +1.0). CZO=+1.0 is the top surface of the element, i.e. the element surface in the positive outward normal vector direction from the mid-plane.

4. At least two points must be defined.

5. If the element has multiple in-plane integration points – the same temperature distribution is used at each in-plane integration point.

6. If a shell's temperature distribution is defined using this card any values defined by *LOAD_THERMAL_NODE are ignored for that shell.

*LOAD_VOLUME_LOSS

Purpose: To represent the effect of tunneling on surrounding structures, it is common to assume that a pre-defined fraction (e.g., 2%) of the volume occupied by the tunnel is lost during the construction process. Available for solid elements only. This feature is currently unavailable in MPP.

Note: This keyword card will be available starting in release 3 of version 971.

Card	1	2	3	4	5	6	7	8
Variable	PSID	COORD	LCUR	FX	FY	FZ	PMIN	FACTOR
Type	I	I	I	F	F	F	F	F
Default	none	0	0	1	1	1	-1.e20	.01

VARIABLE	DESCRIPTION
PSID	Part Set ID
COORD	(Leave blank at present)
LCUR	Curve ID containing volume fraction lost vs. time
FX	Fraction of strain occurring in X-direction
FY	Fraction of strain occurring in Y-direction
FZ	Fraction of strain occurring in Z-direction
PMIN	(Leave blank)
FACTOR	Feedback factor

Remarks:

Volume loss is modeled by a process similar to thermal contraction: if the material is unrestrained it will shrink while remaining unstressed; if restrained, stresses will become more tensile. Typically the material surrounding the tunnel offers partial restraint; the volume loss algorithm adjusts the applied "thermal" strains to attempt to achieve the desired volume loss. Optionally, FX, FY and FZ may be defined: these will be treated as ratios for the X, Y and Z strains; this feature can be used to prevent contraction parallel to the tunnel axis.

The total volume of all the parts in the part set is monitored and output at the time-history interval (on *DATABASE_BINARY_D3THDT) to a file named *vloss_output*. This file contains lines of data (*time, volume1, volume2, volume3…*) where *volume1* is the total volume of elements controlled by the first *LOAD_VOLUME_LOSS card, *volume2* is the total volume of elements controlled by the second *LOAD_VOLUME_LOSS card, etc.

*NODE

The keywords defined in this section include:

***NODE_**{*OPTION*}

***NODE_MERGE_SET**

***NODE_MERGE_TOLERANCE**

***NODE_RIGID_SURFACE**

***NODE_SCALAR_**{*OPTION*}

***NODE_TRANSFORM**

***NODE_**{*OPTION*}

Available options include:

<BLANK>

MERGE

Purpose: Define a node and its coordinates in the global coordinate system. Also, the boundary conditions in global directions can be specified. Generally, nodes are assigned to elements; however, exceptions are possible, see remark 2 below. The nodal point ID must be unique relative to other nodes defined in the *NODE section. The MERGE option is usually applied to boundary nodes on disjoint parts and only applies to nodes defined when the merge option is invoked. With this option, nodes with identical coordinates are replaced during the input phase by the first node encountered that shares the coordinate. During the merging process a tolerance is used to determine whether a node should be merged. This tolerance can be defined using the keyword *NODE_MERGE_TOLERANCE keyword, which is recommended over the default value. See the *NODE_MERGE_TOLERANCE input description in the next section.

Card Format (I8,3E16.0,2F8.0)

Card 1	1	2	3	4	5	6	7	8	9	10
Variable	NID	X		Y		Z		TC	RC	
Type	I	F		F		F		F	F	
Default	none	0.		0.		0.		0.	0.	
Remarks								1	1	

VARIABLE	DESCRIPTION
NID	Node number
X	x coordinate
Y	y coordinate
Z	z coordinate

VARIABLE	DESCRIPTION
TC	Translational Constraint: EQ.0: no constraints, EQ.1: constrained x displacement, EQ.2: constrained y displacement, EQ.3: constrained z displacement, EQ.4: constrained x and y displacements, EQ.5: constrained y and z displacements, EQ.6: constrained z and x displacements, EQ.7: constrained x, y, and z displacements.
RC	Rotational constraint: EQ.0: no constraints, EQ.1: constrained x rotation, EQ.2: constrained y rotation, EQ.3: constrained z rotation, EQ.4: constrained x and y rotations, EQ.5: constrained y and z rotations, EQ.6: constrained z and x rotations, EQ.7: constrained x, y, and z rotations.

Remarks:

1. Boundary conditions can also be defined on nodal points in a local (or global) system by using the keyword *BOUNDARY_SPC. For other possibilities also see the *CONSTRAINED keyword section of the manual.

2. A node without an element or a mass attached to it will be assigned a very small amount of mass and rotary inertia. Generally, massless nodes should not cause any problems but in rare cases may create stability problems if these massless nodes interact with the structure. Warning messages are printed when massless nodes are found. Also, massless nodes are used with rigid bodies to place joints, see *CONSTRAINED_EXTRA_NODES_*OPTION* and *CONSTRAINED_NODAL_RIGID_BODY.

*NODE_MERGE_SET

Purpose: The MERGE_SET option is applied to a set of boundary nodes on disjoint part. With this option, nodes with identical coordinates that are members of any node set ID defined by this keyword are replaced during the input phase by one node within the set or sets. Of the nodes sharing the same coordinates, the node chosen is the one with the smallest ID. During the merging process a tolerance is used to determine whether a node should be merged. This tolerance can be defined using the keyword *NODE_MERGE_TOLERANCE keyword, which is recommended over the default value. See the *NODE_MERGE_TOLERANCE input description in the next section. Only nodes contained within the specified sets will be merged. Nodes contained within the set are defined by the *NODE keyword. With is option, the keyword *NODE_MERGE is not needed.

Card Format (I10)

Card 1	1	2	3	4	5	6	7	8
Variable	NSID							
Type	I							
Default	none							

VARIABLE	DESCRIPTION
NSID	Node set ID containing list of nodes to be considered for merging.

***NODE_MERGE_TOLERANCE**

Purpose: Define a tolerance is determine whether a node should be merged for the keyword, *NODE_MERGE.

Card Format (E10.0)

Card 1	1	2	3	4	5	6	7	8
Variable	TOLR							
Type	F							
Default	yes							

VARIABLE	DESCRIPTION
TOLR	Physical distance used to determine whether to merge a nodal pair of nearby nodes. See remark below.

Remarks:

If the tolerance, TOLR, is undefined or if it is defaulted to zero, a value is computed as:

$$TOLR = 10^{-5} \cdot \frac{XMAX + YMAX + ZMAX - XMIN - YMIN - ZMIN}{3 \cdot \sqrt[3]{NUMNP}}$$

where XMIN, XMAX, YMIN,YMAX, ZMIN, and ZMAX represent the minimum and maximum values of the (x,y,z) nodal point coordinates in the global coordinate system, and NUMNP is the number of nodal points.

*NODE_RIGID_SURFACE

Purpose: Define a rigid node and its coordinates in the global coordinate system. These nodes are used to define rigid road surfaces and they have no degrees of freedom. The nodal points are used in the definition of the segments that define the rigid surface. See *CONTACT_RIGID_SURFACE. The nodal point ID must be unique relative to other nodes defined in the *NODE section.

Card Format (I8,3E16.0)

Card 1	1	2	3	4	5	6	7	8	9	10
Variable	NID	X		Y		Z				
Type	I	F		F		F				
Default	none	0.		0.		0.				
Remarks										

VARIABLE	DESCRIPTION
NID	Node number
X	x coordinate
Y	y coordinate
Z	z coordinate

***NODE_SCALAR_**{*OPTION*}

Available options include:

<BLANK>

VALUE

Purpose: Define a scalar nodal point which has one degree-of-freedom. The scalar point ID must be unique relative to other nodes defined in the *NODE section.

Define the following card if and if no option is picked

Card Format (2I8)

Card 1	1	2	3	4	5	6	7	8	9	10
Variable	NID	NDOF								
Type	I	I								
Default	none	0								
Remarks										

Define the following card for and only for option VALUE

Card Format (I8,3E16.0,I8)

Card 1	1	2	3	4	6	7	9
Variable	NID	X1	X2	X3	NDOF		
Type	I	F	F	F	I		
Default	none	0	0	0	0		
Remarks							

VARIABLE	DESCRIPTION
NID	Scalar node ID.
NDOF	Number of degrees-of-freedom EQ.0: fully constrained EQ.1: one degree-of-freedom EQ.2: two degrees-of-freedom EQ.3: three degrees-of-freedom
XI	Initial value of Ith degree of freedom.

***NODE_TRANSFORM**

Purpose: Perform a transformation on a node set based on a transformation defined by the keyword *DEFINE_TRANSFORMATION.

Card 1	1	2	3	4	5	6	7	8
Variable	TRSID	NSID						
Type	I	I						
Default	none	none						

VARIABLE	DESCRIPTION
TRSID	The ID of the transformation defined under *DEFINE_TRANSFORMATION.
NSID	Node set ID of the set of nodes to be subject to the transformation.

*PARAMETER

Two keywords are used in this section.

***PARAMETER**

***PARAMETER_EXPRESSION**

***PARAMETER_**{*OPTION*}

The available options are

<BLANK>

LOCAL

Purpose: Define the numerical values of parameter names referenced throughout the input file. The parameter definitions, if used, should be placed at the beginning of the input file following *KEYWORD or at the beginning of an include file if the LOCAL option is specified.

Define as many cards as necessary.

Card 1

Variable	PRMR1	VAL1	PRMR2	VAL2	PRMR3	VAL3	PRMR4	VAL4
Type	A	I or F	A	I or F	A	I or F	A	I or F
Default	none	none	none	none	none	none	none	none

Card 2…

Variable	PRMRn	VALn	PRMRn+1	VALn+1	...	...		
Type	A	I or F	A	I or F				
Default	none	none	none	none				

VARIABLE	DESCRIPTION
PRMRn	Define the nth parameter in a field of 10. Within this field the first character must be either an "R" for a real number or an "I" for an integer. Lower or upper case for "I" or "R" is okay. Following the type designation, define the name of the parameter using up to, but not exceeding seven characters. For example, when defining a shell thickness named, "SHLTHK", both inputs "RSHLTHK" or "R SHLTHK" can be used and placed anywhere in the field of 10. When referencing SHLTHK in the input file see Remark 1 below.

VARIABLE	DESCRIPTION
VALn	Define the numerical value of the n parameter as either a real or integer number consistent with preceding definition for PRMRn.

Remarks:

1. Parameters can be referenced anywhere in the input by placing an "&" at the first column of its field followed by the name of the parameter without blanks. If a minus sign "-" is placed directly before "&", i.e., "-&", with no space the sign of the numerical value will be switched.

2. *PARAMETER_LOCAL behaves like the *PARAMETER keyword with one difference. A parameter defined by *PARAMETER without the LOCAL option is visible and available at any later point in the input processing. Parameters defined via the LOCAL versions disappear when the input parser finishes reading the file in which they appear. LOCAL variables can temporarily mask non-LOCAL variables.

   For example, suppose you have the following input files:

   main.k:
```
*PARAMETER
R  VAL1    1.0
*PARAMETER
R  VAL2    2.0
*PARAMETER
R  VAL3    3.0
*INCLUDE
```
   file1

   file1:
```
*PARAMETER
R  VAL1    10.0
*PARAMETER_LOCAL
R  VAL2    20.0
*PARAMETER_LOCAL
R  VAL4    40.0
*INCLUDE
```
   file2

   …..

   Then, inside file2 we will see VAL1=10.0, VAL2=20.0, VAL3=3.0 and VAL4=40.0. In main.k, after returning from file1, we will see VAL1=10.0, VAL2=2.0, and VAL3=3.0. VAL4 will not exist.

   This allows for include files that can set all their own parameters without clobbering the parameters in the rest of the input.

***PARAMETER_DUPLICATION**

Purpose: The purpose is to control how the code behaves if a duplicate parameter definition is found in the input.

Card 1	1	2	3	4	5	6	7	8
Variable	DFLAG							
Type	I							
Default	1							

VARIABLE	DESCRIPTION
DFLAG	Flag to control treatment of duplicate parameter definitions: EQ.1: issue a warning and ignore the new definition (default) EQ.2: issue a warning and accept the new definition EQ.3: issue an error and ignore (terminates at end of input) EQ.4: accept silently EQ.5: ignore silently

Remarks:

A _LOCAL variable appearing in a file, which masks a non-_LOCAL parameter, won't trigger these actions; however, a _LOCAL that masks another _LOCAL or a non-_LOCAL that masks a non-_LOCAL will.

***PARAMETER_EXPRESSION**_{*OPTION*}

The available options are

<BLANK>

LOCAL

Purpose: Define the numerical values of parameter names referenced throughout the input file. Like the *PARAMETER keyword, but allows for general algebraic expressions, not simply fixed values. The LOCAL option allows for include files to contain their own unique expressions without clobbering the expressions in the rest of the input. See the *PARAMETER keyword above.

Define as many cards as necessary.

Card 1 1 2 3 4 5 6 7 8

Variable	PRMR1	EXPRESSION1
Type	A	A
Default	none	none

Card 2…

Variable	PRMRn	EXPRESSIONn
Type	A	A
Default	none	none

VARIABLE	DESCRIPTION
PRMRn	Define the nth parameter in a field of 10. Within this field the first character must be either an "R" for a real number or an "I" for an integer. Lower or upper case for "I" or "R" is okay. Following the type designation, define the name of the parameter using up to, but not exceeding seven characters. For example, when defining a shell thickness named, "SHLTHK", both inputs "RSHLTHK" or "R SHLTHK" can be used and placed anywhere in the field of 10. When referencing SHLTHK in the input file see Remark 1 below.

VARIABLE	DESCRIPTION
EXPRESSIONn	General expression which is evaluated, having the result stored in PRMRn. The following functions are available: sin, cos, tan, csc, sec, ctn, asin, acos, atan, atan2, sinh, cosh, tanh, asinh, acosh, atanh, min, max, sqrt, mod, abs, sign, int, aint, nint, anint, float, exp, log, log10, float, and general arithmetic expressions involving +, -, *, /, and **. The standard rules regarding operator precedence are obeyed, and nested parentheses are allowed. The expression can reference previously defined parameters (with or without the leading &). The expression can be continued on multiple lines simply by leaving the first 10 characters of the continuation line blank.

Remarks:

1. Parameters can be referenced anywhere in the input by placing an "&" at the first column of its field followed by the name of the parameter without blanks. Expressions can be included in the input when placed between brackets “<>” as long as the total line length does not exceed 80 columns.

2. The integer and real properties of constants and parameters are honored when evaluating expressions. So 2/5 becomes 0, but 2.0/5 becomes 0.4.

3. The sign, atan2, min, max, and mod functions all take two arguments. The others all take only 1.

4. Functions that use an angle as their argument, e.g., sin or cos, assume the angle is in radians.

*PART

The following keywords are used in this section:

***PART**_{*OPTION1*}_{*OPTION2*}_{*OPTION3*}_{*OPTION4*}

***PART_ADAPTIVE_FAILURE**

***PART_COMPOSITE**_{*OPTION*}

***PART_MODES**

***PART_SENSOR**

***PART_MOVE**

***PART**_{*OPTION1*}_{*OPTION2*}_{*OPTION3*}_{*OPTION4*}

For *OPTION1* the available options are

<BLANK>

INERTIA

REPOSITION

For *OPTION2* the available options are

<BLANK>

CONTACT

For *OPTION3* the available options are

<BLANK>

PRINT

For *OPTION4* the available options are

<BLANK>

ATTACHMENT_NODES

Options 1, 2, 3, and 4 may be specified in any order on the *PART card.

Purpose: Define parts, i.e., combine material information, section properties, hourglass type, thermal properties, and a flag for part adaptivity.

The INERTIA option allows the inertial properties and initial conditions to be defined rather than calculated from the finite element mesh. This applies to rigid bodies, see *MAT_RIGID, only. The REPOSITION option applies to deformable materials and is used to reposition deformable materials attached to rigid dummy components whose motion is controlled by either CAL3D or MADYMO. At the beginning of the calculation each component controlled by CAL3D/MADYMO is automatically repositioned to be consistent with the CAL3D/MADYMO input. However, deformable materials attached to these components will not be repositioned unless this option is used.

The CONTACT option allows part based contact parameters to be used with the automatic contact types a3, 4, a5, a10, 13, a13, 15 and 26, that is

*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE

*CONTACT_SINGLE_SURFACE,

*CONTACT_AUTOMATIC_NODES_TO_SURFACE,

*CONTACT_AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE,

*CONTACT_AUTOMATIC_SINGLE_SURFACE,

*CONTACT_AIRBAG_SINGLE_SURFACE,

*CONTACT_ERODING_SINGLE_SURFACE,

*CONTACT_AUTOMATIC_GENERAL.

The default values to use for these contact parameters can be specified on the *CONTACT input section card.

The PRINT option allows user control over whether output data is written into the ASCII files MATSUM and RBDOUT. See *DATABASE_ASCII.

Card 1

Variable	HEADING	
Type	C	
Default	none	
Remarks	1	

Card 2

Variable	PID	SECID	MID	EOSID	HGID	GRAV	ADPOPT	TMID
Type	I	A8	A8	A8	A8	I	I	A8
Default	none	none	none	0	0	0	0	0

Additional Cards are required for the INERTIA option. See remarks 3 and 4.

Card 3	1	2	3	4	5	6	7	8
Variable	XC	YC	ZC	TM	IRCS	NODEID		
Type	F	F	F	F	I	I		

Card 4

Variable	IXX	IXY	IXZ	IYY	IYZ	IZZ		
Type	F	F	F	F	F	F		

Card 5

Variable	VTX	VTY	VTZ	VRX	VRY	VRZ		
Type	F	F	F	F	F	F		

Optional card required for IRCS=1. Define two local vectors or a local coordinate system ID.

Card 6	1	2	3	4	5	6	7	8
Variable	XL	YL	ZL	XLIP	YLIP	ZLIP	CID	
Type	F	F	F	F	F	F	I	
Remark	2	2	2	2	2	2	none	

An additional Card is required for the REPOSITION option.

Optional	1	2	3	4	5	6	7	8
Variable	CMSN	MDEP	MOVOPT					
Type	I	I	I					

Additional Card is required for the CONTACT option.

<u>WARNING</u>: If FS, FD, DC, and VC are specified they will not be used unless FS is set to a negative value (-1.0) in the *CONTACT section. These frictional coefficients apply only to contact types: SINGLE_SURFACE, AUTOMATIC_GENERAL, AUTOMATIC_ SINGLE_SURFACE, AUTOMATIC_NODES_TO_..., AUTOMATIC_SURFACE_..., AUTOMATIC_ONE_WAY_..., and ERODING_SINGLE_SURFACE. Default values are input via *CONTROL_CONTACT input.

Optional	1	2	3	4	5	6	7	8
Variable	FS	FD	DC	VC	OPTT	SFT	SSF	
Type	F	F	F	F	F	F	F	

An additional Card is required for the PRINT option. This option applies to rigid bodies and provides a way to turn off ASCII output in files RBDOUT and MATSUM.

Optional	1	2	3	4	5	6	7	8
Variable	PRBF							
Type	I							

An additional Card is required for the ATTACHMENT_NODES option. All nodes are treated as attachment nodes if this option is not used. Attachment nodes apply to rigid bodies only. The motion of these nodes, which must belong to the rigid body, are updated each cycle. Other nodes in the rigid body are updated only for output purposes. Include all nodes in the attachment node set which interact with the structure through joints, contact, merged nodes, applied nodal point loads, and applied pressure. Include all nodes in the attachment node set if their displacements, accelerations, and velocities are to be written into an ASCII output file. Body force loads are applied to the c.g. of the rigid body.

Optional	1	2	3	4	5	6	7	8
Variable	ANSID							
Type	I							

VARIABLE	DESCRIPTION
HEADING	Heading for the part
PID	Part identification
SECID	Section identification defined in the *SECTION section
MID	Material identification defined in the *MAT section
EOSID	Equation of state identification defined in the *EOS section. Nonzero only for solid elements using an equation of state to compute pressure.
HGID	Hourglass/bulk viscosity identification defined in the *HOURGLASS Section: EQ.0: default values are used.
GRAV	Part initialization for gravity loading. This option initializes hydrostatic pressure in the part due to gravity acting on an overburden material. This option applies to brick elements only and must be used with the *LOAD_DENSITY_DEPTH option: EQ.0: all parts initialized, EQ.1: only current material initialized.
ADPOPT	Indicate if this part is adapted or not. (See also *CONTROL_ ADAPTIVITY): LT.0: R-adaptive remeshing for 2-D solids, |ADOPT| gives the load curve ID that defines the element size as a function of time. EQ.0: adaptive remeshing is inactive for this part ID, EQ.1: H-adaptive for 3-D shells. EQ.2: R-adaptive remeshing for 2-D solids, 3-D tetrahedrons and 3-D EFG.

VARIABLE	DESCRIPTION
TMID	Thermal material property identification defined in the *MAT_THERMAL Section. Thermal properties must be specified for all solid, shell, and thick shell parts if a thermal or coupled thermal structural/analysis is being performed. Beams and discrete elements are not considered in thermal analyses. EQ.0: defaults to MID
XC	x-coordinate of center of mass. If nodal point, NODEID, is defined XC, YC, and ZC are ignored and the coordinates of the nodal point, NODEID, are taken as the center of mass.
YC	y-coordinate of center of mass
ZC	z-coordinate of center of mass
TM	Translational mass
IRCS	Flag for inertia tensor reference coordinate system: EQ.0: global inertia tensor, EQ.1: local inertia tensor is given in a system defined by the orientation vectors.
NODEID	Nodal point defining the CG of the rigid body. This node should be included as an extra node for the rigid body; however, this is not a requirement. If this node is free, its motion will not be updated to correspond with the rigid body after the calculation begins.
IXX	I_{xx}, xx component of inertia tensor (see Remark 4)
IXY	$I_{xy,}$ xy component of inertia tensor (see Remark 4)
IXZ	$I_{xz,}$ xz component of inertia tensor (see Remark 4)
IYY	I_{yy}, yy component of inertia tensor (see Remark 4)
IYZ	$I_{yz,}$ yz component of inertia tensor (see Remark 4)
IZZ	I_{zz}, zz component of inertia tensor (see Remark 4)
VTX	initial translational velocity of rigid body in x direction
VTY	initial translational velocity of rigid body in y direction
VTZ	initial translational velocity of rigid body in z direction
VRX	initial rotational velocity of rigid body about x axis

VARIABLE	DESCRIPTION
VRY	initial rotational velocity of rigid body about y axis
VRZ	initial rotational velocity of rigid body about z axis
XL	x-coordinate of local x-axis. Origin lies at (0,0,0).
YL	y-coordinate of local x-axis
ZL	z-coordinate of local x-axis
XLIP	x-coordinate of vector in local x-y plane
YLIP	y-coordinate of vector in local x-y plane
ZLIP	z-coordinate of vector in local x-y plane
CID	Local coordinate system ID, see *DEFINE_COORDINATE_.... With this option leave fields 1-6 blank.
CMSN	CAL3D segment number/MADYMO system number. See the numbering in the corresponding program.
MDEP	MADYMO ellipse/plane number: GT.0: ellipse number, EQ.0: default, LT.0: absolute value is plane number.
MOVOPT	Flag to deactivate moving for merged rigid bodies, see *CONSTRAINED_RIGID_BODIES. This option allows a merged rigid body to be fixed in space while the nodes and elements of the generated CAL3D/MADYMO parts are repositioned: EQ.0: merged rigid body is repositioned, EQ.1: merged rigid body is not repositioned.
FS	Static coefficient of friction. The functional coefficient is assumed to be dependent on the relative velocity v_{rel} of the surfaces in contact $\mu_c = FD + (FS - FD)e^{-DC\cdot\lvert v_{rel}\rvert}$.
FD	Dynamic coefficient of friction. The functional coefficient is assumed to be dependent on the relative velocity v_{rel} of the surfaces in contact $\mu_c = FD + (FS - FD)e^{-DC\cdot\lvert v_{rel}\rvert}$.
DC	Exponential decay coefficient. The functional coefficient is assumed to be dependent on the relative velocity v_{rel} of the surfaces in contact $\mu_c = FD + (FS - FD)e^{-DC\cdot\lvert v_{rel}\rvert}$.

VARIABLE	DESCRIPTION
VC	Coefficient for viscous friction. This is necessary to limit the friction force to a maximum. A limiting force is computed $F_{lim} = VC \cdot A_{cont}$. A_{cont} being the area of the segment contacted by the node in contact. The suggested value for *VC* is to use the yield stress in shear $VC = \frac{\sigma_o}{\sqrt{3}}$ where σ_o is the yield stress of the contacted material.
OPTT	Optional contact thickness. This applies to shells only.
SFT	Optional thickness scale factor for PART ID in automatic contact (scales true thickness). This option applies only to contact with shell elements. True thickness is the element thickness of the shell elements.
SSF	Scale factor on default slave penalty stiffness for this PART ID whenever it appears in the contact definition. If zero, SSF is taken as unity.
PRBF	Print flag for RBDOUT and MATSUM files. EQ.0: default is taken from the keyword *CONTROL_OUTPUT, EQ.1: write data into RBDOUT file only EQ.2: write data into MATSUM file only EQ.3: do not write data into RBDOUT and MATSUM
ANSID	Attachment node set ID. This option should be used very cautiously and applies only to rigid bodies. The attachment point nodes are updated each cycle whereas other nodes in the rigid body are updated only in the output databases. All loads seen by the rigid body must be applied through this nodal subset or directly to the center of gravity of the rigid body. If the rigid body is in contact this set must include all interacting nodes. EQ.0: All nodal updates are skipped for this rigid body. The null option can be used if the rigid body is fixed in space or if the rigid body does not interact with other parts, e.g., the rigid body is only used for some visual purpose.

Remarks:

1. HEADING default is standard material description, e.g. Material Type 1.

2. The local cartesian coordinate system is defined as described in *DEFINE_ COORDINATE_VECTOR. The local z-axis vector is the vector cross product of the x-axis and the in plane vector. The local y-axis vector is finally computed as the vector cross product of the z-axis vector and the x-axis vector. The local coordinate system defined by CID has the advantage that the local system can be defined by nodes in the rigid body which makes repositioning of the rigid body in a preprocessor much easier since the local system moves with the nodal points.

3. When specifying mass properties for a rigid body using the inertia option, the mass contributions of deformable bodies to nodes which are shared by the rigid body should be considered as part of the rigid body.

4. If the inertia option is used, all mass and inertia properties of the body must be specified for there are no default values. The inertia terms are always with respect to the center of mass of the rigid body. The reference coordinate system defines the orientation of the axes, not the origin. Note that the off-diagonal terms of the inertia tensor are opposite in sign from the products of inertia.

5. The initial velocity of the rigid body may be overwritten by the *INITIAL_VELOCITY card.

See parameter IRIGID on this card.

*PART_ADAPTIVE_FAILURE

Purpose: This is an option for two-dimensional adaptivity to allow a part that is singly connected to split into two parts. This option is under development and will be generalized in the future to allow the splitting of parts that are multiply connected.

Card 1	1	2	3	4	5	6	7	8
Variable	PID	T						
Type	I	F						

VARIABLE	DESCRIPTION
PID	Part ID
T	Thickness. When the thickness of the part reaches this minimum value the part is split into two parts. *The value for T should be on the order of the element thickness of a typical element.*

***PART_COMPOSITE_{*OPTION*}**

Available options include:

<BLANK>

CONTACT

TSHELL

Purpose: The following input provides a simplified method of defining a composite material model for shell elements and thick shell elements that eliminates the need for user defined integration rules and part ID's for each composite layer. The material ID, thickness, material angle and thermal material ID for each through-thickness integration point of a composite shell or thick shell are provided below (up to two integration points per card). The integration point data should be given sequentially starting with the bottommost integration point. The total number of integration points is determined by the number of entries on these cards. For shells, the total thickness of the composite shell is the sum of the integration point thickness THICKi; consequently, the shell thickness is assumed to be uniform. For thick shells, the total thickness is defined by the location of nodes on the top and bottom surface, so the THICKi values are scaled to fit the element. When *PART_COMPOSITE is used, a section definition, *SECTION_SHELL or *SECTION_TSHELL, and integration rule definition, *INTEGRATION_SHELL, are unnecessary.

The CONTACT option allows part based contact parameters to be used with the automatic contact types a3, 4, a5, a10, 13, a13, 15 and 26, which are listed under the *PART definition above.

Card 1

Variable	HEADING	
Type	C	
Default	none	

Card 2

Variable	PID	ELFORM	SHRF	NLOC	MAREA	HGID	ADPOPT	ITHELFM
Type	I	I	F	F	F	A8	I	I

Default	none	0	0.0	0.0	0.0	0	0	0

Additional Card is required for the CONTACT option.

WARNING: If FS, FD, DC, and VC are specified they will not be used unless FS is set to a negative value (-1.0) in the *CONTACT section. These frictional coefficients apply only to contact types: SINGLE_SURFACE, AUTOMATIC_GENERAL, AUTOMATIC_SINGLE_SURFACE, AUTOMATIC_NODES_TO_..., AUTOMATIC_SURFACE_..., AUTOMATIC_ONE_WAY_..., and ERODING_SINGLE_SURFACE. Default values are input via *CONTROL_CONTACT input.

Optional	1	2	3	4	5	6	7	8
Variable	FS	FD	DC	VC	OPTT	SFT	SSF	
Type	F	F	F	F	F	F	F	

The material ID, thickness, and material angle for each through-thickness integration point of a composite shell are provided below (up to two integration points per card). The integration point data should be given sequentially starting with the bottommost integration point. The total number of integration points is determined by the number of entries on these cards. The next "*" card terminates this input.

Card 3	1	2	3	4	5	6	7	8
Variable	MID1	THICK1	B1	TMID1	MID2	THICK2	B2	TMID2
Type	I	F	F	I	I	F	F	I

Cards 4…	1	2	3	4	5	6	7	8
Variable	MID3	THICK3	B3	TMID3	Etc.			
Type	I	F	F	I	I	F	F	I

VARIABLE	DESCRIPTION
HEADING	Heading for the part
PID	Part ID

VARIABLE	DESCRIPTION
ELFORM	Element formulation options for thin shells: EQ.1: Hughes-Liu, EQ.2: Belytschko-Tsay, EQ.3: BCIZ triangular shell, EQ.4: C^0 triangular shell, EQ.6: S/R Hughes-Liu, EQ.7: S/R co-rotational Hughes-Liu, EQ.8: Belytschko-Leviathan shell, EQ.9: Fully integrated Belytschko-Tsay membrane, EQ.10: Belytschko-Wong-Chiang, EQ.11: Fast (co-rotational) Hughes-Liu, EQ.16: Fully integrated shell element (very fast), Element formulation options for thick shells: EQ.1: one point reduced integration, EQ.2: selective reduced 2 x 2 in plane integration, EQ.3: assumed strain 2 x 2 in plane integration, EQ.5: assumed strain reduced integration.
SHRF	Shear correction factor which scales the transverse shear stress. The shell formulations in LS-DYNA, with the exception of the BCIZ and DK elements, are based on a first order shear deformation theory that yields constant transverse shear strains which violates the condition of zero traction on the top and bottom surfaces of the shell. The shear correction factor is an attempt to compensate for this error.
NLOC	Location of reference surface for three dimensional shell elements. If nonzero, the offset distance from the plane of the nodal points to the reference surface of the shell in the direction of the shell normal vector is a value $offset = -0.50 \times NLOC \times (average\ shell\ thickness)$. This offset is not considered in the contact subroutines unless CNTCO is set to 1 in *CONTROL_SHELL. Alternatively, the offset can be specified by using the OFFSET option in the *ELEMENT_SHELL input section. EQ. 1.0: top surface, EQ. 0.0: mid-surface (default), EQ.-1.0: bottom surface.
MAREA	Non-structural mass per unit area. This is additional mass which comes from materials such as carpeting. This mass is not directly included in the time step calculation.
HGID	Hourglass/bulk viscosity identification defined in the *HOURGLASS Section: EQ.0: default values are used.
ADPOPT	Indicate if this part is adapted or not. Also see, *CONTROL_ ADAPTIVITY:

EQ.0: no adaptivity,
EQ.1: H-adaptive for 3-D shells.

VARIABLE	DESCRIPTION
ITHELFM	Thermal shell formulation EQ.0: Default is governed by TSHELL on *CONTROL_SHELL EQ.1: Thick thermal shell EQ.2: Thin thermal shell
FS	Static coefficient of friction. The functional coefficient is assumed to be dependent on the relative velocity v_{rel} of the surfaces in contact $\mu_c = FD + (FS - FD)e^{-DC\cdot\lvert v_{rel}\rvert}$.
FD	Dynamic coefficient of friction. The functional coefficient is assumed to be dependent on the relative velocity v_{rel} of the surfaces in contact $\mu_c = FD + (FS - FD)e^{-DC\cdot\lvert v_{rel}\rvert}$.
DC	Exponential decay coefficient. The functional coefficient is assumed to be dependent on the relative velocity v_{rel} of the surfaces in contact $\mu_c = FD + (FS - FD)e^{-DC\cdot\lvert v_{rel}\rvert}$.
VC	Coefficient for viscous friction. This is necessary to limit the friction force to a maximum. A limiting force is computed $F_{lim} = VC \cdot A_{cont}$. A_{cont} being the area of the segment contacted by the node in contact. The suggested value for *VC* is to use the yield stress in shear $VC = \frac{\sigma_o}{\sqrt{3}}$ where σ_o is the yield stress of the contacted material.
OPTT	Optional contact thickness. This applies to shells only.
SFT	Optional thickness scale factor for PART ID in automatic contact (scales true thickness). This option applies only to contact with shell elements. True thickness is the element thickness of the shell elements.
SSF	Scale factor on default slave penalty stiffness for this PART ID whenever it appears in the contact definition. If zero, SSF is taken as unity.
MIDi	Material ID of integration point *i*, see *MAT_…. Section.
THICKi	Thickness of integration point *i*.
Bi	Material angle of integration point *i*.
TMIDi	Thermal material ID of integration point *i*

***PART_MODES**

Purpose: Define mode shapes for a flexible rigid body. Currently, flexible rigid bodies cannot share nodes with other flexible rigid bodies or rigid bodies; however, interconnections to other flexible rigid bodies or to rigid bodies can use the penalty joint option. The flexible rigid bodies are not implemented with the Lagrange multiplier joint option. The deformations are modeled using the modes shapes obtained experimentally or in a finite element analysis, e.g., NASTRAN.pch file or an LSTC eigout file. These modes should include both constraint and attachment modes. For stress recovery in flexible rigid bodies, use of linear element formulations is recommended. A lump mass matrix is assumed in the implementation. Also see the keyword control card: *CONTROL_RIGID.

Card 1

1	2	3	4	5	6	7	8
PID	NMFB	FORM	ANSID	FORMAT	KMFLAG	NUPDF	SIGREC
I	I	I	I	I	I	I	

Card 2

1
FILENAME
C
none

Define the following cards if and only if KMFLAG=1. Use as many cards as necessary to identify the NMFB kept modes. After NMFB modes are defined no further input is expected.

Cards 3...

1	2	3	4	5	6	7	8
MODE1	MODE2	MODE3	MODE4	MODE5	MODE6	MODE7	MODE8
I	I	I	I	I	I	I	I
none	nont	none	nont	none	nont	none	nont

Read optional modal damping cards here. A keyword card (with a "*" in column 1) terminates this input.

Card	1	2	3	4	5	6	7	8
Variable	MSTART	MSTOP	DAMPF					
Type	I	I	F					
Default	none	none	none					

VARIABLE	DESCRIPTION
PID	Part identification. This part must be a rigid body.
NMFB	Number of kept modes in flexible body. The number of modes in the file, FILENAME, must equal or exceed NMFB. If KMFLAG=0 the first NMFB modes in the file are used.
FORM	Flexible body formulation. See remark 5 below. EQ.0: exact EQ.1: fast
ANSID	Attachment node set ID (optional).
FORMAT	Input format of modal information: EQ.0: NASTRAN.pch file. EQ.1: (not supported) EQ.2: NASTRAN.pch file (LS-DYNA binary version). The binary version of this file is automatically created if a NASTRAN.pch file is read. The name of the binary file is the name of the NASTRAN.pch file but with ".bin" appended. The binary file is smaller and can be read much faster. EQ.3: LS-DYNA d3eigv binary eigenvalue database (see *CONTROL_IMPLICIT_EIGENVALUE). EQ.4: LS-DYNA d3mode binary constraint/attachment mode database (see *CONTROL_IMPLICIT_MODE). EQ.5: Both d3eigv and d3mode databases are input. Database names must be "d3eigv" and "d3mode", and FILENAME below is ignored. NMFB above gives the total number of modes in both databases.
KMFLAG	Kept mode flag. Selects method for identifying modes to keep. EQ.0: the first NMFB modes in the file, FILENAME, are used. EQ.1: define NMFB kept modes with additional input.

VARIABLE	DESCRIPTION
NUPDF	Nodal update flag. If active, an attachment node set, ANSID, must be defined. EQ.0: all nodes of the rigid part are updated each cycle. EQ.1: only attachment nodes are fully updated. All nodes in the body are output based on the rigid body motion without the addition of the modal displacements. For maximum benefit an attachment node set can also be defined with the PART_ATTACHMENT_ NODES option. The same attachment node set ID should be used here.
SIGREC	Stress recovery flag. EQ.0: no stress recovery EQ.1: stress recovery only EQ.2: stress recovery and then set the recovery stress as initial stress when switching to flexible body. (Shell formulations 16, 18, 20, 21 and Solid formulation 2) EQ.3: only for shell elform=16, the recovery stress is based on elform=21, and then set the recovery stress as initial stress for elform=16 when switching to flexible body.
FILENAME	The path and name of a file which contains the modes for this rigid body.
MODEn	Keep normal mode, MODEn.
MSTART	First mode for damping, $(1 \le MSTART \le NMFB)$.
MSTOP	Last mode for damping, MSTOP, $(1 \le MSTOP \le NMFB)$. All modes between MSTART and MSTOP inclusive are subject to the same modal damping coefficient, DAMPF.
DAMPF	Modal damping coefficient, ζ.

Remarks:

1. The format of the file which contains the normal modes follows the file formats of NASTRAN output for modal information.

2. The mode set typically combines both normal modes and attachment modes. The eigenvalues for the attachment modes are computed from the stiffness and mass matrices.

3. The part ID specified must be either a single rigid body or a master rigid body (see *CONSTRAINED_RIGID_BODIES) which can be made up of many rigid parts.

4. The modal damping is defined by the modal damping coefficient ζ., where a value of 1.0 equals critical damping. For a one degree of freedom model system, the relationship

between the damping and the damping coefficient is $c = 2\zeta\omega_n m$, where c is the damping, m is the mass, and ω_n is the natural frequency, $\sqrt{k/m}$.

5. There are two formulation options. The first is a formulation that contains all the terms of the flexible body equations, and its cost grows approximately as the square of the number of modes. The second formulation ignores most of the second order terms appearing in the exact equations and its cost grows linearly with the number of modes. Users are responsible for determining which formulation is appropriate for their problems. In general, if the angular velocities are small and if the deflections are small with respect to the geometry of the system it is safe to use the second (faster) formulation.

***PART_SENSOR**

Purpose: Activate and deactivate parts, based on sensor defined in ELEMENT_SEATBELT_ SENSOR. This option applies to discrete beam element only.

Define one card. Card Format (3I10)

Card 1	1	2	3	4	5	6	7	8
Variable	PID	SIDA	ACTIVE					
Type	I	I	I					
Default	0	0	0					

VARIABLE	DESCRIPTION
PID	Part ID, which is controlled by sensor
SIDA	Sensor ID to activate or deactivate part.
ACTIVE	Flag. If zero, the part is active from time zero until a signal is received by the part to deactivate. If one, the part is inactive from time zero and becomes active when a signal is received by the part to activate. The history variables for inactive parts are initialized at time zero.

*PART_MOVE

Purpose: Translate a part by an incremental displacement in either a local or a global coordinate system. This option currently applies to parts defined either by shell and solid elements. All nodal points of the given part ID are moved. Care must be observed since parts that share boundary nodes with the part being moved must also be moved to avoid severe mesh distortions.

Define one card. Card Format (I8,3E16.0)

Card 1	1	2	3	4	5	6	7	8	9	10
Variable	PID	XMOV		YMOV		ZMOV		CID		
Type	I	F		F		F		I		
Default	none	0.		0.		0.		0		

VARIABLE	DESCRIPTION
PID	Part identification
XMOV	Move shell/solid part ID, PID, in the x-direction by the incremental distance, XMOV.
YMOV	Move shell/solid part ID, PID, in the y-direction by the incremental distance, YMOV.
ZMOV	Move shell/solid part ID, PID, in the z-direction by the incremental distance, ZMOV.
CID	Coordinate system ID to define incremental displacement in local coordinate system. All displacements, XMOV, YMOV, and ZMOV, are with respect to CID. EQ.0: global

*PERTURBATION

The keyword *PERTURBATION provides a way of defining deviations from the designed structure such as buckling imperfections. These perturbations can be viewed in LS-PREPOST as user-defined fringe plots. Available options are:

***PERTURBATION_MATERIAL**

***PERTURBATION_NODE**

***PERTURBATION_SHELL_THICKNESS**

PERTURBATION_OPTION*

Available options are:

NODE

SHELL_THICKNESS

MATERIAL

Purpose: Define a perturbation (stochastic field) over the whole model or a portion of the model, typically to trigger an instability. The NODE option modify the three dimensional coordinates for the whole model or a node set. For the SHELL_THICKNESS option the shell thicknesses are perturbed for the whole model or a shell set. The MATERIAL option perturbs a material parameter value for all the elements associated with the material.

Define for the MATERIAL option. Required

Card 1 1 2 3 4 5 6 7 8

Variable	TYPE	PID	SCL	CMP	ICOORD	CID		
Type	I	I	F	I	I	I		
Default	1	0	1.0	7	0	0		

Define for the NODE option. Required

Card 1 1 2 3 4 5 6 7 8

Variable	TYPE	NID	SCL	CMP	ICOORD	CID		
Type	I	I	F	I	I	I		
Default	1	0	1.0	7	0	0		

Define for the SHELL_THICKNESS option. Required

Card 1

Variable	TYPE	EID	SCL	ICOORD	CID			
Type	I	I	F	I	I			
Default	1	0	1.0	0	0			

Define if TYPE is 1 (harmonic field). One or many definitions.

Card 2 1 2 3 4 5 6 7 8

Variable	AMPL	XWL	XOFF	YWL	YOFF	ZWL	ZOFF	
Type	F	F	F	F	F	F	F	
Default	1.0	0.0	0.0	0.0	0.0	0.0	0.0	

Define if TYPE is 2 (fade field). One definition only.

Card 2

Variable	FADE							
Type	F							
Default	1.0							

Define if TYPE is 3 (file field). One definition only.

Card 2

Variable	FNAME							
Type	A							
Default	None							

Define if TYPE is 4 (spectral field). One definition only.

Card 2	1	2	3	4	5	6	7	8
Variable	CSTYPE	ELLIP1	ELLIP2	RND				
Type	I	F	F	I				
Default	None	1.0	1.0	0				

Define if TYPE is 4 (spectral field). One, two, or three definitions, depending on CSTYPE.

Card 3

Variable	CFTYPE	CFC1	CFC2	CFC3				
Type	I	F	F	F				
Default	None	1.0	1.0	1.0				

VARIABLE	DESCRIPTION
TYPE	Type of perturbation EQ.1: Harmonic Field EQ.2: Fade out all perturbations at this node set EQ.3: Read perturbations from a file EQ.4: Spectral field

PID	Part ID.
NID	Node set ID. Specify 0 to perturb all the nodes in the model.
EID	Element set ID. Specify 0 to perturb all the elements in the model.
SCL	Scale factor
CMP	Component. For the NODE option, these are given below. For the MATERIAL option, see the description of the material. EQ.1: x coordinate EQ.2: y coordinate EQ.3: z coordinate EQ.4: x and y coordinate EQ.5: y and z coordinate EQ.6: z and x coordinate EQ.7: x, y, and z coordinate
ICOORD	Coordinate system to use; see remarks 7, 8 and 9 EQ.0: Global Cartesian EQ.1: Cartesian EQ.2: Cylindrical (computed and applied) EQ.3: Spherical (computed and applied) EQ.-2: Computed in cartesian but applied in cylindrical EQ.-3 Computed in cartesian but applied in spherical
CID	Coordinate system ID, see *DEFINE_COORDINATE_NODES
AMPL	Amplitude of the harmonic perturbation
XWL	x wavelength of the harmonic field
XOFF	x offset of harmonic field
YWL	y wavelength of the harmonic field
YOFF	y offset of harmonic field
ZWL	z wavelength of the harmonic field
ZOFF	z offset of harmonic field
FADE	Parameter controlling the distance over which all *PERTURBATION_NODE are faded to zero
FNAME	Name of file containing the perturbation definitions
CSTYPE	Correlation structure EQ.1: 3D isotropic. The X, Y and Z correlations are described using one correlation function. Define CFC1.

EQ.2: 3D product. The X, Y and Z correlations are described using a correlation function each. Define CFC1, CFC2 and CFC3.
EQ.3: 2D isotropic. A correlation function describes the X correlation while the YZ isotropic relationship is described using another correlation function. Define CFC1 and CFC2.
EQ.4: 2D isotropic. The XZ isotropic relationship is described using a correlation function, while another correlation function describes the Y correlation while. Define CFC1 and CFC2.
EQ.5: 2D isotropic. The XY isotropic relationship is described using a correlation function, while another correlation function describes the Z correlation while. Define CFC1 and CFC2.
EQ.6: 3D elliptic. Define CSE1, CSE2 and CFC1.
EQ.7: 2D elliptic. A correlation function describes the X correlation while the YZ elliptic relationship is described using another correlation function. Define CSE1 and CFC1.
EQ.8: 2D elliptic. A correlation function describes the Y correlation while the ZX elliptic relationship is described using another correlation function. Define CSE1 and CFC1.
EQ.9: 2D elliptic. The XY elliptic relationship is described using a correlation function, while another correlation function describes the Z correlation while. Define CSE1 and CFC1.

ELLIP1	Elliptic constant for 2D and 3D elliptic fields
ELLIP2	Elliptic constant for 3D elliptic field
RND	Seed for random number generator. EQ.0: LS-DYNA will generate a random seed GT.0: Value to be used as seed
CFTYPE	Correlation function EQ.1: Gaussian EQ.2: Exponential EQ.3: Exponential Cosine EQ.4: Rational EQ.5: Linear
CFC*i*	Correlation function constant *i*

Remarks:

1. The perturbation can be viewed in LS-PREPOST. For the NODE option, LS-DYNA creates files named *pert_node_x/y/z/res*, which can be viewed as user-defined fringe plots. For the SHELL_THICKNESS and MATERIAL options, the files are named *pert_shell_thickness* and *pert_mat* respectively.

2. Perturbations specified using separate *PERTURBATION cards are created separately and then added together. This is true as well for special cases such as CMP=7 in which case the x, y and z fields are created separately and added together afterwards, which can result in an absolute amplitude greater than specified using AMPL or SCL.

3. The harmonic perturbation is

$$p_{CMP}(x,y,z) = SCL * AMPL\left[\sin\left(2\pi\frac{x+XOFF}{XWL}\right)+\sin\left(2\pi\frac{y+YOFF}{YWL}\right)+\sin\left(2\pi\frac{z+ZOFF}{ZWL}\right)\right]$$

Note that the harmonic perturbations can sum to values greater than *SCL*AMPL*.

4. The fade perturbation is $p'(x,y,z) = SCL\left(1-\frac{1}{e^{\mathrm{FADE}\,x'}}\right)p(x,y,z)$ with x' the shortest distance to a node in the node set specified and FADE the parameter controlling the sharpness of the fade perturbation.

5. The file FNAME must contain the perturbation in the LS-DYNA keyword format. This file can be created from the d3plot results using the LS-PREPOST Output capability. The data must be arranged into two columns with the first column being the node ids. Lines starting with the character $ will be ignored.

6. The correlation functions are defined as follows:

Gaussian: $B(t) = e^{-(at)^2}$

Exponential: $B(t) = e^{-|at|^b}$

Exponent and Cosine: $B(t) = e^{-|at|}\cos(bt)$

Rational: $B(t) = (1+|at|^b)^{-c}$

Piecewise Linear: $B(t) = (1-|at|)\chi(1-|at|)$

With χ the Heaviside step function and a, b and c corresponding to CFC1, CFC2 and CFC3.

7. For the cylindrical coordinate system option (ICOORD=2), the default is to use the global coordinate system for the location of the cylindrical part, with the base of the cylinder located at the origin, and the global z-axis aligned with the cylinder axis. For cylindrical parts not located at the global origin, define a coordinate system (numbered CID) using *DEFINE_COORDINATE_NODES by selecting any three nodes on the base of the cylinder in a clockwise direction (resulting in the local z-axis to be aligned with the cylinder).

8. For the spherical coordinate system (ICOORD=3), the coordinates are the radius, zenith angle ($0 - \pi$), and the azimuth angle ($0 - 2\pi$). The default is to use the global coordinate system with the zenith measured from the z-axis and the azimuth measured from the x-axis in the xy-plane. For spherical parts not located at the global origin, define a coordinate system using

*DEFINE_COORDINATE_NODES by selecting any three nodes as follows: the first node is the center of the sphere, the second specifies the x-axis of the coordinate system, while the third point specifies the plane containing the new y-axis. The z-axis will be normal to this plane.

9. It is possible to compute the perturbations in a Cartesian coordinate system, but to apply them in a cylindrical or spherical coordinate system (ICOORD=-2,-3). This is the natural method of doing say a radial perturbation of a sphere using a spectral perturbation field. We expect that computing the perturbation in the spherical coordinate system should be rare (ICOORD=3). Computing a perturbation in a cylindrical coordinate system should be common though; for example, a circumferential harmonic perturbation.

10. Only *MAT238 (*MAT_PERT_PIECEWISE_LINEAR_PLASTICITY) and solid elements in an explicit analysis can be perturbed using *PERTURBATION_MATERIAL. See the documentation of this material for allowable components. Only one part per model can be perturbed. The material perturbation is applied on an element-wise basis as $\text{value}_{new} = (1+p)\text{value}_{base}$ with p the perturbation value for the element as given in the file *pert_mat*. Values of p less than -1 are accordingly illegal, because the material behavior is not defined.

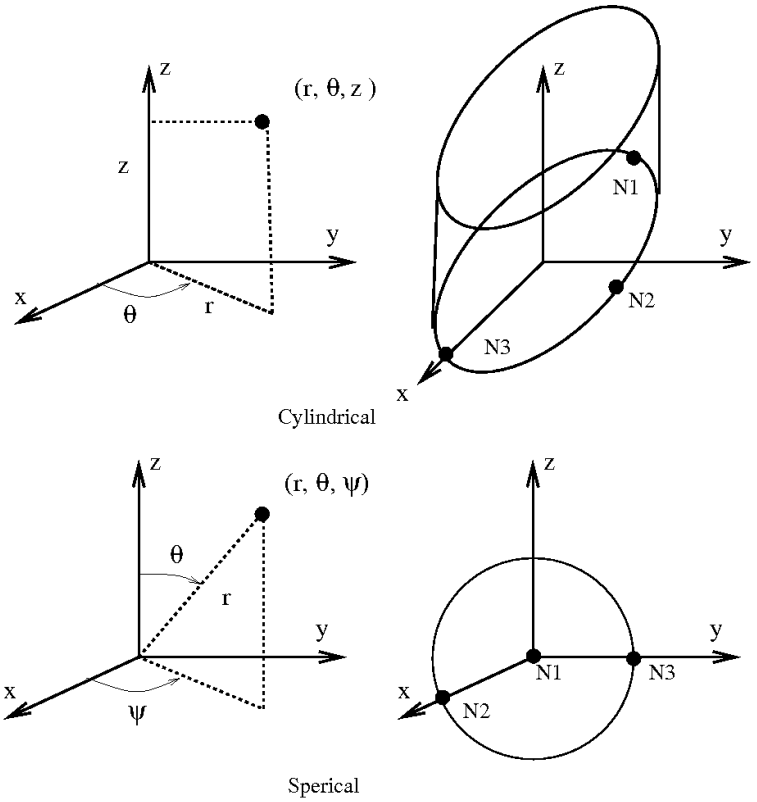


Figure 1 Creating coordinate systems for *PERTURBATION. N1, N2 and N3 are the nodes that must be specified using *DEFINE_COORDINATE_NODES to obtain the coordinate system shown.

*RAIL

Two keywords are defined in this section.

***RAIL_TRACK**

***RAIL_TRAIN**

*RAIL_TRACK

Purpose: Wheel-rail contact algorithm intended for railway applications but can also be used for other purposes. The wheel nodes (defined on *RAIL_TRAIN) represent the contact patch between wheel and rail. A penalty method is used to constrain the wheel nodes to slide along the track. A track consists of two rails, each of which is defined by a set of beam elements.

Card 1 of 2

Card 1	1	2	3	4	5	6	7	8
Variable	ID	BSETID1	NORGN1	LCUR1	OSET1	SF1	GA1	
Type	I	I	I	I	F	F	F	
Default	none	None	None	None	0.0	1.0	0.0	

Card 2

Variable	BLANK	BSETID2	NORGN2	LCUR2	OSET2	SF2	GA2	
Type	-	I	I	I	F	F	F	
Default	-	None	None	None	0.0	1.0	0.0	

VARIABLE	DESCRIPTION
ID	Track ID
BSETID1,2	Beam set ID for rails 1 and 2 containing all beam elements that make up the rail, see *SET_BEAM.
NORGN1,2	Reference node at one end of each rail, used as the origin for the roughness curve. The train will move in a direction away from this node.
LCUR1,2	Load curve ID (see *DEFINE_CURVE) defining track roughness (vertical displacement from line of beam elements) of the rail as a function of distance from the reference node NORIGIN. Distance from reference node on x-axis of curve, roughness on y-axis. Default: no roughness.

VARIABLE	DESCRIPTION
OSET1,2	Origin of curve LCUR is shifted by distance OSET from the reference node.
SF1,2	Roughness values are scaled by SF. Default: 1.0.
GA1,2	Shear stiffness of rail per unit length (used to calculate local rail shear deformation within each beam element). GA = shear modulus x cross-sectional area. Default: local shear deformation is ignored.

Remarks:

*RAIL_TRACK and *RAIL_TRAIN were written by Arup to represent wheel-rail contact. They have been used to generate loading on models of bridges for vibration predictions, stress calculations and for estimating accelerations experienced by passengers. Other non-railway uses are possible: the algorithm causes the "train" nodes to follow the line defined by the "rail" beam elements and transfers forces between them. In some cases (especially vibration modeling), double precision versions of LS-DYNA may give superior results because of the small relative deflections between wheel and rail.

Track modeling

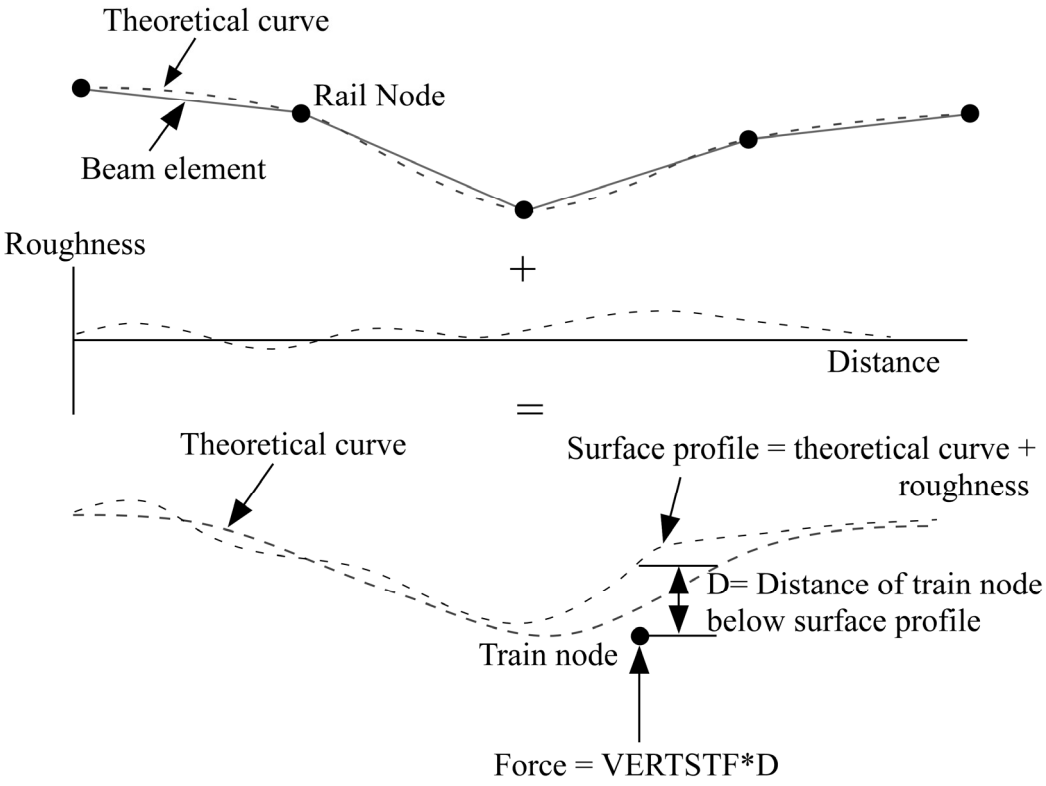
The rails of the track should be modeled by two parallel lines of beam elements. The track can be curved or straight and the rails can be modeled as deformable or rigid. If required, rail pads, sleepers and ballast may also be modeled – typically with spring, damper and beam elements. It is also possible to use this algorithm to control the motion of simple road vehicle models: beam element "rails" made of null material can be embedded in the road surface. It is recommended that the mesh size of the two rails should be similar: LS-DYNA calculates a local coordinate system for each train node based on the alignment of the currently contacted beam element and the nearest node on the other rail.

Because wheel-rail contact stiffness is generally very high, and wheel masses are large, small deviations from a straight line or smooth curve can lead to large transient forces. It is recommended that great care be taken in generating and checking the geometry for the track, especially where the track is curved. Some pre-processors write the coordinates with insufficient precision to the LS-DYNA input file, and this can cause unintended roughness in the geometry. For the same reason, if the line of the track were taken as straight between nodes, spurious forces would be generated when the wheel passes from one rail element to the next. This is avoided because the *RAIL algorithm calculates a theoretical curved centerline for the rail element to achieve continuity of slope from one element to the next. Where the length of the rail elements is similar to or shorter than the maximum section dimension, shear deformation may be significant and it is possible to include this in the theoretical centerline calculation to further reduce spurious forces at the element boundaries (inputs GA1, GA2).

Roughness (small deviations in the vertical profile from a perfect straight line) does exist in real life and is a principal source of vibration. *RAIL allows the roughness to be modeled by a load curve giving the vertical deviation (in length units) of the rail surface from the theoretical centerline of the beam elements as a function of distance along the track from the origin node of the rail. The roughness curve is optional. Ideally, roughness profiles measured from both rails of

the same piece of track should be used so that the relationship between bump and roll modes is correctly captured.

Whether roughness is included or not, it is important to select as the origin nodes (NORIGIN1 and NORIGIN2) the nodes at the end of the rails away from which the train will be traveling. The train can start at any point along the rails but must travel away from the origin nodes.



Train modeling

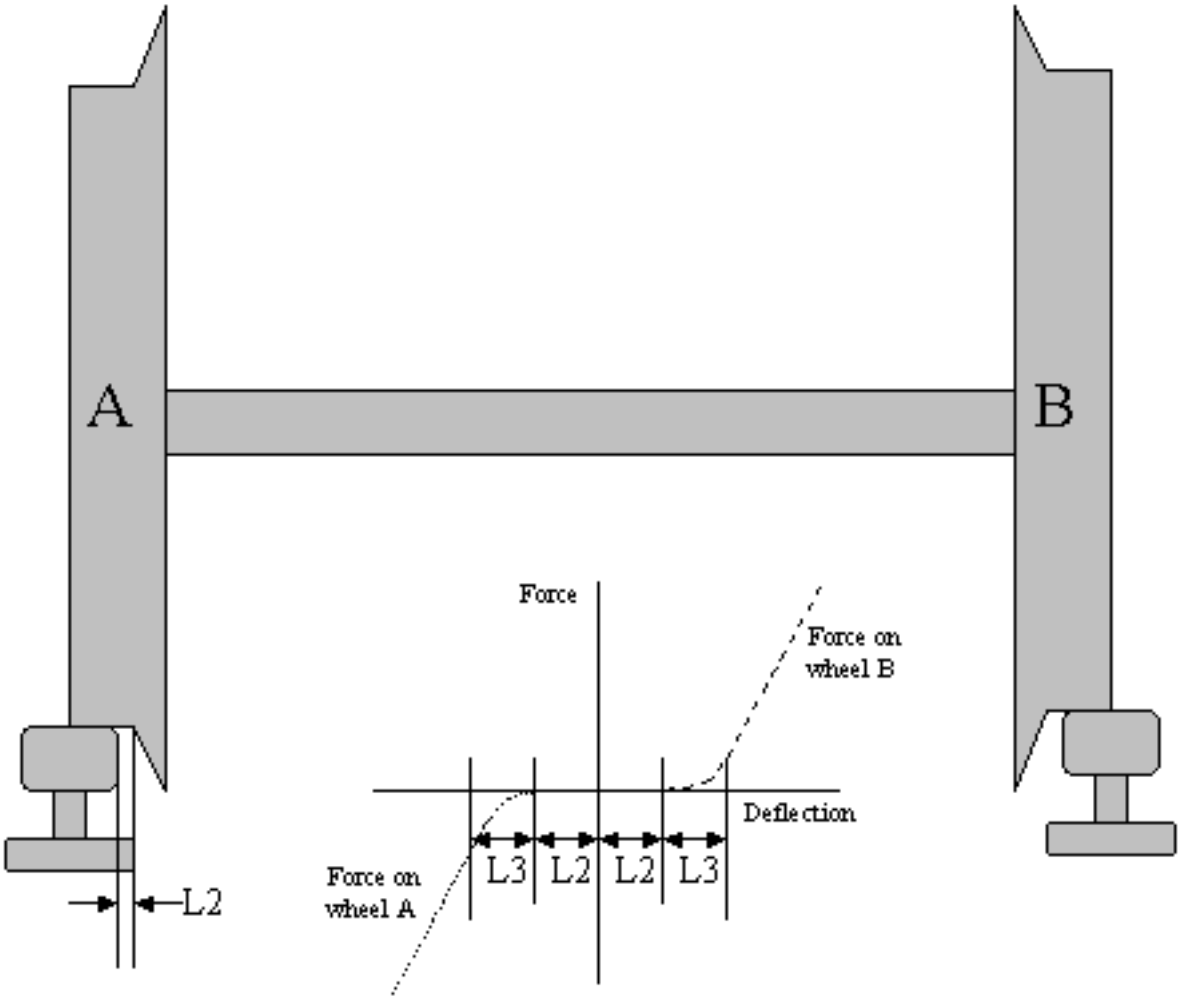
The vehicle models are typically modeled using spring, damper and rigid elements, or simply a point mass at each wheel position. Each node in the set referred to on *RAIL_TRAIN represents the contact patch of one wheel (note: not the center of the wheel). These nodes should be initially on or near the line defined by either of the two rails. LS-DYNA will move the train nodes initially onto the rails to achieve the correct initial wheel-rail forces. If the results are viewed with magnified displacements, the initial movements can appear surprising.

Wheel roughness input is available. This will be applied in addition to track roughness. The input curve must continue for the total rolled distance – it is not assumed to repeat with each wheel rotation. This is to avoid problems associated with ensuring continuity between the start

and end of the profile around the wheel circumference, especially since the profiles might be generated from roughness spectra rather than taken directly from measured data.

Wheel-rail interface

The wheel-rail interface model is a simple penalty function designed to ensure that the train nodes follow the line of the track. It does not attempt to account for the shape of the rail profile. Vertical and lateral loads are treated independently. For this reason, the algorithm is not suitable for rail vehicle dynamics calculations.



Wheel-rail contact stiffness is input on *RAIL_TRAIN. For vertical loads, a linear force-deflection relationship is assumed in compression; no tensile force is generated (this corresponds to the train losing contact with the rail). Typical contact stiffness is 2MN/mm. Lateral deflections away from the theoretical centerline of the rail beams are also penalized by a linear force-deflection relationship. The lateral force is applied only to wheels on the side towards which the train has displaced (corresponding to wheel flanges that run inside the rails). Optionally, a "gap" can be defined (input parameter L2) such that the wheel set can drift laterally by L2 length units before any lateral force is generated. A further option is to allow smooth transition between "gap" and "contact" by means of a transition distance (input parameter L3). Generally, with straight tracks a simple linear stiffness is sufficient. With curved tracks, a

reasonable gap and transition distance should be defined to avoid unrealistic forces being generated in response to small inaccuracies in the distance between the rails. Gravity loading is expected, in order to maintain contact between rail and wheel. This is normally applied by an initial phase of dynamic relaxation. To help achieve convergence quickly, or in some cases avoid the need for dynamic relaxation altogether, the initial force expected on each train node can be input (parameter FINIT on *RAIL_TRAIN). LS-DYNA positions the nodes initially such that the vertical contact force will be FINIT at each node. If the suspension of the rail vehicles is modeled, it is recommended that the input includes carefully calculated precompression of the spring elements; if this is not done, achieving initial equilibrium under gravity loading can be very time consuming.

The *RAIL algorithm ensures that the train follows the rails, but does not provide forward motion. This is generally applied using *INITIAL_VELOCITY, or for straight tracks, *BOUNDARY_PRESCRIBED_MOTION.

Output
LS-DYNA generates an additional ASCII output file **train_force_*n***, where ***n*** is an integer updated to avoid overwriting any existing files. The file contains the forces on each train node, output at the same time intervals as the binary time history file (DT on *DATABASE_BINARY_D3THDT).

Checking
It is recommended that track and train models be tested separately before adding the *RAIL cards. Check that the models respond stably to impulse forces and that they achieve equilibrium under gravity loading. The majority of problems we have encountered have been due to unstable behavior of train or track. Often, these are first detected by the *RAIL algorithm and an error message will result.

*RAIL_TRAIN

Purpose: Define train properties. A train is defined by a set of nodes in contact with a rail defined by *RAIL_TRACK.

Card 1 of 2

Card 1

Variable	ID	NSETID	(omit)	FINIT	(omit)	TRID	LCUR	OFFS
Type	I	I	F	F	F	I	I	F
Default	none	None	0.0	0.0	0.0	0	None	0.0

Card 2

Variable	VERTSTF	LATSTF	V2	V3	L2	L3		
Type	F	F	F	F	F	F		
Default	0.0	0.0	0.0	0.0	0.0	0.0		

VARIABLE	DESCRIPTION
ID	Train ID
NSETID	Node set ID containing all nodes that are in contact with rails.
(omit)	Unused variable – leave blank.
FINIT	Estimate of initial vertical force on each wheel (optional) – speeds up the process of initial settling down under gravity loading.
(omit)	Unused variable – leave blank.
TRID	ID of track for this train, see *RAIL_TRACK.
LCUR	Load curve ID (see *DEFINE_CURVE) containing wheel roughness (distance of wheel surface away from perfect circle) vs. distance traveled. The curve does not repeat with each rotation of the wheel – the last point should be at a greater distance than the train is expected to travel. Default: no wheel roughness.

VARIABLE	DESCRIPTION
OFFS	Offset distance used to generate different roughness curves for each wheel from the roughness curve LCUR. The curve is offset on the x-axis by a different whole number multiple of OFFS for each wheel.
VERTSTF	Vertical stiffness of rail contact.
LATSTF	Lateral stiffness of rail contact.
V2,V3	Unused variables – leave blank.
L2	Lateral clearance from rail to wheel rim. Lateral force is applied to a wheel only when it has moved more than L2 away from the other rail, i.e. the wheel rims are assumed to be near the inner face of the rail.
L3	Further lateral distance before full lateral stiffness applies (force-deflection curve follows a parabola up to this point).

*RIGIDWALL

Two keywords are used in this section to define rigid surfaces:

***RIGIDWALL_GEOMETRIC**_*OPTION*_{*OPTION*}_{*OPTION*}}_{*OPTION*}

***RIGIDWALL_PLANAR**_{*OPTION*}_{*OPTION*}_{*OPTION*}

The RIGIDWALL option provides a simple way of treating contact between a rigid surface and nodal points of a deformable body, called slave nodes. Slave nodes which belong to rigid parts are not, in general, checked for contact with only one exception. The RIGIDWALL_PLANAR option may be used with nodal points of rigid bodies if the planar wall defined by this option is fixed in space and the RWPNAL parameter is set to a positive nonzero value on the control card, *CONTROL_CONTACT.

When the rigid wall defined in this section moves with a prescribed motion, the equations of rigid body mechanics are not involved. For a general rigid body treatment with arbitrary surfaces and motion, refer to the *CONTACT_ENTITY definition. The *CONTACT_ENTITY option is for treating contact between rigid and deformable surfaces only.

***RIGIDWALL_GEOMETRIC**_*OPTION*_{*OPTION*}_{*OPTION*}}_{*OPTION*}

Available options include:

FLAT

PRISM

CYLINDER

SPHERE

If prescribed motion is desired an additional option is available:

MOTION

One of the shape types **[FLAT, PRISM, CYLINDER, SPHERE]** must be specified, followed by the optional definition of **MOTION**, both on the same line with ***RIGIDWALL_ GEOMETRIC.** If an ID number is specified the additional option is available:

ID

If active, the ID card is the first card following the keyword. To view the rigid wall, the option:

DISPLAY

is available. With this option a rigid body is automatically defined which represents the shape, the physical position of the wall, and follows the walls motion if the MOTION option is active. Additional input is optional if DISPLAY is active.

Purpose: Define a rigid wall with an analytically described form. Four forms are possible. A prescribed motion is optional. For general rigid bodies with arbitrary surfaces and motion, refer to the *CONTACT_ENTITY definition. This option is for treating contact between rigid and deformable surfaces only.

The following card is read if and only if the ID option is specified.

Optional 1 2-8

Variable	RWID	HEADING
Type	I	A70

The heading is picked up by some of the peripheral LS-DYNA codes to aid in post-processing.

VARIABLE	DESCRIPTION
RWID	Rigid wall ID. This must be a unique number.
HEADING	Rigid wall descriptor. It is suggested that unique descriptions be used.

For GEOMETRIC options:

- Cards 1 and 2 are required for all geometric shapes.
- Card 3 is required, but is dependent upon which shape is specified.
- Optional Card A is required if MOTION is specified.

Card 1 - Required for all shape types

Card 1	1	2	3	4	5	6	7	8
Variable	NSID	NSIDEX	BOXID	BIRTH	DEATH			
Type	I	I	I	F	F			
Default	none	0	0	0.	1.0E+20			

VARIABLE	DESCRIPTION
NSID	Nodal set ID containing slave nodes, see *SET_NODE_*OPTION*: EQ.0: all nodes are slave to rigid wall.
NSIDEX	Nodal set ID containing nodes that exempted as slave nodes, see *SET_NODE_*OPTION*.
BOXID	If defined, only nodes in box are included as slave nodes to rigid wall.
BIRTH	Birth time of rigid wall. The time values of the load curves that control the motion of the wall are offset by the birth time.
DEATH	Death time of rigid wall. At this time the wall is deleted from the calculation. If dynamic relaxation is active at the beginning of the calculation and if BIRTH=0.0, the death time is ignored during the dynamic relaxation.

Card 2 - Required for all shape types.

Card 2	1	2	3	4	5	6	7	8
Variable	XT	YT	ZT	XH	YH	ZH	FRIC	
Type	F	F	F	F	F	F	F	
Default	0.	0.	0.	0.	0.	0.	0.	
Remarks								

VARIABLE	DESCRIPTION
XT	x-coordinate of tail of any outward drawn normal vector, **n**, originating on wall (tail) and terminating in space (head), see Figure 28.1.
YT	y-coordinate of tail of normal vector **n**
ZT	z-coordinate of tail of normal vector **n**
XH	x-coordinate of head of normal vector **n**
YH	y-coordinate of head of normal vector **n**
ZH	z-coordinate of head of normal vector **n**
FRIC	Interface friction: EQ.0.0: frictionless sliding after contact, EQ.1.0: stick condition after contact, 0.<FRIC<1.: Coulomb friction coefficient.

Card 3 - Required if FLAT is specified after the keyword.

A plane with a finite size or with an infinite size can be defined, see Figure 28.1. The vector **m** is computed as the vector cross product **n** X **l**. The origin, which is the tail of the normal vector, is the corner point of the finite size plane.

Card 3	1	2	3	4	5	6	7	8
Variable	XHEV	YHEV	ZHEV	LENL	LENM			
Type	F	F	F	F	F			
Default	0.	0.	0.	infinity	infinity			

VARIABLE	DESCRIPTION
XHEV	x-coordinate of head of edge vector **l**, see Figure 28.1.
YHEV	y-coordinate of head of edge vector **l**
ZHEV	z-coordinate of head of edge vector **l**
LENL	Length of **l** edge. A zero value defines an infinite size plane.
LENM	Length of **m** edge. A zero value defines an infinite size plane.

Card 3 - Required if PRISM is specified after the keyword.

The description of the definition of a plane with finite size is enhanced by an additional length in the direction negative to **n**, see Figure 28.1.

Card 3	1	2	3	4	5	6	7	8
Variable	XHEV	YHEV	ZHEV	LENL	LENM	LENP		
Type	F	F	F	F	F	F		
Default	none	0.	0.	infinity	infinity	infinity		

VARIABLE	DESCRIPTION
XHEV	x-coordinate of head of edge vector **l,** see Figure 28.1.
YHEV	y-coordinate of head of edge vector **l**
ZHEV	z-coordinate of head of edge vector **l**
LENL	Length of **l** edge. A zero value defines an infinite size plane.
LENM	Length of **m** edge. A zero value defines an infinite size plane.
LENP	Length of prism in the direction negative to **n**, see Figure 28.1.

Card 3 - Required if CYLINDER is specified after the keyword.

The tail of **n** specifies the top plane of the cylinder. The length is defined in the direction negative to **n**. See Figure 28.1.

Card 3	1	2	3	4	5	6	7	8
Variable	RADCYL	LENCYL						
Type	F	F						
Default	none	infinity						

VARIABLE	DESCRIPTION
RADCYL	Radius of cylinder
LENCYL	Length of cylinder, see Figure 28.1. Only if a value larger than zero is specified is a finite length assumed.

Card 3 - Required if SPHERE is specified after the keyword.

The center of the sphere is identical to the tail of **n**, see Figure 28.1.

Card 3	1	2	3	4	5	6	7	8
Variable	RADSPH							
Type	F							
Default	0.							

VARIABLE	DESCRIPTION
RADSPH	Radius of sphere

Optional Card A - Required if MOTION is specified after the keyword.

Optional Card A	1	2	3	4	5	6	7	8
Variable	LCID	OPT	VX	VY	VZ			
Type	I	I	F	F	F			
Default	none	none	none	none	none			

VARIABLE	DESCRIPTION
LCID	Stonewall motion curve number, see *DEFINE_CURVE.
OPT	Type of motion: EQ.0: velocity specified, EQ.1: displacement specified.
VX	x-direction cosine of velocity/displacement vector
VY	y-direction cosine of velocity/displacement vector
VZ	z-direction cosine of velocity/displacement vector

Optional Card B – Not required, but may be useful if the DISPLAY option is specified. If a "*" keyword is defined, this input is not read in and default values will be set.

Card 1	1	2	3	4	5	6	7	8
Variable	PID	RO	E	PR				
Type	I	I	I	F				
Default	none	0	0	0.				

VARIABLE	DESCRIPTION
PID	Unique part ID for moving geometric rigid wall. If zero, a part ID will be set that is larger than the maximum of all user defined part ID's.
RO	Density of rigid wall. The default is set to 1.0E-09.
E	Young's modulus. The default is set to 1.0E-04.
PR	Poisson's ratio. The default is set to 0.30.

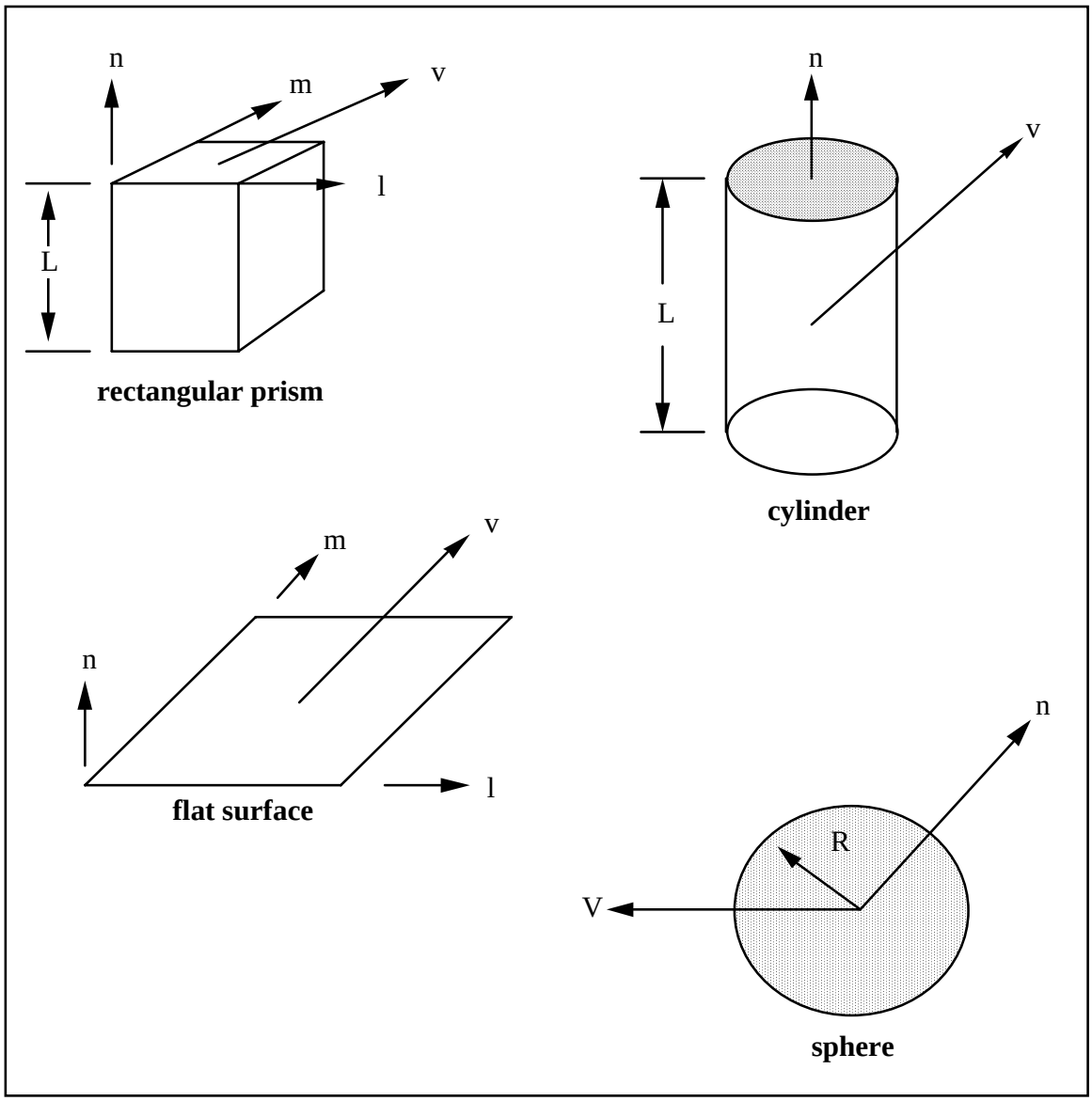


Figure 28.1. Vector **n** determines the orientation of the generalized stonewalls. For the prescribed motion options the wall can be moved in the direction **V** as shown.

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$  *RIGIDWALL_GEOMETRIC_SPHERE_MOTION
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Define a rigid sphere:
$     - with a radius of 8
$     - centered at (x,y,z) = (20,20,9)
$     - that moves in the negative z-direction with a specified displacement
$          given by a load curve (load curve: lcid = 5)
$     - which prevents all nodes within a specified box from penetrating the
$          sphere (box number: boxid = 3), these nodes can slide on the sphere
$          without friction
$
*RIGIDWALL_GEOMETRIC_SPHERE_MOTION
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$     nsid    nsidex     boxid
                             3
$
$       xt        yt        zt        xh        yh        zh      fric
       20.0      20.0       9.0      20.0      20.0       0.0       0.0
$
$   radsph
        8.0
$
$     lcid       opt        vx        vy        vz
          5         1       0.0       0.0      -1.0
$
$
*DEFINE_BOX
$    boxid       xmn       xmx       ymn       ymx       zmn       zmx
          3       0.0      40.0       0.0      40.0      -1.0       1.0
$
$
*DEFINE_CURVE
$     lcid      sidr      scla      sclo      offa      offo
          5
$           abscissa           ordinate
                0.0                0.0
             0.0005               15.0
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

***RIGIDWALL_PLANAR**_{*OPTION*}_{*OPTION*}_{*OPTION*}

Available options include:

<BLANK>

ORTHO

FINITE

MOVING

FORCES

The ordering of the options in the input below must be observed but the ordering of the options on the command line is unimportant, i.e.; the **ORTHO** card is first, the **FINITE** definition card below must precede the **MOVING** definition card, and the **FORCES** definition card should be last. The **ORTHO** option does not apply if the **MOVING** option is used. If an ID number is specified the additional option is available:

ID

If active, the ID card is the first card following the keyword.

Purpose: Define planar rigid walls with either finite or infinite size (**FINITE**). Orthotropic friction can be defined (**ORTHO**). Also, the plane can possess a mass and an initial velocity (**MOVING**); otherwise, the wall is assumed to be stationary. The **FORCES** option allows the specification of segments on the rigid walls on which the contact forces are computed. In order to achieve a more physical reaction related to the force versus time curve, the SOFT value on the **FORCES** card can be specified.

ID Card Define if and only if ID option is active.

Card 1	1	2	3	4	5	6	7	8
Variable	RWID							
Type	I							
Default	none							

VARIABLE	DESCRIPTION
RWID	Rigid wall ID. Up to 8 characters can be used.

- Cards 1 and 2 are required.
- Optional Cards A and B are required if ORTHO is specified.
- Optional Card C is required if FINITE is specified.
- Optional Card D is required if MOVING is specified.
- Optional Card E is required if FORCES is specified.

Required.

Card 1

1	2	3	4	5	6	7	8
Variable	NSID	NSIDEX	BOXID	OFFSET	BIRTH	DEATH	RWKSF	
Type	I	I	I	F	F	F	F	
Default	none	0	0	0.	0.	1.0E+20	1.0	

VARIABLE	DESCRIPTION
NSID	Nodal set ID containing slave nodes, see *SET_NODE_*OPTION*: EQ.0: all nodes are slave to rigid wall.
NSIDEX	Nodal set ID containing nodes that exempted as slave nodes, see *SET_NODE_*OPTION*.
BOXID	All nodes in box are included as slave nodes to rigid wall, see *DEFINE_ BOX. If options NSID or NSIDEX are active then only the subset of nodes activated by these options are checked to see if they are within the box.
OFFSET	All nodes within a normal offset distance, OFFSET, to the rigid wall are included as slave nodes for the rigid wall. If options NSID, NSIDEX, or BOXID are active then only the subset of nodes activated by these options are checked to see if they are within the offset distance. This option applies to the PLANAR wall only.
BIRTH	Birth time of rigid wall. The time values of the load curves that control the motion of the wall are offset by the birth time.
DEATH	Death time of rigid wall. At this time the wall is deleted from the calculation. If dynamic relaxation is active at the beginning of the calculation and if BIRTH=0.0, the death time is ignored during the dynamic relaxation.
RWKSF	Stiffness scaling factor. If RWKSF is also specified in *CONTROL_ CONTACT, the stiffness is scaled by the product of the two values.

Required.

Card 2

1	2	3	4	5	6	7	8
Variable	XT	YT	ZT	XH	YH	ZH	FRIC	WVEL
Type	F	F	F	F	F	F	F	F
Default	0.	0.	0.	0.	0.	0.	0.	0.

VARIABLE	DESCRIPTION
XT	x-coordinate of tail of any outward drawn normal vector, **n**, originating on wall (tail) and terminating in space (head), see Figure 28.3.
YT	y-coordinate of tail of normal vector **n**
ZT	z-coordinate of tail of normal vector **n**
XH	x-coordinate of head of normal vector **n**
YH	y-coordinate of head of normal vector **n**
ZH	z-coordinate of head of normal vector **n**
FRIC	Interface friction: EQ.0.0: frictionless sliding after contact, EQ.1.0: no sliding after contact, 0.<FRIC<1.: Coulomb friction coefficient. EQ.2.0: node is welded after contact with frictionless sliding. Welding occurs if and only if the normal value of the impact velocity exceeds the critical value specified by WVEL. EQ.3.0: node is welded after contact with no sliding. Welding occurs if and only if the normal value of the impact velocity exceeds the critical value specified by WVEL.
WVEL	Critical normal velocity at which nodes weld to wall (FRIC = 2 or 3).

Optional Cards A and B - Required if ORTHO is specified after the keyword.

See Figure 28.2 for the definition of orthotropic friction.

Optional Card A

	1	2	3	4	5	6	7	8
Variable	SFRICA	SFRICB	DFRICA	DFRICB	DECAYA	DECAYB		
Type	F	F	F	F	F	F		
Default	0.	0.	0	0	0.	0.		

Optional Card B

Variable	NODE1	NODE2	D1	D2	D3			
Type	I	I	F	F	F			
Default	0.	0.	0	0	0.			

VARIABLE	DESCRIPTION
SFRICA	Static friction coefficient in local a-direction, μ_{sa}, see Figure 28.2
SFRICB	Static friction coefficient in local b-direction, μ_{sb}
DFRICA	Dynamic friction coefficient in local a-direction, μ_{ka}
DFRICB	Dynamic friction coefficient in local b-direction, μ_{kb}
DECAYA	Decay constant in local a-direction, d_{va}
DECAYB	Decay constant in local b-direction, d_{vb}
NODE1	Node 1, alternative to definition with vector d below. See Figure 28.2. With the node definition the direction changes if the nodal pair rotates.
NODE2	Node 2
D1	d_1, x-component of vector, alternative to definition with nodes above. See Figure 28.2. This vector is fixed as a function of time.

VARIABLE	DESCRIPTION
D2	d_2, y-component of vector
D3	d_3, z-component of vector

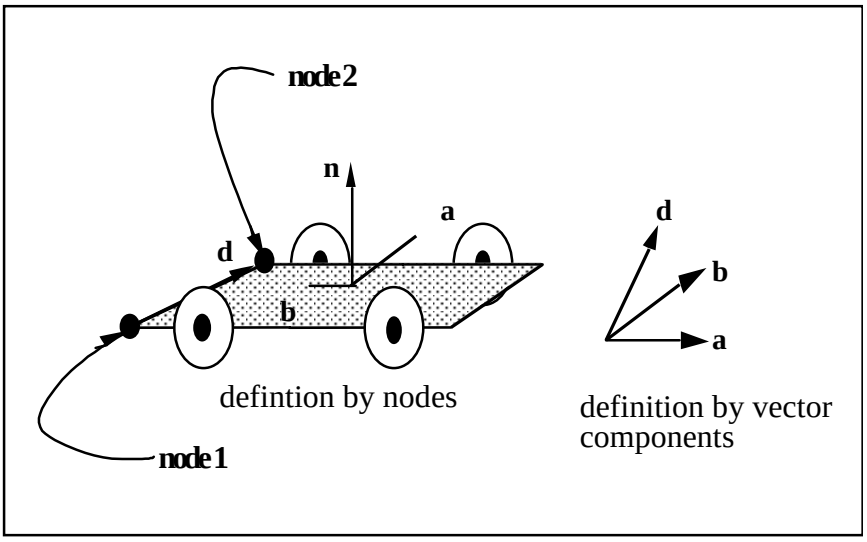


Figure 28.2. Definition of orthotropic friction vectors. The two methods of defining the vector, **d**, are shown. If vector **d** is defined by nodes 1 and 2, the local coordinate system may rotate with the body which contains the nodes; otherwise, **d** is fixed in space, thus on the rigid wall, and the local system is stationary.

Remarks:

1. The coefficients of friction are defined in terms of the static, dynamic and decay coefficients and the relative velocities in the local a and b directions as

$$\mu_a = \mu_{ka} + (\mu_{sa}\mu_{ka})e^{d_{va}V_{relative,a}}$$
$$\mu_b = \mu_{kb} + (\mu_{sb}\mu_{kb})e^{d_{vb}V_{relative,b}}$$

2. Orthotropic rigid walls can be used to model rolling objects on rigid walls where the frictional forces are substantially higher in a direction transverse to the rolling direction. To use this option define a vector **d** to determine the local frictional directions via:

$$\underset{\sim}{b} = \underset{\sim}{n} \times \underset{\sim}{d} \text{ and that } \underset{\sim}{a} = \underset{\sim}{b} \times \underset{\sim}{n}$$

where **n** is the normal vector to the rigid wall. If **d** is in the plane of the rigid wall, then **a** is identical to **d**.

Optional Card C - Required if FINITE is specified after the keyword.

See Figure 28.3. The **m** vector is computed as the vector cross product **m**=**n** X **l**. The origin, the tail of the normal vector, is taken as the corner point of the finite size plane.

Optional Card C	1	2	3	4	5	6	7	8
Variable	XHEV	YHEV	ZHEV	LENL	LENM			
Type	F	F	F	F	F			
Default	0.	0.	0.	infinity	infinity			

VARIABLE	DESCRIPTION
XHEV	x-coordinate of head of edge vector **l**, see Figure 28.3.
YHEV	y-coordinate of head of edge vector **l**
ZHEV	z-coordinate of head of edge vector **l**
LENL	Length of **l** edge
LENM	Length of **m** edge

Optional Card D - Required if MOVING is specified after keyword.

Note: The MOVING option is not compatible with the ORTHO option.

Optional Card D	1	2	3	4	5	6	7	8
Variable	MASS	V0						
Type	F	F						
Default	none	0.						

VARIABLE	DESCRIPTION
MASS	Total mass of stonewall
V0	Initial velocity of stonewall in direction of defining vector, **n**

Optional Card E - Required if FORCES is specified after the keyword.

This option allows the force distribution to be monitored on the plane. Also four points can be defined for visualization of the rigid wall. A shell or membrane element must be defined with these four points as the connectivity for viewing in LS-PREPOST.

Optional Card E	1	2	3	4	5	6	7	8
Variable	SOFT	SSID	N_1	N_2	N_3	N_4		
Type	I	I	I	I	I	I		
Default	0	0	0	0	0	0		
Remarks		1	2					

VARIABLE	DESCRIPTION
SOFT	Number of cycles to zero relative velocity to reduce force spike
SSID	Segment set identification number for defining areas for force output, see *SET_SEGMENT and remark 1 below.
N1-N4	Optional node for visualization

Remarks:

1. The segment set defines areas for computing resultant forces. These segments translate with the moving stonewall and allow the forced distribution to be determined. The resultant forces are written in file "RWFORC."

2. These four nodes are for visualizing the movement of the wall, i.e., they move with the wall. To view the wall in LS-PREPOST it is necessary to define a single shell element with these four nodes as its connectivity. The single element must be deformable (non rigid) or else the segment will be treated as a rigid body and the nodes will have their motion modified independently of the stonewall.

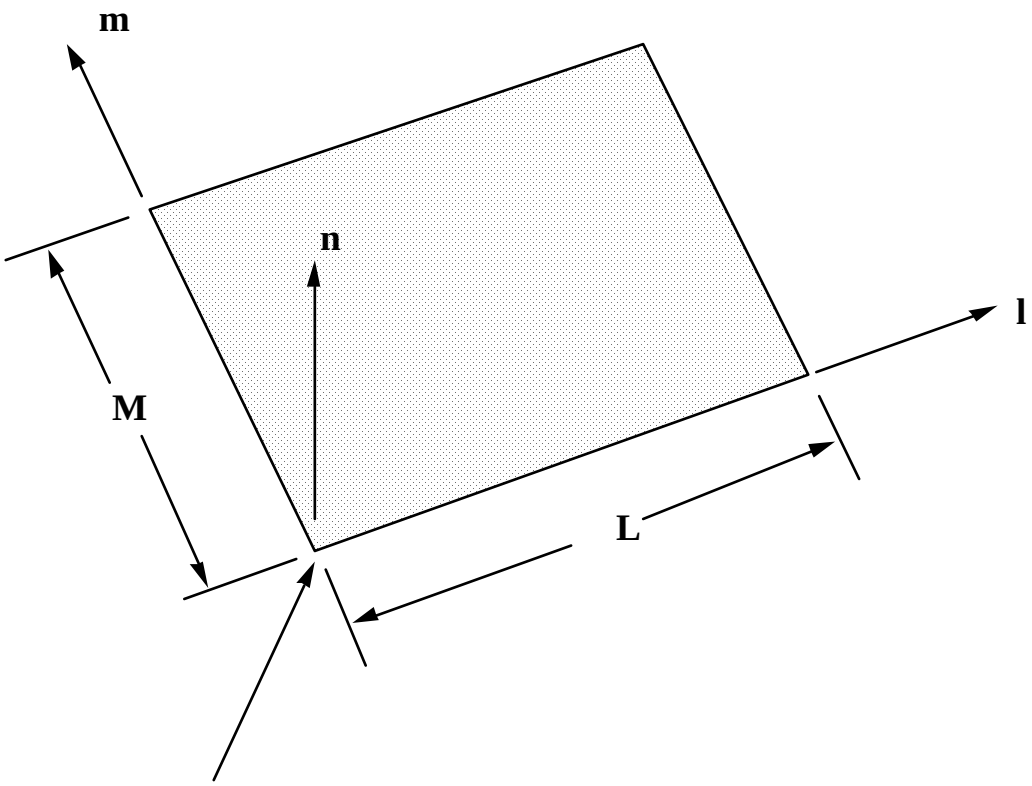


Figure 28.3. Vector **n** is normal to the stonewall. An optional vector **l** can be defined such that **m**=**n** ∞ **l**. The extent of the stonewall is limited by defining **L** (LENL) and **M** (LENM). A zero value for either of these lengths indicates that the stonewall is infinite in that direction.

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$  *RIGIDWALL_PLANAR_MOVING_FORCES
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Define a moving planar rigid wall:
$     - that is parallel to the y-z plane starting at x = 250 mm
$     - with an initial velocity of 8.94 mm/ms in the negative z-direction
$     - that has a mass of 800 kg
$     - which prevents all nodes in the model from penetrating the wall
$     - with a friction coefficient for nodes sliding along the wall of 0.1
$     - track the motion of the wall by creating a node (numbered 99999)
$         at the tail of the wall and assigning the node to move with the wall
$
*RIGIDWALL_PLANAR_MOVING_FORCES
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$     nsid    nsidex     boxid
         0         0         0
$
$       xt        yt        zt        xh        yh        zh      fric
     250.0       0.0       0.0       0.0       0.0       0.0       0.1
$
$  SW mass    SW vel
    800.00      8.94
$
$     soft      ssid     node1     node2     node3     node4
         0         0     99999
$
$
*NODE
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$    nid               x               y               z      tc      rc
   99999           250.0             0.0             0.0       0       0
$
$
*DATABASE_HISTORY_NODE
$   Define nodes that output into nodout
$      id1       id2       id3
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
     99999
$
*DATABASE_NODOUT
$       dt
       0.1
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

*SECTION

In this section, the element formulation, integration rule, nodal thicknesses, and cross sectional properties are defined. All section identifiers (SECID's) defined in this section must be unique, i.e., if a number is used as a section ID for a beam element then this number cannot be used again as a section ID for a solid element. The keyword cards in this section are defined in alphabetical order:

***SECTION_ALE2D**

***SECTION_BEAM**_{*OPTION*}

***SECTION_DISCRETE**

***SECTION_POINT_SOURCE**

***SECTION_POINT_SOURCE_MIXTURE**

***SECTION_SEATBELT**

***SECTION_SHELL**_{*OPTION*}

***SECTION_SOLID**_{*OPTION*}

***SECTION_SPH**_{*OPTION*}

***SECTION_TSHELL**

The location and order of these cards in the input file are arbitrary.

An additional option **_TITLE** may be appended to all the ***SECTION** keywords. If this option is used then an addition line is read for each section in 80a format which can be used to describe the section. At present LS-DYNA does make use of the title. Inclusion of titles gives greater clarity to input decks.

* SECTION_ALE2D

Purpose: This card defines the section properties for 2D ALE elements

Card 1 Format

	1	2	3	4	5	6	7	8
Variable	SECID	ALEFORM	AET	ELFORM				
Type	I	I	I	I				
Default	none	none	0	none				
Remarks								

VARIABLE	DESCRIPTION
SECID	Section ID. SECID is referenced on the *PART card and must be unique
ALEFORM	ALE formulation: EQ.6 : Single material Eulerian formulation EQ.11: Multi-Material ALE formulation
AET	Ambient Element Type EQ.4: Pressure inflow
ELFORM	Element formulation: EQ.13: Plane strain (x-y plane) EQ.14: Axisymmetric solid (y-axis of symmetry) – area weighted

Remarks:

***SECTION_BEAM**_*{OPTION}*

Available options include:

<BLANK>

AISC

such that the keyword cards appear:

***SECTION_BEAM**

***SECTION_BEAM_AISC**

Purpose: Define cross sectional properties for beam, truss, discrete beam, and cable elements.

The AISC option may be used to specify standard steel sections as specified by the American Institute of Steel Construction, and is described separately after *SECTION_BEAM

Card 1

1	2	3	4	5	6	7	8
Variable	SECID	ELFORM	SHRF	QR/IRID	CST	SCOOR	NSM	
Type	A8	I	F	F	F	F	F	
Default	none	1	1.0	2.0	0.0	0.0	0.0	

Define the appropriate card format depending on the value of ELFORM (1-9) above.

Card 2

1	2	3	4	5	6	7	8
Integrated beam type 1,11	TS1	TS2	TT1	TT2	NSLOC	NTLOC		
Resultant 2	A	ISS	ITT	J	SA	IST		
Truss 3	A	RAMPT	STRESS					
Resultant 2,3, and 12 *alternative*	STYPE	D1	D2	D3	D4	D5	D6	
Integrated beam type 4,5	TS1	TS2	TT1	TT2				

Card 2 (continued)	1	2	3	4	5	6	7	8
Discrete 6	VOL	INER	CID	CA	OFFSET	RRCON	SRCON	TRCON
Scalar 6	VOL	INER	CID	DOFN1	DOFN2			
2D shells 7,8	TS1	TS2	TT1	TT2				
Spot weld 9	TS1	TS2	TT1	TT2	PRINT			
Resultant 12 1st card	A	ISS	ITT	J	SA	IST		
Resultant 12 2nd card	YS	ZS	IYR	IZR	IRR	IW	IWR	
Type	A & F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
SECID	Section ID. SECID is referenced on the *PART card. A unique number or label not exceeding 8 characters must be specified.
ELFORM	Element formulation options: EQ.1: Hughes-Liu with cross section integration (default), EQ.2: Belytschko-Schwer resultant beam (resultant), EQ.3: truss (resultant), see remark 2. EQ.4: Belytschko-Schwer full cross-section integration, EQ.5: Belytschko-Schwer tubular beam with cross-section integration, EQ.6: discrete beam/cable, EQ.7: 2D plane strain shell element (xy plane), EQ.8: 2D axisymmetric volume weighted shell element (xy plane), EQ.9: spotweld beam, see *MAT_SPOTWELD. EQ.11: integrated warped beam EQ.12: resultant warped beam Note that the 2D and 3D element types must not be mixed, and different types of 2D elements must not be used together. For example, the plane strain element type must not be used with the axisymmetric element type. In 3D the different beam elements types, i.e., 1-6 and 9 can be freely mixed together.

VARIABLE	DESCRIPTION
SHRF	Shear factor. This factor is not needed for truss, resultant beam, discrete beam, and cable elements. The recommended value for rectangular sections is 5/6, the default is 1.0.
QR/IRID	Quadrature rule or rule number for user defined rule for integrated beams: EQ.1.0: one integration point, EQ.2.0: 2×2 Gauss quadrature (default beam), EQ.3.0: 3×3 Gauss quadrature, EQ.4.0: 3×3 Lobatto quadrature, EQ.5.0: 4×4 Gauss quadrature EQ.-n: where |n| is the number of the user defined rule. IRID integration rule n is defined using *INTEGRATION_BEAM card.
CST	Cross section type, not needed for truss, resultant beam, discrete beam, and cable elements: EQ.0.0: rectangular, EQ.1.0: tubular (circular only), EQ.2.0: arbitrary (user defined integration rule).
SCOOR	Location of triad for tracking the rotation of the discrete beam element, see the parameter CID below. The force and moment resultants in the output databases are referenced to this triad. The flags -3.0, -1.0, 0.0, 1.0, and 3.0 are inactive if the option to update the local system is active in the CID definition. EQ.-3.0: beam node 1, the angular velocity of node 1 rotates triad, EQ.-2.0: beam node 1, the angular velocity of node 1 rotates triad but the r-axis is adjusted to lie along the line between the two beam nodal points. This option is not recommended for zero length discrete beams., EQ.-1.0: beam node 1, the angular velocity of node 1 rotates triad, EQ. 0.0: centered between beam nodes 1 and 2, the average angular velocity of nodes 1 and 2 is used to rotate the triad, EQ.+1.0: beam node 2, the angular velocity of node 2 rotates triad. EQ.+2.0: beam node 2, the angular velocity of node 2 rotates triad. but the r-axis is adjusted to lie along the line between the two beam nodal points. This option is not recommended for zero length discrete beams. EQ.+3.0: beam node 2, the angular velocity of node 2 rotates triad. *If the magnitude of SC00R is less than or equal to unity then zero length discrete beams are assumed with infinitesimal separation between the nodes in the deformed state. For large separations or nonzero length beams set |SCOOR| to 2 or 3.*

VARIABLE	DESCRIPTION
NSM	Nonstructural mass per unit length. This option applies to beam types 1-5 and does not apply to discrete, 2D, and spotweld beams, respectively.
TS1	Beam thickness (CST=0.0, 2.0) or outer diameter (CST = 1.0) in s direction at node n_1. Note that the thickness defined on the *ELEMENT_ BEAM_THICKNESS card overrides the definition give here.
TS2	Beam thickness (CST=0.0, 2.0) or outer diameter (CST = 1.0) in s direction at node n_2. For truss elements only, it is the ramp up time for the stress initialization by dynamic relaxation.
TT1	Beam thickness (CST=0.0, 2.0) or inner diameter (CST = 1.0) in t direction at node n_1. For truss elements only, it is the stress for the initialization of the stress by dynamic relaxation.
TT2	Beam thickness (CST=0.0, 2.0) or inner diameter (CST = 1.0) in t direction at node n_2.
NSLOC	Location of reference surface normal to s axis for Hughes-Liu beam elements only. See Remark 5. EQ.1.0: side at s = 1.0, EQ.0.0: center, EQ.-1.0: side at s = -1.0.
NTLOC	Location of reference surface normal to t axis for Hughes-Liu beam elements only. See Remark 5. EQ.1.0: side at t =1.0, EQ.0.0: center, EQ.-1.0: side at t = -1.0.
A	Cross-sectional area. The definition on *ELEMENT_BEAM_ THICKNESS overrides the value defined here, see Figure 29.1.
ISS	I_{ss}, moment of inertia about local s-axis. The definition on *ELEMENT_BEAM_THICKNESS overrides the value defined here, see Figure 29.1.
ITT	I_{tt}, moment of inertia about local t-axis. The definition on *ELEMENT_BEAM_THICKNESS overrides the value defined here, see Figure 29.1.
J	J, torsional constant. The definition on *ELEMENT_BEAM_ THICKNESS overrides the value defined here, see Figure 29.1. If J is zero, then J is reset to the sum of ISS+ITT as an approximation for warped beam.

VARIABLE	DESCRIPTION
SA	Shear area. The definition on *ELEMENT_BEAM_THICKNESS overrides the value defined here, see Figure 29.1.
IST	I_{st}, product moment of inertia w.r.t. local s- and t-axis. This is only non-zero for unsymmetric cross sections and it can take positive and negative values, e.g. it is negative for SECTION_03.
YS	*s* coordinate of sheer center of cross-section. (The coordinate system is located at the centroid.)
ZS	*t* coordinate of sheer center of cross-section. (The coordinate system is located at the centroid.)
IYR	$\int_A s \cdot r^2 dA$, where $r^2 = s^2 + t^2$
IZR	$\int_A t \cdot r^2 dA$, where $r^2 = s^2 + t^2$
IRR	$\int_A r^4 dA$, where $r^2 = s^2 + t^2$
IW	Warping constant. $\int_A \omega^2 dA$, where ω is the sectorial area.
IWR	$\int_A \omega r^2 dA$
RAMPT	Optional ramp-up time for dynamic relaxation. At the end of the ramp-up time, a uniform stress, STRESS, will exist in the truss in the truss element. This option will not work for hyperelastic materials.
STRESS	Optional initial stress for dynamic relaxation. At the end of dynamic relaxation a uniform stress equal to this value should exist in the truss element.
STYPE	Section type (A format): EQ.SECTION_01: I-shape EQ.SECTION_02: Channel EQ.SECTION_03: L-shape EQ.SECTION_04: T-shape EQ.SECTION_05: Tubular box EQ.SECTION_06: Z-shape EQ.SECTION_07: Trapezoidal EQ.SECTION_08: Circular EQ.SECTION_09: Tubular EQ.SECTION_10: I-shape1 EQ.SECTION_11: Solid box EQ.SECTION_12: Cross EQ.SECTION_13: H-shape EQ.SECTION_14: T-shape1 EQ.SECTION_15: I-shape2 EQ.SECTION_16: Channel1 EQ.SECTION_17: Channel2 EQ.SECTION_18: T-shape2 EQ.SECTION_19: Box-shape1 EQ.SECTION_20: Hexagon EQ.SECTION_21: Hat-shape EQ.SECTION_22: Hat-shape1

VARIABLE	DESCRIPTION
D1-D6	Input parameters for section option using STYPE above.
VOL	Volume of discrete beam. If the mass density of the material model for the discrete beam is set to unity, the magnitude of the lumped mass can be defined here instead. This lumped mass is partitioned to the two nodes of the beam element. The translational time step size for the type 6 beam is dependent on the volume, mass density, and the translational stiffness values, so it is important to define this parameter. Defining the volume is also essential for mass scaling if the type 6 beam controls the time step size.
INER	Mass moment of inertia for the six degree of freedom discrete beam. This lumped inertia is partitioned to the two nodes of the beam element. The rotational time step size for the type 6 beam is dependent on the lumped inertia and the rotational stiffness values, so it is important to define this parameter if the rotational springs are active. Defining the rotational inertia is also essential for mass scaling if the type 6 beam rotational stiffness controls the time step size.
CID	Coordinate system ID for orientation (material types 66-69, 93, 95, 97), see *DEFINE_COORDINATE_option. If CID=0, a default coordinate system is defined in the global system or on the third node of the beam, which is used for orientation. This option is not defined for material types than act between two nodal points, such as cable elements. The coordinate system rotates with the discrete beam, see SCOOR above.
CA	Cable area, materials type ID 71, *MAT_CABLE.
OFFSET	Offset for cable. For a definition see materials type ID 71, *MAT_ CABLE.
RRCON	r-rotational constraint for local coordinate system EQ.0.0: Coordinate ID rotates about r axis with nodes. EQ.1.0: Rotation is constrained about the r-axis
SRCON	s-rotational constraint for local coordinate system EQ.0.0: Coordinate ID rotates about s axis with nodes. EQ.1.0: Rotation is constrained about the s-axis
TRCON	t-rotational constraint for local coordinate system EQ.0.0: Coordinate ID rotates about t axis with nodes. EQ.1.0: Rotation is constrained about the t-axis
DOFN1	Active degree-of-freedom at node 1, a number between 1 and 6 where 1 in x-translation and 4 is x-rotation.
DOFN2	Active degree-of-freedom at node 2, a number between 1 and 6.

VARIABLE	DESCRIPTION
PRINT	Output spot force resultant from spotwelds. EQ.0.0: Data is output to SWFORC file. EQ.1.0: Output is suppressed.

Remarks:

1. For implicit calculations all of the beam element choices are implemented:

2. For the truss element, define the cross-sectional area, A, only.

3. The local coordinate system rotates as the nodal points that define the beam rotate. In some cases this may lead to unexpected results if the nodes undergo significant rotational motions. In the definition of the local coordinate system using *DEFINE_COORDINATE_ NODES, if the option to update the system each cycle is active then this updated system is used. This latter technique seems to be more stable in some applications.

4. The integrated warped beam (type 11) is a 7 degree of freedom beam that must be used with an integration rule of the open standard cross sections, see *INTEGRATION_BEAM. To incorporate the additional degrees of freedom corresponding to the twist rates, the user should declare one scalar node (*NODE_SCALAR) for each node attached to a warped beam. This degree of freedom is associated to the beam element using the warpage option on the *ELEMENT_BEAM card.

5. Beam offsets are sometimes necessary for correctly modeling beams that act compositely with other elements such as shells or other beams. A beam offset extends from the beam's N1-to-N2 axis to the reference axis of the beam. The beam reference axis lies at the origin of the local s and t axes, i.e., halfway between the outermost surfaces of the beam cross-section. Note that for cross-sections that are not doubly symmetric, e.g, a T-section, the reference axis does not pass through the centroid of the cross-section. The offset in the positive s-direction is s-offset = -0.5 * NSLOC * (beam cross-section dimension in s-direction). Similarly, the offset in the positive t-direction is t-offset = -0.5 * NTLOC * (beam cross-section dimension in t-direction). See also *ELEMENT_BEAM_OFFSET for an alternate approach to defining beam offsets.

```
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$   *SECTION_BEAM
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Define a Belytschko-Schwer resultant beam (elform = 2) with the following
$  properties.  This beam models the connection/stiffening beams of a medium
$  size roadside sign.
$
$    cross sectional area:                          a =     515.6 mm2
$    2nd moment of area about s-axis:             iss =  99,660.0 mm4
$    2nd moment of area about t-axis:             iss =  70,500.0 mm4
$    2nd polar moment of area about beam axis:      j = 170,000.0 mm4
$
*SECTION_BEAM
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$      sid    elform      shrf   qr/irid       cst
       111         2
$
$        a       iss       itt         j        sa
     515.6   99660.0   70500.0  170000.0
$
*SECTION_BEAM_TITLE
    Main beam member
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$      sid    elform      shrf   qr/irid       cst
       111         2
$
$        a       iss       itt         j        sa
     515.6   99660.0   70500.0  170000.0
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

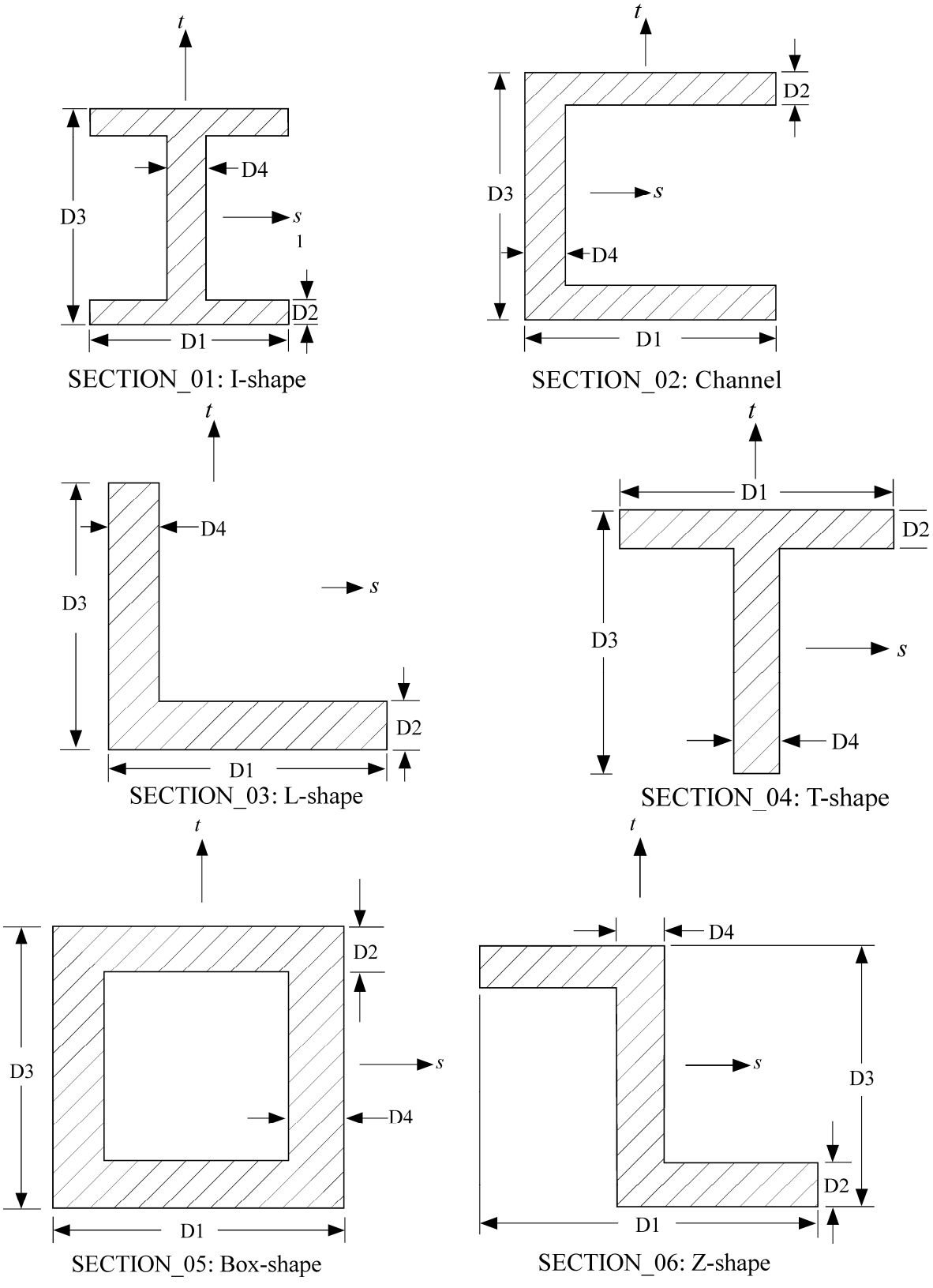
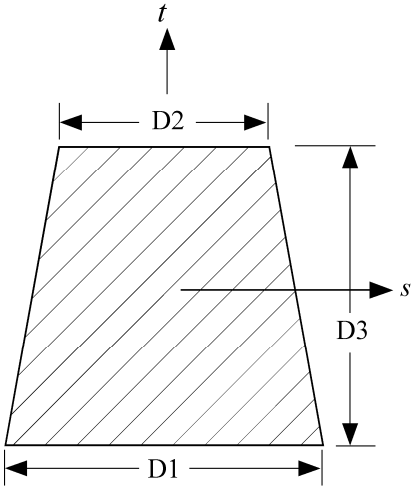
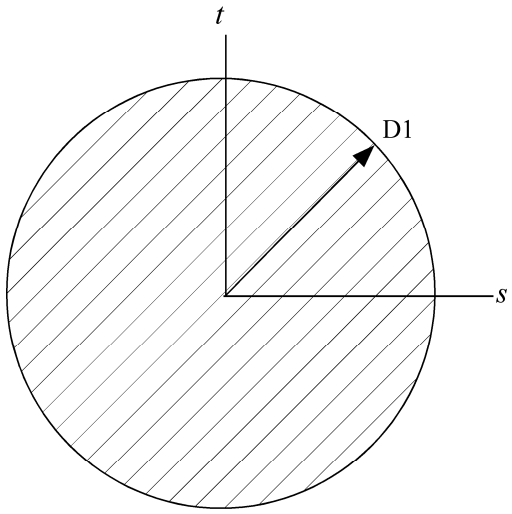


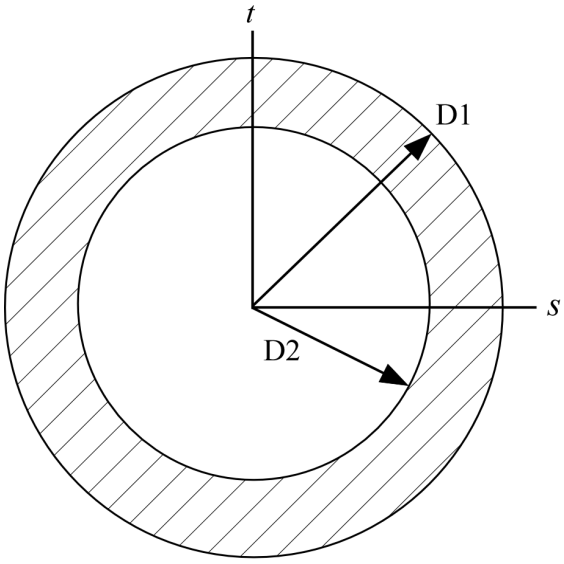
Figure 29.1. Properties of beam cross section for several common cross sections.



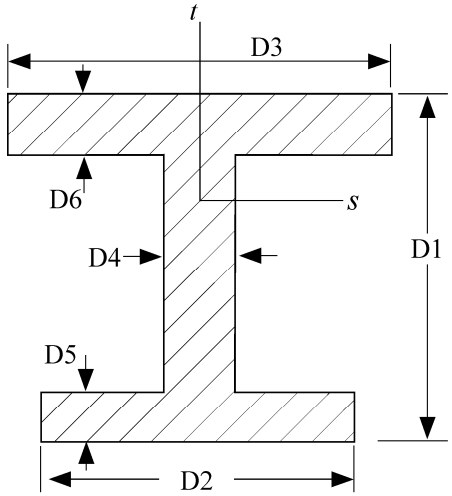
SECTION_07: Trapezoidal



SECTION_08: Circular

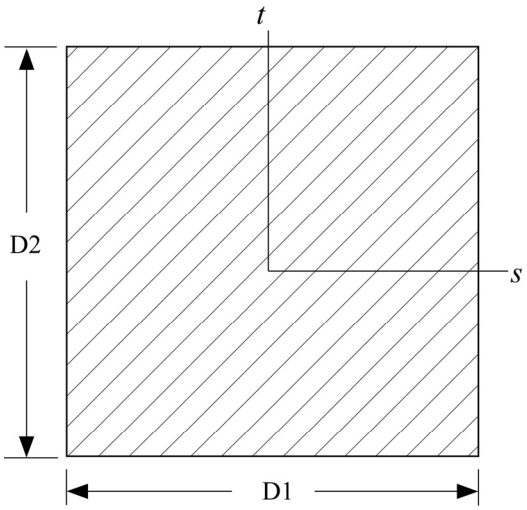


SECTION_09: Tubular

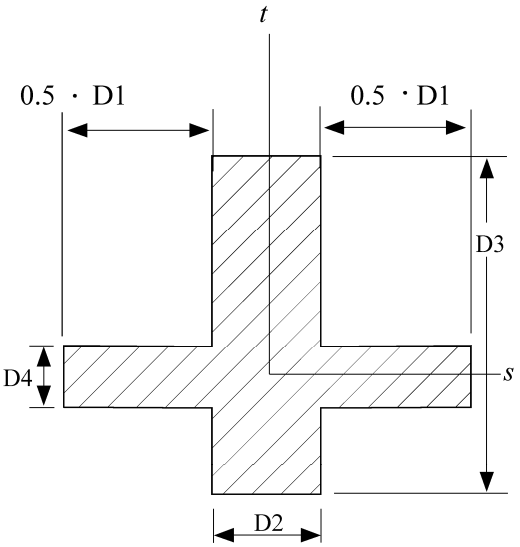


SECTION_10: I-shape1

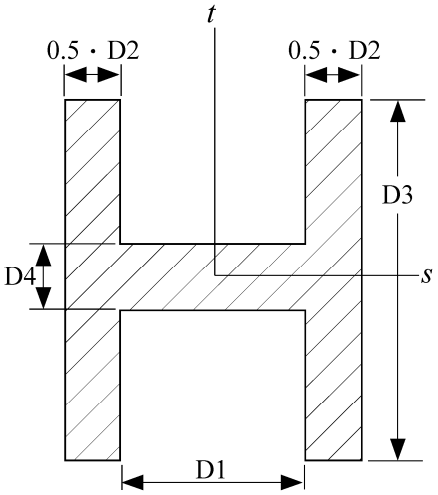
Figure 29.1. Properties of beam cross section for several common cross sections (cont'd).



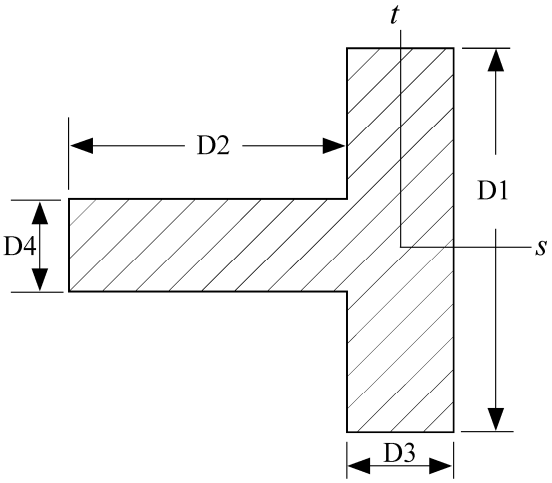
SECTION_11: Solid box



SECTION_12: Cross

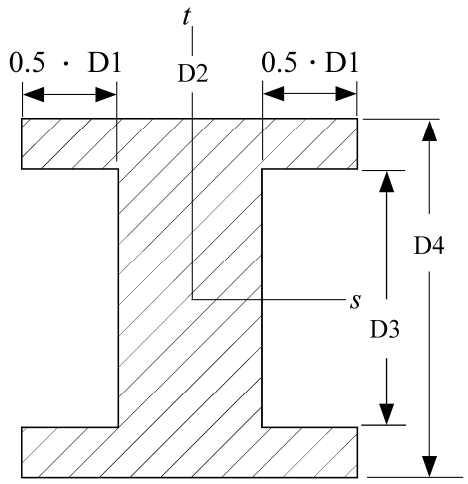


SECTION_13: H-shape

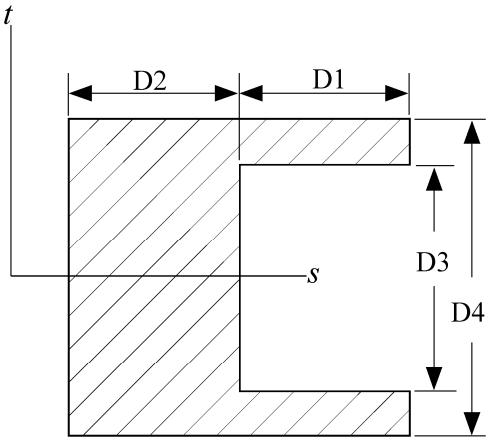


SECTION_14: T-shape1

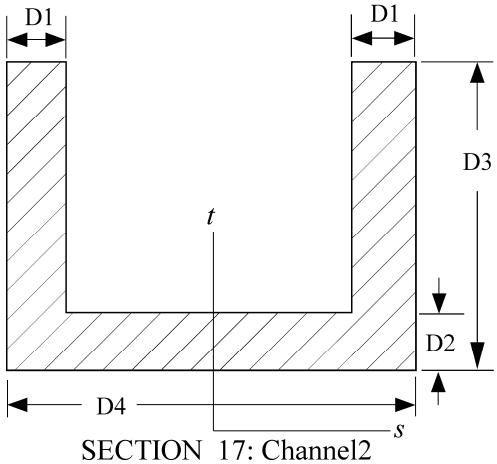
Figure 29.1. Properties of beam cross section for several common cross sections (cont'd).



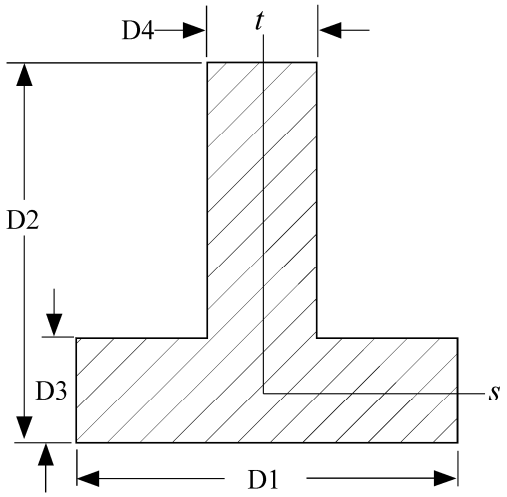
SECTION_15: I-shape2



SECTION_16: Channel1

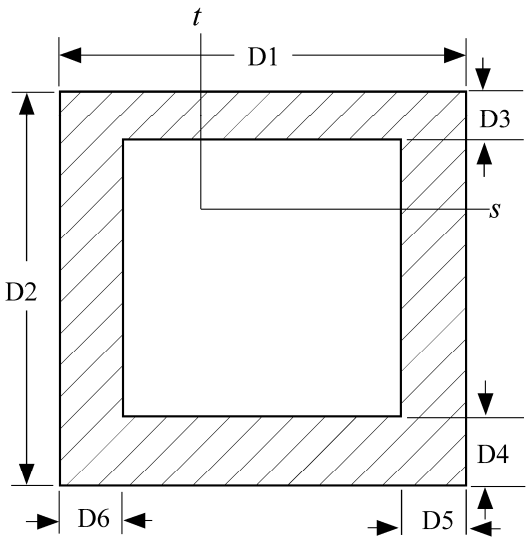


SECTION_17: Channel2

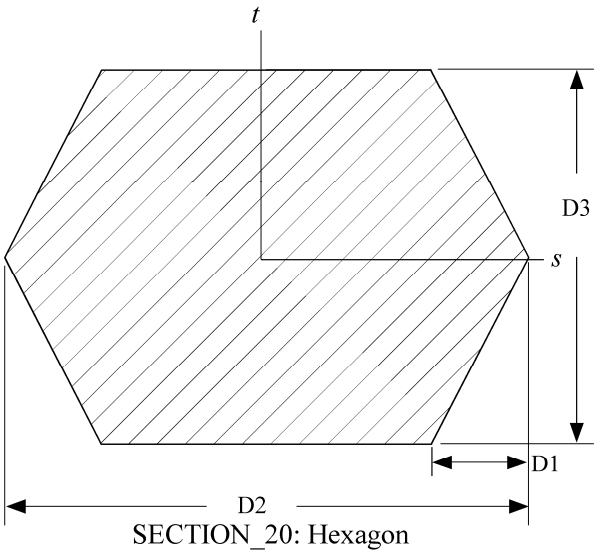


SECTION_18: T-shape

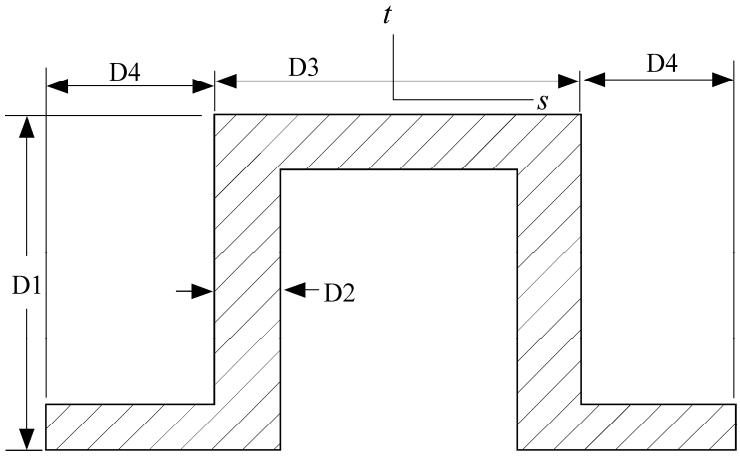
Figure 29.1. Properties of beam cross section for several common cross sections (cont'd).



SECTION_19: Box-shape1



SECTION_20: Hexagon



SECTION_21: Hat-shape

Figure 29.1. Properties of beam cross section for several common cross sections (cont'd).

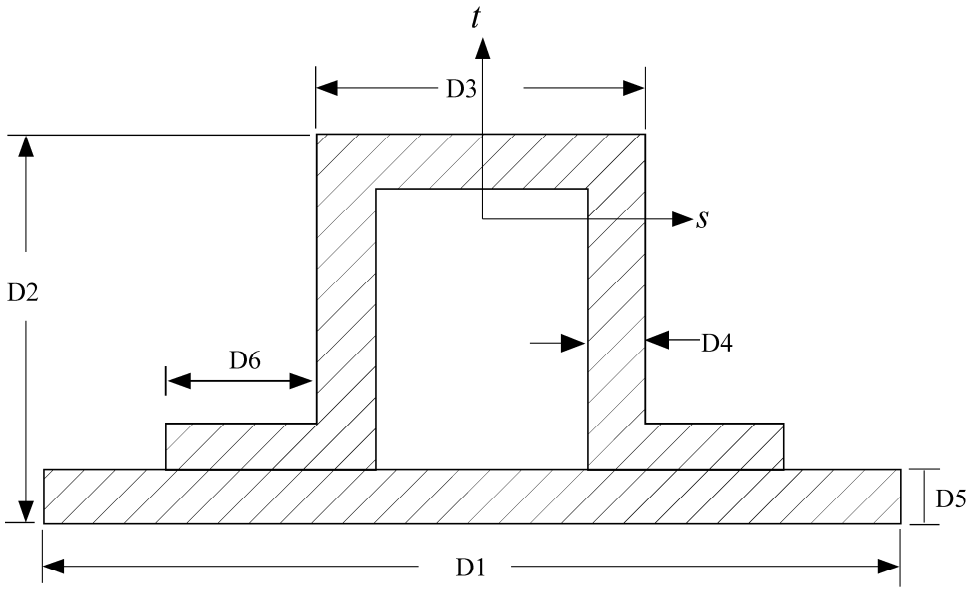


Figure 29.1. Properties of beam cross section for several common cross sections (cont'd).

***SECTION_BEAM_AISC**

Purpose: Define cross-sectional properties for beams and trusses using section labels from the AISC Steel Construction Manual, 2005, 13th Edition, as published in the AISC Shapes Database V13.1.1

Card 1 1 2-8

Variable	SECID	LABEL
Type	I	A70

Define the appropriate card format according to the type of beam formulation

Card 2 1 2 3 4 5 6 7 8

Integrated beam type 1,11	ELFORM	SHRF	NSM	LFAC	NSLOC	NTLOC	K	
Resultant 2, 12	ELFORM	SHRF	NSM	LFAC				
Truss 3	ELFORM	LFAC	RAMPT	STRESS				
Integrated beam type 4,5	ELFORM	SHRF	NSM	LFAC	K			
Type	I	F	F	F	F & I	F	I	

VARIABLE	DESCRIPTION
SECID	Section ID. SECID is referenced on the *PART card. A unique number or label not exceeding 8 characters must be specified.
LABEL	AISC section label
ELFORM	Element formulation (see *SECTION_BEAM). Only types 1–5,11,12 are allowed
SHRF	Shear factor (see *SECTION_BEAM)

VARIABLE	DESCRIPTION
NSM	Non-structural mass per unit length
LFAC	Length scale factor to convert dimensions from standard units
NSLOC	Location of reference surface (see *SECTION_BEAM)
NTLOC	Location of reference surface (see *SECTION_BEAM)
K	Integration refinement parameter (see *INTEGRATION_BEAM)
RAMPT	Optional ramp-up time (see *SECTION_BEAM)
STRESS	Optional initial stress (see *SECTION_BEAM)

Remarks:

This keyword uses the dimensions of the standard AISC beams sections — as defined by the section label — to define *SECTION_BEAM and *INTEGRATION_BEAM cards with the appropriate parameters.

The AISC section label may be specified either as the shape designation as seen in the AISC Steel Construction Manual, 2005, or the designation according to the AISC Naming Convention for Structural Steel Products for Use in Electronic Data Interchange (EDI), 2001. As per the EDI convention, the section labels are to be case-sensitive and space sensitive, i.e. “W36X150” is acceptable but “W36 x 150” is not. Labels can be specified in terms of either the U.S. Customary units (in) or metric units (mm), which will determine the length units for the section dimensions. The parameter LFAC may be used as a multiplier to convert the dimensions to other lengths units.

***SECTION_DISCRETE**

Purpose: Defined spring and damper elements for translation and rotation. These definitions must correspond with the material type selection for the elements, i.e., *MAT_SPRING_... and *MAT_DAMPER_...

Card 1	1	2	3	4	5	6	7	8
Variable	SECID	DRO	KD	V0	CL	FD		
Type	A8	I	F	F	F	F		

Card 2

Variable	CDL	TDL						
Type	F	F						

VARIABLE	DESCRIPTION
SECID	Section ID. SECID is referenced on the *PART card. A unique number or label not exceeding 8 characters must be specified.
DRO	Displacement/Rotation Option: EQ.0: the material describes a translational spring/damper, EQ.1: the material describes a torsional spring/damper.
KD	Dynamic magnification factor. See Remarks 1 and 2 below.
V0	Test velocity
CL	Clearance. See Remark 3 below.
FD	Failure deflection (twist for DRO=1). Negative for compression, positive for tension.
CDL	Deflection (twist for DRO=1) limit in compression. See Remark 4 below.
TDL	Deflection (twist for DRO=1) limit in tension. See Remark 4 below.

Remarks:

1. The constants from KD to TDL are optional and do not need to be defined.

2. If k_d is nonzero, the forces computed from the spring elements are assumed to be the static values and are scaled by an amplification factor to obtain the dynamic value:

$$F_{dynamic} = \left(1. + k_d \frac{V}{V_0} \right) F_{static}$$

where

V = absolute value of the relative velocity between the nodes.

V_0 = dynamic test velocity.

For example, if it is known that a component shows a dynamic crush force at 15m/s equal to 2.5 times the static crush force, use k_d =1.5 and V_0=15.

3. Here, "clearance" defines a compressive displacement which the spring sustains before beginning the force-displacement relation given by the load curve defined in the material selection. If a non-zero clearance is defined, the spring is compressive only.

4. The deflection limit in compression and tension is restricted in its application to no more than one spring per node subject to this limit, and to deformable bodies only. For example in the former case, if three springs are in series, either the center spring or the two end springs may be subject to a limit, but not all three. When the limiting deflection is reached, momentum conservation calculations are performed and a common acceleration is computed in the appropriate direction. An error termination will occur if a rigid body node is used in a spring definition where deflection is limited.

   Constrained boundary conditions on the *NODE cards and the BOUNDARY_SPC cards must not be used for nodes of springs with deflection limits.

5. Discrete elements can be included in implicit applications.

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$  *SECTION_DISCRETE
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Note: These examples are in kg, mm, ms, kN units.
$
$  A translational spring (dro = 0) is defined to have a failure deflection
$  of 25.4 mm (fd = 25.4).  The spring has no dynamic effects or
$  deflection limits, thus, those parameters are not set.
$
*SECTION_DISCRETE
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$      sid       dro        kd        v0        cl        fd
       104         0                                    25.4
$
$      cdl       tdl

$
$
$  Define a translational spring that is known to have a dynamic crush force
$  equal to 2.5 times the static force at a 15 mm/ms deflection rate.
$  Additionally, the spring is known to be physically constrained to deflect
$  a maximum of 12.5 mm in both tension and compression.
$
*SECTION_DISCRETE
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$      sid       dro        kd        v0        cl        fd
       107         0       1.5      15.0
$
$      cdl       tdl
      12.5      12.5
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

*SECTION_POINT_SOURCE

Purpose: This command provides the inlet boundary condition for single gas in flow (inflation potential) via a set of point source(s). It also provides the inflator orifice geometry information. It requires 3 curves defining the inlet condition for the inflator gas coming into the tank or an airbag as input ($\overline{T}_{gas\_corrected}(t)$, $v_r(t)$, and $vel(t)$). Please see also the *ALE_TANK_TEST card for additional information.

Card 1

1	2	3	4	5	6	7	8
SECID	LCIDT	LCIDVOLR	LCIDVEL	NIDLC001	NIDLC002	NIDLC003	
A8	I	I	I	I	I	I	
0	0	0	0	0	0	0	

Card 2

1	2	3	4	5	6	7	8
NODEID	VECID	ORIFAREA					
I	I	F					
0	0	0.0					

VARIABLE	DESCRIPTION
SECID	Section ID. A unique number or label not exceeding 8 characters must be specified.
LCIDT	Temperature load curve ID
LCIDVOLR	Relative volume load curve ID
LCIDVEL	Inlet flow velocity load curve ID
NIDLC001	The 1st node ID defining a local coordinate (See Remark 2).
NIDLCOO2	The 2nd node ID defining a local coordinate (See Remark 2).
NIDLCOO3	The 3rd node ID defining a local coordinate (See Remark 2).

VARIABLE	DESCRIPTION
NODEID	The node ID(s) defining the point source(s).
VECID	The vector ID defining the direction of flow at each point source.
ORIFAREA	The orifice area at each point source.

Remarks:

1. In an airbag inflator tank test, the tank pressure data is measured. This pressure is used to derive $\dot{m}(t)$ and the estimated $\bar{T}_{gas}(t)$, usually via a lumped-parameter method, a system of conservation equations and EOS. Subsequently $\dot{m}(t)$ and $\bar{T}_{gas}(t)$ (stagnation temperature) are used as input to obtain $\bar{T}_{gas\_corrected}(t)$ (static temperature), $v_r(t)$, and $vel(t)$. These 3 curves are then used to describe inflator gas inlet condition (see *ALE_TANK_TEST for more information).

2. In a car crash model, the inflator housing may get displaced during the impact. The 3 node IDs defines the local reference coordinate system to which the point sources are attached. These 3 reference nodes may be located on a rigid body which can translate and rotate as the inflator moves during the impact. This allows for the point sources to move in time. These reference nodes may be used as the point sources themselves.

3. If the *ALE_TANK_TEST card is present, please see the Remarks under that card.

Example:

Consider a tank test model which consists of the inflator gas (PID 1) and the air inside the tank (PID 2). The 3 load curves define the thermodynamic and kinetic condition of the incoming gas. The nodes define the center of the orifice, and the vector the direction of flow at each orifice.

```
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|...8
*PART
inflator gas
$      PID     SECID       MID     EOSID      HGID      GRAV    ADPOPT      TMID
         1         1         1         0         0         0         0         0
*SECTION_POINT_SOURCE
$    SECID     LCIDT  LCIDVOLR   LCIDVEL NIDLCOOR1 NIDLCOOR2 NIDLCOOR3
         1         3         4         5         0         0         0
$   NODEID    VECTID      AREA
     24485         3    15.066
       ...
     24557         3    15.066
*PART
air inside the tank
$      PID     SECID       MID     EOSID      HGID      GRAV    ADPOPT      TMID
         2         2         2         0         0         0         0         0
*SECTION_SOLID
$    SECID    ELFORM       AET
         2        11         0
*ALE_MULTI-MATERIAL_GROUP
$      SID   SIDTYPE
         1         1
         2         1
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|...8
```

*SECTION_POINT_SOURCE_MIXTURE

Purpose: This command provides (a) an element formulation for a solid ALE part of the type similar to ELFORM=11 of *SECTION_SOLID, and (b) the inlet gas injection boundary condition for multiple-gas mixture in-flow via a set of point source(s). It also provides the inflator orifice geometry information. This must be used in combination with the *MAT_GAS_MIXTURE and/or *INITIAL_GAS_MIXTURE card (see Remark 1).

Card 1

Variable	SECID	LCIDT	Not Used	LCIDVEL	NIDLC001	NIDLC002	NIDLC003	IDIR
Type	A8	I		I	I	I	I	I
Default	none	none		none	none	none	none	0

Card 2

Variable	LCMDOT1	LCMDOT2	LCMDOT3	LCMDOT4	LCMDOT5	LCMDOT6	LCMDOT7	LCMDOT8
Type	I	I	I	I	I	I	I	I
Default	none	none	none	none	none	none	none	none

Card 3

Variable	NODEID	VECID	ORIFAREA					
Type	I	I	F					
Default	none	none	0.0					

VARIABLE	DESCRIPTION
SECID	Section ID. A unique number or label not exceeding 8 characters must be specified.

VARIABLE	DESCRIPTION
LCIDT	Inflator gas mixture average stagnation temperature load curve ID (all gases of the mixture are assumed to have the same average temperature).
LCIDVEL	User-defined inflator gas mixture average velocity load curve ID. If LCIDVEL=0 or blank, LSDYNA will estimate the inlet gas velocity.
NIDLC001	The 1st node ID defining a local coordinate (see Remark 2).
NIDLC002	The 2nd node ID defining a local coordinate (see Remark 2).
NIDLC003	The 3rd node ID defining a local coordinate (see Remark 2).
IDIR	A flag for constraining the nodal velocity of the nodes of the ALE element containing a point source. If IDIR=0 (default), then the ALE nodes behind the point source (relative position of nodes based on the vector direction of flow of point source) will have zero velocity. If IDIR=1, then all ALE nodes will have velocity distributed based on energy conservation. The latter option seems to be more robust in airbag modeling (see Remark 6).
LCMDOT1	The mass flow rate load curve ID of the1st gas in the mixture.
LCMDOTn	The mass flow rate load curve ID of the nth gas in the mixture.
LCMDOT8	The mass flow rate load curve ID of the 8th gas in the mixture.
NODEID	The node ID(s) defining the point sources (see Remark 6).
VECID	The vector ID defining the direction of flow at each point source.
ORIFAREA	The orifice area at each point source.

Remarks:

1. This command is used to define a part that acts as the ideal gas mixture injection source. The associated ALE material (gas mixture) may not be present at time zero, but can be introduced (injected) into an existing ALE domain. For airbag application, the input from control volume analysis, inlet mass flow rate, $\dot{m}(t)$, and, inlet stagnation gas temperature, $\bar{T}_{gas}(t)$ may be used as direct input for ALE analysis. If available, the user may input a load curve for the gas mixture average inlet velocity. If not, LS-DYNA will estimate the inlet gas velocity.

2. The gas mixture is assumed to have a uniform temperature ($\bar{T} \approx T_i$) and inlet velocity. However, the species in the mixture may each have a different inlet mass flow rate.

3. A brief review of the concept used is presented. The total energy (e_T) is the sum of internal (e_i) and kinetic $\left(\frac{V^2}{2}\right)$ energies, (per unit mass).

$$e_T = e_i + \frac{V^2}{2}$$

$$C_V T_{stag} = C_V T + \frac{V^2}{2}$$

$$T_{stag} = T + \frac{V^2}{2C_V}$$

The distinction between stagnation and static temperatures is shown above. C_V is the constant-volume heat capacity. The gas mixture average internal energy per unit mass in terms of mixture species contribution is

$$e_i = \bar{C}_V \bar{T} = \sum_i \left(\frac{\rho_i}{\rho_{mixture}}\right) C_{V_i} T_i = \left[\sum_i \left(\frac{\rho_i}{\rho_{mixture}}\right) C_{V_i}\right] \bar{T}$$

$$\bar{C}_V = \left[\sum_i \left(\frac{\rho_i}{\rho_{mixture}}\right) C_{V_i}\right]$$

Since we approximate $\bar{T} \approx T_i$, then gas mixture average static temperature is related to the mixture average internal energy per unit mass as following

$$\bar{T} = \frac{e_i}{\left[\sum_i \left(\frac{\rho_i}{\rho_{mixture}}\right) C_{V_i}\right]}$$

Note that the "i" subscript under "e" denotes "internal" energy, while the other "i" subscripts denote the "ith" species in the gas mixture. The total mixture pressure is the sum of the partial pressures of the individual species.

$$\bar{p} = \sum_i p_i$$

The ideal gas EOS applies to each individual species (by default)

$$P_i = \rho_i \left(C_{P_i} - C_{V_i}\right) T_i$$

4. Generally, it is not possible to conserve both momentum and kinetic (KE) at the same time. Typically, internal energy (IE) is conserved and KE may not be. This may result in some KE loss (hence, total energy loss). For many analyses this is tolerable, but for

airbag application, this may lead to the reduction of the inflating potential of the inflator gas.

In *MAT_GAS_MIXTURE computation, any kinetic energy not accounted-for during advection is stored in the internal energy. Therefore, there is no kinetic energy loss, and the total energy of the element is conserved over the advection step. This is a simple, ad hoc approach that is not rigorously derived for the whole system based on first principles. Therefore it is not guaranteed to apply universally to all scenarios. It is the user's responsibility to validate the model with data.

5. Since ideal gas is assumed, there is no need to define the EOS for the gases in the mixture.

6. In general, it is best to locate a point source near the center of an ALE element. Associated with each point source is an area and a vector indicating flow direction. Each point source should occupy 1 ALE element by itself, and there should be at least 2 empty ALE elements between any 2 point sources. A point source should be located at least 3 elements away from the free surface of an ALE mesh for stability.

Example 1:

Consider a tank test model without coupling which consists of:

- a background mesh with air (PID 1 = gas 1) initially inside that mesh (tank space), and
- the inflator gas mixture (PID 2 consisting of inflator gases 2, 3, and 4).

The mixture is represented by one AMMGID and the air by another AMMGID.

The tank internal space is simply modeled with an Eulerian mesh of the same volume. The Tank itself is not modeled thus no coupling is required. The inflator gases fill up this space mixing with the air initially inside the tank.

The background air (gas 1) is included in the gas mixture definition in this case because that air will participate in the mixing process. Only include in the mixture those gases that actually undergo mixing (gases 1, 2, 3 and 4). Note that for an airbag model, the "outside" air should not be included in the mixture (it should be defined independently) since it does not participate in the mixing inside the airbag. This is shown in the next example.

The nodes define the center of the orifices, and the vectors define the directions of flow at these orifices.

```
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|....8
*PART
Tank background mesh, initially filled with air, allows gas mixture to flow in.
$      PID     SECID       MID     EOSID      HGID      GRAV    ADPOPT      TMID
         1         1         1         0         0         0         0         0
*SECTION_SOLID
$    SECID    ELFORM       AET
         1        11         0
$ The next card defines the properties of the gas species in the mixture.
*MAT_GAS_MIXTURE
$      MID
         1
$      Cv1       Cv2       Cv3       Cv4       Cv5       Cv6       Cv7       Cv8
    654.47    482.00   2038.30    774.64       0.0       0.0       0.0       0.0
$      Cp1       Cp2       Cp3       Cp4       Cp5       Cp6       Cp7       Cp8
    941.32    666.67   2500.00   1071.40       0.0       0.0       0.0       0.0

$ The next card specifies that gas 1 (background air) occupies PID 1 at time 0.
*INTIAL_GAS_MIXTURE
$      SID     STYPE    AMMGID     TEMP0
         1         1         1    293.00
$     RHO1      RHO2      RHO3      RHO4      RHO5      RHO6      RHO7      RHO8
   1.20E-9       0.0       0.0       0.0       0.0       0.0       0.0       0.0
*PART
The gas mixture (inlet) definition (no initial mesh required for this PID)
$      PID     SECID       MID     EOSID      HGID      GRAV    ADPOPT      TMID
         2         2         1         0         0         0         0         0
*SECTION_POINT_SOURCE_MIXTURE
$    SECID     LCIDT   NOTUSED   LCIDVEL NIDLCOOR1 NIDLCOOR2 NIDLCOOR3      IDIR
         2         1         0         5         0         0         0         0
$  LCMDOT1   LCMDOT2   LCMDOT3   LCMDOT4   LCMDOT5   LCMDOT6   LCMDOT7   LCMDOT8
         0         2         3         4         0         0         0         0
$   NODEID    VECTID      AREA
     24485         1      25.0
       ...
     24557         1      25.0
*ALE_MULTI-MATERIAL_GROUP
$      SID   SIDTYPE
         1         1
         2         1
*DEFINE_VECTOR
$   VECTID     XTAIL     YTAIL     ZTAIL     XHEAD     YHEAD     ZHEAD
         1       0.0       0.0       0.0       0.0       1.0       0.0
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|....8
```

Example 2:

Consider an airbag inflation model which consists of:

- a background Eulerian mesh for air initially outside the airbag (PID 1)
- the inflator gas mixture (PID 2 consisting of inflator gases 1, 2, and 3).

The mixture is represented by one AMMGID and the air by another AMMGID.

The background air (PID 1) is NOT included in the gas mixture definition in this case because that air will NOT participate in the mixing process. Only include in the mixture those gases that actually undergo mixing (gases 1, 2, and 3). Gases 1, 2, and 3 in this example correspond to gases 2, 3, and 4 in example 1. Compare the air properties in PID 1 here to that of example 1. Note that the *INITIAL_GAS_MIXTURE card is not required to initialize the background mesh in this case.

```
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|....8
*PART
Tank background mesh, initially filled with air, allows gas mixture to flow in.
$       PID     SECID       MID     EOSID      HGID      GRAV    ADPOPT      TMID
         1         1         1         0         0         0         0         0
*SECTION_SOLID
$    SECID    ELFORM       AET
         1        11         0
*MAT_NULL
$       MID       RHO      PCUT        MU     TEROD     CEROD        YM        PR
         1   1.20E-9   -1.0E-6       0.0       0.0       0.0       0.0       0.0
*EOS_IDEAL_GAS
$    EOSID        CV0       CP0     COEF1     COEF2        T0    RELVOL0
         1    654.47    941.32       0.0       0.0    293.00       1.0
$ The next card defines the properties of the gas species in the mixture.
*PART
The gas mixture (inlet) definition (no initial mesh required for this PID)
$       PID     SECID       MID     EOSID      HGID      GRAV    ADPOPT      TMID
         2         2         2         0         0         0         0         0
*SECTION_POINT_SOURCE_MIXTURE
$    SECID     LCIDT   NOTUSED   LCIDVEL NIDLCOOR1 NIDLCOOR2 NIDLCOOR3      IDIR
         2         1         0         5         0         0         0         0
$  LCMDOT1   LCMDOT2   LCMDOT3   LCMDOT4   LCMDOT5   LCMDOT6   LCMDOT7   LCMDOT8
         2         3         4         0         0         0         0         0
$   NODEID    VECTID      AREA
     24485         1      25.0
       ...
     24557         1      25.0
*MAT_GAS_MIXTURE
$       MID
         2
$       Cv1       Cv2       Cv3       Cv4       Cv5       Cv6       Cv7       Cv8
    482.00   2038.30    774.64       0.0       0.0       0.0       0.0
$       Cp1       Cp2       Cp3       Cp4       Cp5       Cp6       Cp7       Cp8
    666.67   2500.00   1071.40       0.0       0.0       0.0       0.0
$ The next card specifies that gas 1 (background air) occupies PID 1 at time 0.
*ALE_MULTI-MATERIAL_GROUP
$       SID   SIDTYPE
         1         1
         2         1
*DEFINE_VECTOR
$    VECTID      XTAIL     YTAIL     ZTAIL     XHEAD     YHEAD     ZHEAD
         1        0.0       0.0       0.0       0.0       1.0       0.0
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|....8
```

*SECTION_SEATBELT

Purpose: Define section properties for the seat belt elements. This card is required for the *PART Section. Currently, only the ID is required.

Card 1	1	2	3	4	5	6	7	8
Variable	SECID	AREA	THICK					
Type	A8	F	F					

VARIABLE	DESCRIPTION
SECID	Section ID. A unique number or label not exceeding 8 characters must be specified.
AREA	Optional Area of cross-section used in contact
THICK	Optional Thickness used in contact

Remarks:

1. Seatbelt elements are not implemented for implicit calculations.

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$  *SECTION_SEATBELT
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Define a seat belt section that is referenced by part 10.  Nothing
$  more than the sid is required.
$
*SECTION_SEATBELT
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$      sid
       111
$
$
*PART
Seatbelt material
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$      pid       sid       mid     eosid      hgid    adpopt
        10       111       220
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

***SECTION_SHELL_**{*OPTION*}

Available options include:

<BLANK>

ALE

EFG

THERMAL

such that the keyword cards appear:

***SECTION_SHELL**

***SECTION_SHELL_ALE**

***SECTION_SHELL_EFG**

***SECTION_SHELL_THERMAL**

Purpose: Define section properties for shell elements.

Card 1

Variable	SECID	ELFORM	SHRF	NIP	PROPT	QR/IRID	ICOMP	SETYP
Type	A8	I	F	F	F	F	I	I
Default	none		1.0	2	0.0	0.0	0	1
Remarks		1,7,8						

Card 2

Variable	T1	T2	T3	T4	NLOC	MAREA	IDOF	EDGSET
Type	F	F	F	F	F	F	F	I
Default	0.0	T1	T1	T1	0.0	0.0	0.0	

Remarks							7	8

Optional Section Cards if ICOMP=1. Define NIP angles putting 8 on each card.

Card	1	2	3	4	5	6	7	8
Variable	B1	B2	B3	B4	B5	B6	B7	B8
Type	F	F	F	F	F	F	F	F

Optional Section Card for ALE option.
Also see *CONTROL_ALE and *ALE_SMOOTHING.

Card	1	2	3	4	5	6	7	8
Variable	AFAC	BFAC	CFAC	DFAC	EFAC	START	END	AAFAC
Type	F	F	F	F	F	F	F	F

Optional Section Card for EFG option.
Also see *CONTROL_EFG.

Card	1	2	3	4	5	6	7	8
Variable	DX	DY	ISPLINE	IDILA				
Type	F	F	I	I				
Default	1.01	1.01	0	0				

Optional Section Card for THERMAL option.

Card	1	2	3	4	5	6	7	8
Variable	ITHELFM							
Type	I							
Default	0							

Define the next 3 cards if and only if ELFORM=101,102,103,104 or 105.
Also see Appendix C

Card	1	2	3	4	5	6	7	8
Variable	NIPP	NXDOF	IUNF	IHGF	ITAJ	LMC	NHSV	ILOC
Type	I	I	I	I	I	I	I	I
Default	0	0	0	0	0	0	0	0

Define NIPP cards according to the following format
Also see Appendix C

Card	1	2	3	4	5	6	7	8
Variable	XI	ETA	WGT					
Type	F	F	F					
Default	None	None	None					

Define LMC property parameters using 8 parameters per card.
Also see Appendix C

Card	1	2	3	4	5	6	7	8
Variable	P1	P2	P3	P4	P5	P6	P7	P8
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

VARIABLE	DESCRIPTION
SECID	Section ID. SECID is referenced on the *PART card. A unique number or label not exceeding 8 characters must be specified.
ELFORM	Element formulation options, see Remarks 1 and 2 below: EQ.1: Hughes-Liu, EQ.2: Belytschko-Tsay, EQ.3: BCIZ triangular shell, EQ.4: C^0 triangular shell, EQ.5: Belytschko-Tsay membrane, EQ.6: S/R Hughes-Liu, EQ.7: S/R co-rotational Hughes-Liu, EQ.8: Belytschko-Leviathan shell, EQ.9: Fully integrated Belytschko-Tsay membrane, EQ.10: Belytschko-Wong-Chiang, EQ.11: Fast (co-rotational) Hughes-Liu, EQ.12: Plane stress (x-y plane), EQ.13: Plane strain (x-y plane), EQ.14: Axisymmetric solid (y-axis of symmetry) - area weighted,

VARIABLE	DESCRIPTION
	EQ.15: Axisymmetric solid (y-axis of symmetry) - volume weighted, EQ.16: Fully integrated shell element (very fast), EQ.17: Fully integrated DKT, triangular shell element, EQ.18: Fully integrated linear DK quadrilateral/triangular shell EQ.20: Fully integrated linear assumed strain C^0 shell (See Remarks). EQ.21: Fully integrated linear assumed strain C^0 shell (5 DOF). EQ.22: Linear shear panel element (3 DOF per node, see remarks) EQ.23: 8-node quadratic quadrilateral shell (under development) EQ.24: 6-node quadratic triangular shell (under development) EQ.25: Belytschko-Tsay shell with thickness stretch. EQ.26: Fully integrated shell with thickness stretch. EQ.27: C^0 triangular shell with thickness stretch. EQ.41: Mesh-free shell local approach. EQ.42: Mesh-free shell global approach. EQ.43: Mesh-free plane strain formulation (x-y plane). EQ.44: Mesh-free axisymmetric solid formulation (y-axis of symmetry). EQ.46: Cohesive element for two-dimensional plane strain, plane stress, and area-weighted axisymmetric problems (type 14 shells). EQ.47: Cohesive element for two-dimensional volume-weighted axisymmetric problems (use with type 15 shells). EQ.99: Simplified linear element for time-domain vibration studies. See Remark 4 below. GT.100.AND.LT.106: User defined shell
	The type 18 element is only for linear static and normal modes. It can also be used for linear springback in sheet metal stamping.
	Note that the 2D and 3D element types must not be mixed, and different types of 2D elements must not be used together. For example, 2D axisymmetric calculations can use either element types 14 or 15 but these element types must not be mixed together. Likewise, the plane strain element type must not be used with either the plane stress element or the axisymmetric element types. In 3D, the different shell elements types, i.e., 1-11 and 16, can be freely mixed together.
SHRF	Shear correction factor which scales the transverse shear stress. The shell formulations in LS-DYNA, with the exception of the BCIZ and DK elements, are based on a first order shear deformation theory that yields constant transverse shear strains which violates the condition of zero traction on the top and bottom surfaces of the shell. The shear correction factor is attempt to compensate for this error. A suggested value is 5/6 for isotropic materials. This value is incorrect for sandwich or laminated shells; consequently, laminated/sandwich shell theory is now an option in some of the constitutive models, e.g., material types 22, 54, and 55.

NIP	Number of through thickness integration points. Either Gauss (default) or Lobatto integration can be used. The flag for Lobatto integration can be set on the control card, *CONTROL_SHELL. The location of the Gauss and Lobatto integration points are tabulated below.

VARIABLE	DESCRIPTION
	EQ.0.0: set to 2 integration points for shell elements. EQ.1.0: 1 point (no bending) EQ.2.0: 2 point EQ.3.0: 3 point EQ.4.0: 4 point EQ.5.0: 5 point EQ.6.0: 6 point EQ.7.0: 7 point EQ.8.0: 8 point EQ.9.0: 9 point EQ.10.: 10 point GT.10.: trapezoidal or user defined rule Through thickness integration for the two-dimensional elements (options 12-15 above) is not meaningful; consequently, the default is equal to 1 integration point. Fully integrated two-dimensional elements are available for options 13 and 15 by setting NIP equal to a value of 4 corresponding to a 2 by 2 Gaussian quadrature. If NIP is 0 or 1 and the *MAT_SIMPLIFIED_JOHNSON_COOK model is used, then a resultant plasticity formulation is activated. NIP is always set to 1 if a constitutive model based on resultants is used.
PROPT	Printout option (***NOT ACTIVE***): EQ.1.0: average resultants and fiber lengths, EQ.2.0: resultants at plan points and fiber lengths, EQ.3.0: resultants, stresses at all points, fiber lengths.
QR/IRID	Quadrature rule or Integration rule ID, see *INTEGRATION_SHELL: LT.0.0: absolute value is specified rule number, EQ.0.0: Gauss/Lobatto (up to 10 points are permitted), EQ.1.0: trapezoidal, *not recommend for accuracy reasons*.
ICOMP	Flag for orthotropic/anisotropic layered composite material model. This option applies to material types 22, 23, 33, 34, 36, 40, 41-50, 54-56, 58, 59, 103, 116, and 194. EQ.1: a material angle in degrees is defined for each through thickness integration point. Thus, each layer has one integration point.
SETYP	2D solid element type: Defined for ELFORM 13, 14, and 15. EQ.1: Lagrangian EQ.2: Eulerian (single material with voids) EQ.3: ALE
T1	Shell thickness at node n_1, unless the thickness is defined on the *ELEMENT_SHELL_*OPTION* card.

VARIABLE	DESCRIPTION
T2	Shell thickness at node n_2, see comment for T1 above.
T3	Shell thickness at node n_3, see comment for T1 above.
T4	Shell thickness at node n_4, see comment for T1 above.
NLOC	Location of reference surface for three dimensional shell elements. If nonzero, the offset distance from the plane of the nodal points to the reference surface of the shell in the direction of the shell normal vector is a value $offset = -0.50 \times NLOC \times (average\ shell\ thickness)$. This offset is not considered in the contact subroutines unless CNTCO is set to 1 in *CONTROL_SHELL. Alternatively, the offset can be specified by using the OFFSET option in the *ELEMENT_SHELL input section. EQ. 1.0: top surface, EQ. 0.0: mid-surface (default), EQ.-1.0: bottom surface.
MAREA	Non-structural mass per unit area. This is additional mass which comes from materials such as carpeting. This mass is not directly included in the time step calculation. Another and often more convenient alternative for defining distributed mass is by the option: *ELEMENT_MASS_ PART, which allows additional non-structural mass to be distributed by an area weighted distribution to all nodes of a given part ID).
IDOF	Treatment of through thickess strain. LT.0: Same as IDOF.EQ.3 but the contact pressure is averaged over a time –IDOF in order to reduce noise and thus improve stability. EQ.1: The thickness field is continuous across the element edges for metalforming applications. This option applies to element types 25 and 26. EQ.2: The thickness field is discontinuous across the element edges. This is necessary for crashworthiness simulations due to shell intersections, sharp included angles, and non-smooth deformations. This option applies to element types 25, 26 and 27 and is mandatory for element 27. This is the default for these element types. EQ.3: The thickness strain is governed by the contact stress, meaning that the strain is adjusted for the through thickness stress to equilibrate the contact pressure. This option applies to element types 2 and 16.
EDGSET	Edge node set required for shell type seatbelts. Input an ordered set of nodes along one of the transverse edges of a seatbelt. If there is no retractor associated with a belt, the node set can be on either edge. If the retractor exists, the edge must be on the retractor side and input in the same sequence of retractor node set. Therefore, another restriction on the seatbelt usage is that each belt has its own section definition and,

therefore, a unique part ID. See Figure 13.2 in the section *ELEMENT _SEATBELT for additional clarification.

B1 β_1, material angle at first integration point

B2 β_2, material angle at second integration point

B3 β_3, material angle at third integration point

VARIABLE	DESCRIPTION
.	.
.	.
B8	β_8, material angle at eighth integration point
.	.
.	.
Bnip	β_{nip}, material angle at nipth integration point
AFAC	Smoothing weight factor - Simple average: EQ.-1: turn smoothing off.
BFAC	Smoothing weight factor - Volume weighting
CFAC	Smoothing weight factor - Isoparametric
DFAC	Smoothing weight factor - Equipotential
EFAC	Smoothing weight factor - Equilibrium
START	Start time for smoothing
END	End time for smoothing
AAFAC	ALE advection factor
DX,DY	Normalized dilation parameters of the kernel function in X and Y directions. The normalized dilation parameters of the kernel function are introduced to provide the smoothness and compact support properties on the construction of the mesh-free shape functions. Values between 1.0 and 2.0 are recommended. Values smaller than 1.0 are not allowed. Larger values will increase the computation time and will sometimes result in a divergence problem. See Remark 6.
ISPLINE	Replace the choice for the EFG kernel functions definition in *CONTROL_EFG. This allows users to define different ISPLINE in different sections.
IDILA	Replace the choice for the normalized dilation parameter definition in *CONTROL_EFG. This allows users to define different IDILA in different sections.
ITHELFM	Thermal shell formulation .EQ.0: Default is governed by TSHELL on *CONTROL_SHELL .EQ.1: Thick thermal shell .EQ.2: Thin thermal shell
NIPP	Number of in-plane integration points for user-defined shell (0 if resultant/discrete element)

NXDOF	Number of extra degrees of freedom per node for user-defined shell

VARIABLE	DESCRIPTION
IUNF	Flag for using nodal fiber vectors in user-defined shell: EQ.0: Nodal fiber vectors are not used. EQ.1: Nodal fiber vectors are used.
IHFG	Flag for using hourglass stabilization (NIPP.GT.0) EQ.0: Hourglass stabilization is not used EQ.1: LS-DYNA hourglass stabilization is used EQ.2: User-defined hourglass stabilization is used EQ.3: Same as 2, but the resultant material tangent moduli are passed
ITAJ	Flag for setting up finite element matrices (NIPP.GT.0) EQ.0: Set up matrices wrt isoparametric domain EQ.1: Set up matrices wrt physical domain
LMC	Number of property parameters
NHSV	Number of history variables
ILOC	Coordinate system option: EQ.0: Pass all variables in LS-DYNA local coordinate system EQ.1: Pass all variables in global coordinate system
XI	First isoparametric coordinate
ETA	Second isoparametric coordinate
WGT	Isoparametric weight
PI	Ith property parameter

GAUSS INTEGRATION RULE					
NUMBER OF GAUSS POINT	1 POINT	2 POINT	3 POINT	4 POINT	5 POINT
#1	.0	-.5773503	.0	-.8611363	.0
#2		+.5773503	-.7745967	-.3399810	-.9061798
#3			+.7745967	+.3399810	-.5384693
#4				+.8622363	+.5384693
#5					+.9061798
NUMBER OF GAUSS POINT	6 POINT	7 POINT	8 POINT	9 POINT	10 POINT
#1	-.9324695	-.9491080	-.9702896	-.9681602	-.9739066
#2	-.6612094	-.7415312	-.7966665	-.8360311	-.8650634
#3	-.2386192	-.4058452	-.5255324	-.6133714	-.6794096
#4	+.2386192	.0	-.1834346	-.3242534	-.4333954
#5	+.6612094	+.4058452	+.1834346	0.0	-.1488743
#6	+.9324695	+.7415312	+.5255324	+.3242534	+.1488743
#7		+.9491080	+.7966665	+.6133714	+.4333954
#8			+.9702896	+.8360311	+.6794096
#9				+.9681602	+.8650634
#10					+.9739066

Location of through thickness Gauss integration points. The coordinate is referenced to the shell midsurface at location 0. The inner surface of the shell is at -1 and the outer surface is at +1.

LOBATTO INTEGRATION RULE					
NUMBER OF INTEG. POINT	-	-	3 POINT	4 POINT	5 POINT
#1			.0	-1.0	.0
#2			-1.0	-.4472136	-1.0
#3			+1.0	+.4472136	-.6546537
#4				+1.0	+.6546537
#5					+1.0
NUMBER OF INTEG. POINT	6 POINT	7 POINT	8 POINT	9 POINT	10 POINT
#1	-1.0	-1.0	-1.0	-1.0	-1.0
#2	-.7650553	-.8302239	-.8717401	-.8997580	-.9195339
#3	-.2852315	-.4688488	-.5917002	-.6771863	-.7387739
#4	+.2852315	.0	-.2092992	-.3631175	-.4779249
#5	+.7650553	+.4688488	+.2092992	.0	-.1652790
#6	+1.0	+.8302239	+.5917002	+.3631175	+.1652790
#7		+1.0	+.8717401	+.6771863	+.4779249
#8			+1.0	+.8997580	+.7387739
#9				+1.0	+.9195339
#10					+1.0

Location of through thickness Lobatto integration points. The coordinate is referenced to the shell midsurface at location 0. The inner surface of the shell is at -1 and the outer surface is at +1.

Remarks:

1. The default shell formulation is 2 unless overridden by THEORY in *control_shell.

   For implicit calculations the following element formulations are implemented:

   EQ.1: Hughes-Liu,
   EQ.2: Belytschko-Tsay (default),
   EQ.6: S/R Hughes-Liu,
   EQ.10: Belytschko-Wong-Chiang,
   EQ.12: Plane stress (x-y plane),
   EQ.13: Plane strain (x-y plane)
   EQ.15: Axisymmetric solid (y-axis of symmetry) - volume weighted,
   EQ.16: Fully integrated shell element,
   EQ.17: Fully integrated DKT, triangular shell element,
   EQ.18: Taylor 4-node quadrilateral and 3-node triangle (linear only)
   EQ.20: Wilson 3 & 4-node DSE quadrilateral (linear only)
   EQ.21: Fully integrated linear assumed strain C0 shell (5 DOF).

EQ.22: Linear shear panel element (3 DOF per node)

EQ.25: Belytschko-Tsay shell with thickness stretch.
EQ.26: Fully integrated shell element with thickness stretch.
EQ.27: Triangle with thickness stretch.

If another element formulation is requested for an implicit analysis, LS-DYNA will substitute one of the above in place of the one chosen.

2. The linear elements consist of an assembly of membrane and plate elements. The elements have six d.o.f. per node and can therefore be connected to beams, or used in complex shell surface intersections. All elements possess the required zero energy rigid body modes and have exact constant strain and curvature representation, i.e. they pass all the first order patch tests. In addition, the elements have behavior approaching linear bending (cubic displacement) in the plate-bending configuration.

   a. The membrane component of all elements is based on an 8-node/6-node isoparametric mother element which incorporates nodal in-plane rotations through cubic displacement constraints of the sides [Taylor 1987; Wilson 2000].

   b. The plate component of element 18 is based on the Discrete Kirchhoff Quadrilateral (DKQ) [Batoz 1982]. Because the Kirchhoff assumption is enforced, the DKQ is transverse shear rigid and can only be used for thin shells. No transverse shear stress information is available. The triangle is based on a degeneration of the DKQ. This element sometimes gives slightly lower eigenvalues when compared with element type 20.

   c. The plate component of element 20 is based on the 8-node serendipity element. At the mid-side, the parallel rotations and transverse displacements are constrained and the normal rotations are condensed to yield a 4-node element. The element is based on thick plate theory and is recommended for thick and thin plates.

   d. The quadrilateral elements contain a warpage correction using rigid links.

   e. The membrane component of element 18 has a zero energy mode associated with the in-plane rotations. This is automatically suppressed in a non-flat shell by the plate stiffness of the adjacent elements. Element 20 has no spurious zero energy modes.

3. The linear shear panel element resist tangential in plane shearing along the four edges and can only be used with the elastic material constants of *MAT_ELASTIC. Membrane forces and out-of-plane loads are not resisted.

4. Element type 99 is intended for vibration studies carried out in the time domain. These models may have very large numbers of elements and may be run for relatively long durations. The purpose of this element is to achieve substantial CPU savings. This is achieved by imposing strict limitations on the range of applicability, thereby simplifying the calculations:

   - Elements must be rectangular; all edges must parallel to the global X-, Y- or Z-axis;
   - Small displacement, small strain, negligible rigid body rotation;

- Elastic material only

If these conditions are satisfied, the performance of the element is similar to the fully integrated shell (ELFORM=16) but at less CPU cost than the default Belytschko-Tsay shell element (ELFORM=2). Single element torsion and in-plane bending modes are included; meshing guidelines are the same as for fully integrated shell elements.

No damping is included in the element formulation (e.g. volumetric damping). It is strongly recommended that damping be applied, e.g. *DAMPING_PART_MASS or *DAMPING_FREQUENCY_RANGE.

5. SHELL_EFG formulation is only available for the explicit analysis.

6. Loads, lumped masses, discrete element stiffnesses, etc. in axisymmetric simulations are interpreted as values per unit length (circumferentially) in the case where shell formulation 14 is invoked and per unit radian in the case where shell formulation 15 is used.

7. Shell element formulation 25 and 26 are the Belytschko-Tsay element and fully integrated shell element with two additional degrees of freedom that allows a linear variation of strain through the thickness. By default, the thickness field is continuous across the element edges implying that there can be no complex intersections since this would lock up the structure. It assumes a relatively flat surface and is intended primarily for sheets in metal forming. By specifying IDOF=2, the thickness field is decoupled between elements which makes the element suited for crash. If there are any thickness stretch triangles (formulation 27), IDOF must be set to 2.

8. Users must input a set of nodes along one of the transverse edges of a seatbelt. If there is no retractor associated with a belt, the node set can be on either edge. If the retractor exists, the edge should be on the retractor side and input in the same sequence of retractor node set. Therefore, another restriction on the seatbelt usage is each belt has its own section definition and a different part.

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$   *SECTION_SHELL
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Define a shell section that specifies the following:
$     elform = 10  Belytschko-Wong-Chiang shell element formulation.
$        nip = 3   Three through the shell thickness integration points.
$    t1 - t4 = 2.0 A shell thickness of 2 mm at all nodes.
$
*SECTION_SHELL
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$      sid    elform      shrf       nip     propt   qr/irid     icomp
         1        10              3.0000
$
$       t1        t2        t3        t4      nloc
       2.0       2.0       2.0       2.0
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
```

***SECTION_SOLID_**{*OPTION*}

Available options include:

<BLANK>

ALE

EFG

such that the keyword cards appear:

***SECTION_SOLID**

***SECTION_SOLID_ALE**

***SECTION_SOLID_EFG**

Purpose: Define section properties for solid continuum and fluid elements.

Card 1 define for all options

Card 1	1	2	3	4	5	6	7	8
Variable	SECID	ELFORM	AET					
Type	A8	I	I					
Remark		1, 2						

Card 2 define only for the ALE option.
Also see *ALE_SMOOTHING for the smoothing definition.

Card 2	1	2	3	4	5	6	7	8
Variable	AFAC	BFAC	CFAC	DFAC	START	END	AAFAC	
Type	F	F	F	F	F	F	F	

Define only for the EFG option.
Also see *CONTROL_EFG. See Remark 7.

Card 2	1	2	3	4	5	6	7	8
Variable	DX	DY	DZ	ISPLINE	IDILA	IEBT	IDIM	TOLDEF
Type	F	F	F	I	I	I	I	F
Default	1.01	1.01	1.01	0	0	1	1	0.01

Define if and only if ELFORM=101,102,103,104 or 105.
Also see Appendix C

Card	1	2	3	4	5	6	7	8
Variable	NIP	NXDOF	IHGF	ITAJ	LMC	NHSV		
Type	I	I	I	I	I	I		
Default	0	0	0	0	0	0		

Define NIP cards according to the following format.
Also see Appendix C

Card	1	2	3	4	5	6	7	8
Variable	XI	ETA	ZETA	WGT				
Type	F	F	F	F				
Default	None	None	None	None				

Define LMC property parameters using 8 parameters per card.
Also see Appendix C

Card	1	2	3	4	5	6	7	8
Variable	P1	P2	P3	P4	P5	P6	P7	P8
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

VARIABLE	DESCRIPTION
SECID	Section ID. SECID is referenced on the *PART card. A unique number or label not exceeding 8 characters must be specified.
ELFORM	Element formulation options, (see Remark 2 below): EQ.-2: fully integrated S/R solid intended for elements with poor aspect ratio, accurate formulation. EQ.-1: fully integrated S/R solid intended for elements with poor aspect ratio, efficient formulation. EQ.0: 1 point corotational for *MAT_MODIFIED_HONEYCOMB. See remark 3. EQ.1: constant stress solid element (default), EQ.2: fully integrated S/R solid. See remark 4 below, EQ.3: fully integrated quadratic 8 node element with nodal rotations, EQ.4: S/R quadratic tetrahedron element with nodal rotations, EQ.5: 1 point ALE, EQ.6: 1 point Eulerian, EQ.7: 1 point Eulerian ambient, EQ.8: acoustic, EQ.9: 1 point corotational for *MAT_MODIFIED_HONEYCOMB. See remark 3. EQ.10: 1 point tetrahedron. EQ.11: 1 point ALE multi-material element EQ.12: 1 point integration with single material and void. EQ.13: 1 point nodal pressure tetrahedron. See remark 14 below. EQ.14: 8 point acoustic EQ.15: 2 point pentahedron element. EQ.16: 4 or 5 point 10-noded tetrahedron (See Remark 13) EQ.17: 10-noded composite tetrahedron EQ.18: 8 point enhanced strain solid element for linear statics only EQ.19: 4 point cohesive element EQ.20: 4 point cohesive elements with offsets for use with shells EQ.41: Mesh-free solid formulation

VARIABLE	DESCRIPTION
	EQ.99: simplified linear element for time-domain vibration studies. See remarks. GT.100.and.LT.106: User defined solid
AET	Ambient Element type: Can be defined for ELFORM 7, 11 and 12. EQ.1: temperature (not currently available), EQ.2: pressure and temperature (not currently available), EQ.3: pressure outflow, EQ.4: pressure inflow. (Default for ELFORM 7), EQ.5: receptor for blast load (see *LOAD_BLAST_ENHANCED, available only for ELFORM=11).
AFAC	Smoothing weight factor - Simple average: EQ.-1: turn smoothing off.
BFAC	Smoothing weight factor - Volume weighting
CFAC	Smoothing weight factor - Isoparametric
DFAC	Smoothing weight factor - Equipotential
START	Start time for smoothing
END	End time for smoothing
AAFAC	ALE advection factor
DX, DY, DZ	Normalized dilation parameters of the kernel function in X, Y and Z directions. The normalized dilation parameters of the kernel function are introduced to provide the smoothness and compact support properties on the construction of the mesh-free shape functions. Values between 1.0 and 1.5 are recommended. Values smaller than 1.0 are not allowed. Larger values will increase the computation time and will sometimes result in a divergence problem. See Remark 6.
ISPLINE	Replace the choice for the EFG kernel functions definition in *CONTROL_EFG. This allows users to define different ISPLINE in different sections. EQ.0: Cubic spline function (default). EQ.1: Quadratic spline function. EQ.2: Cubic spline function with circular shape.
IDILA	Replace the choice for the normalized dilation parameter definition in *CONTROL_EFG. This allows users to define different IDILA in different sections. EQ.0: Maximum distance based on the background elements. EQ.1: Maximum distance based on surrounding nodes.

VARIABLE	DESCRIPTION
IEBT	Essential boundary condition treatment: See Remark 9 and 10. EQ. 1: Full transformation method (default) EQ.-1: (w/o transformation) EQ. 2: Mixed transformation method EQ. 3: Coupled FEM/EFG method EQ. 4: Fast transformation method EQ.-4: (w/o transformation) EQ.5: Fluid particle method for E.O.S and *MAT_ELASTIC_FLUID materials
IDIM	Domain integration method: See Remark 11. EQ.1: Local boundary integration (default) EQ.2: Two-point Gauss integration EQ.3: Improved Guass integration for IEBT=4 or -4
TOLDEF	Deformation tolerance for the activation of adaptive EFG Semi-Lagrangian and Eulerian kernel. See Remark 12. = 0.0: Lagrangian kernel > 0.0: Semi_Lagrangian kernel < 0.0: Eulerian kernel
NIP	Number of integration points for user-defined solid (0 if resultant/discrete element)
NXDOF	Number of extra degrees of freedom per node for user-defined solid
IHGF	Flag for using hourglass stabilization (NIP.GT.0) EQ.0: Hourglass stabilization is not used EQ.1: LS-DYNA hourglass stabilization is used EQ.2: User-defined hourglass stabilization is used EQ.3: Same as 2, but the resultant material tangent modulus is passed
ITAJ	Flag for setting up finite element matrices (NIP.GT.0) EQ.0: Set up matrices wrt isoparametric domain EQ.1: Set up matrices wrt physical domain
LMC	Number of property parameters
NHSV	Number of history variables
XI	First isoparametric coordinate
ETA	Second isoparametric coordinate
ZETA	Third isoparametric coordinate

VARIABLE	DESCRIPTION
WGT	Isoparametric weight
PI	Ith property parameter

Remarks:

1. The keyword *CONTROL_SOLID activates automatic sorting of tetrahedron and pentahedron elements into type 10 and 15 element formulation, respectively. These latter elements are far more stable than the degenerate solid element. The sorting in performed internally and is transparent to the user.

2. For implicit calculations the following element choices are implemented:
   EQ.1: constant stress solid element,
   EQ.2: fully integrated S/R solid. See remark 5 below,
   EQ.3: fully integrated 8 node solid with rotational DOFs,
   EQ.4: fully integrated S/R 4 node tetrahedron with rotational DOFs,
   EQ.10: 1 point tetrahedron.
   EQ.13: 1 point nodal pressure tetrahedron.
   EQ.15: 2 point pentahedron element.
   EQ.16: 5 point 10 noded tetrahedron
   EQ.18: 8 point enhanced strain solid element for linear statics only,

   If another element formulation is requested, LS-DYNA will substitute, when possible, one of the above in place of the one chosen. The type 1 element, constant stress, is generally much more accurate than the type 2 element, the selective reduced integrated element for implicit problems.

3. Element formulations 0 and 9, applicable only to *MAT_MODIFIED_HONEYCOMB, behave essentially as nonlinear springs so as to permit severe distortions sometimes seen in honeycomb materials. In formulation 0, the local coordinate system follows the element rotation whereas in formulation 9, the local coordinate system is based on axes passing through the centroids of the element faces. Formulation 0 is preferred for severe shear deformation where the barrier is fixed in space. If the barrier is attached to a moving body, which can rotate, then formulation 9 is usually preferred.

4. The selective reduced integrated solid element, element type 2, assumes that pressure is constant throughout the element to avoid pressure locking during nearly incompressible flow. However, if the element aspect ratios are poor, shear locking will lead to an excessively stiff response. A better choice, given poor aspect ratios, is the one point solid element which work well for implicit and explicit calculations. For linear statics, the type 18 enhanced strain element works well with poor aspect ratios. Please note that highly distorted elements should always be avoided since excessive stiffness will still be observed even in the enhanced strain formulations.

5. Element type 99 is intended for vibration studies carried out in the time domain. These models may have very large numbers of elements and may be run for relatively long durations. The purpose of this element is to achieve substantial CPU savings. This is

achieved by imposing strict limitations on the range of applicability, thereby simplifying the calculations:

- Elements must be cubed; all edges must parallel to the global X-, Y- or Z-axis;
- Small displacement, small strain, negligible rigid body rotation;
- Elastic material only

If these conditions are satisfied, the performance of the element is similar to the fully integrated S/R solid (ELFORM=2) but at less CPU cost than the default solid element (ELFORM=1). Single element bending and torsion modes are included, so meshing guidelines are the same as for fully integrated solids – e.g. relatively thin structures can be modeled with a single solid element through the thickness if required. Typically, the CPU requirement per element-cycle is roughly two thirds that of the default solid element.

No damping is included in the element formulation (e.g. volumetric damping). It is strongly recommended that damping be applied, e.g. *DAMPING_PART_MASS or *DAMPING_FREQUENCY_RANGE.

6. The current EFG formulation performs automatic sorting for finite element tetrahedral, pentahedron and hexahedral elements as the background mesh to identify the mesh-free geometry and provide the contact surface definition in the computation.

7. Element type 19 is a cohesive element. The tractions on the mid-surface defined as the mid-points between the nodal pairs 1-5, 2-6, 3-7, and 4-8 are functions of the differences of the displacements between nodal pairs interpolated to the four integration points. The initial volume of the cohesive element may be zero, in which case, the density may be defined in terms of the area of nodes 1-2-3-4. See Appendix A and the user material description for additional details.

   The tractions are calculated in the local coordinate system defined at the centroid of the element, see the Figure below. Defining the rotation matrix from the local to the global coordinate system at time t as $R(t)$, the initial coordinates as X, and the current coordinates as x, the displacements at an integration point are

$$\Delta u = R^T(t)\Delta x - R^T(0)\Delta X$$

$$\Delta x = \sum_{i=1}^{4} N_i(s,t)\Delta x_{i+4,i}$$

$$\Delta X = \sum_{i=1}^{4} N_i(s,t)\Delta X_{i+4,i}$$

   The forces are obtained by integrating the tractions over the midsurface, and rotating them into the global coordinate system.

$$F_i = R(t)\sum_{g=1}^{4} t_g N_i(s_g,t_g)\det(J_g) \text{ for i=1,4 and } F_{i+4} = -F_i$$

8. Element type 20 is identical to element 19 but with offsets for use with shells. The element is assumed to be centered between two layers of shells on the cohesive element's

lower (1-2-3-4) and upper (5-6-7-8) surfaces. The offset distances for both shells are one half the initial thicknesses of the nodal pairs (1-5, 2-6, 3-7, and 4-8) separating the two shells. These offsets are used with the nodal forces to calculate moments that are applied to the shells.

9. The mixed transformation method, the coupled FEM/EFG method and the fast transformation method were implemented in EFG 3D solid formulation. These three new features were added to improve the efficiency on the imposition of essential boundary conditions and the transfer of real nodal values and generalized nodal values. The mixed transformation method is equivalent to the full transformation method with improved efficiency. The behavior of the coupled FEM/EFG method is between FEM and EFG. The fast transformation method provides the most efficient and robust results.

10. Current fluid particle formulation (IEBT=5) only supports for the 4-noded background element.

11. For compressible material like foam and soil, IDIM=1 (default) is recommended. For nearly incompressible material like metal and rubber, IDIM=2 is recommended.

12. This parameter is introduced to improve the negative volume problem usually seen in the large deformation analysis. For the same analysis, the larger value of Toldef, the earlier Semi-Lagrangian or Eulerian kernel is introduced into the EFG computation and more cpu time is expected. Value between 0.0 and 0.1 is suggested in the crashworthiness analysis. Semi-Lagrangian kernel is suggested for the solid materials and Eulerian kernel is suggested for the fluid and E.O.S. materials.

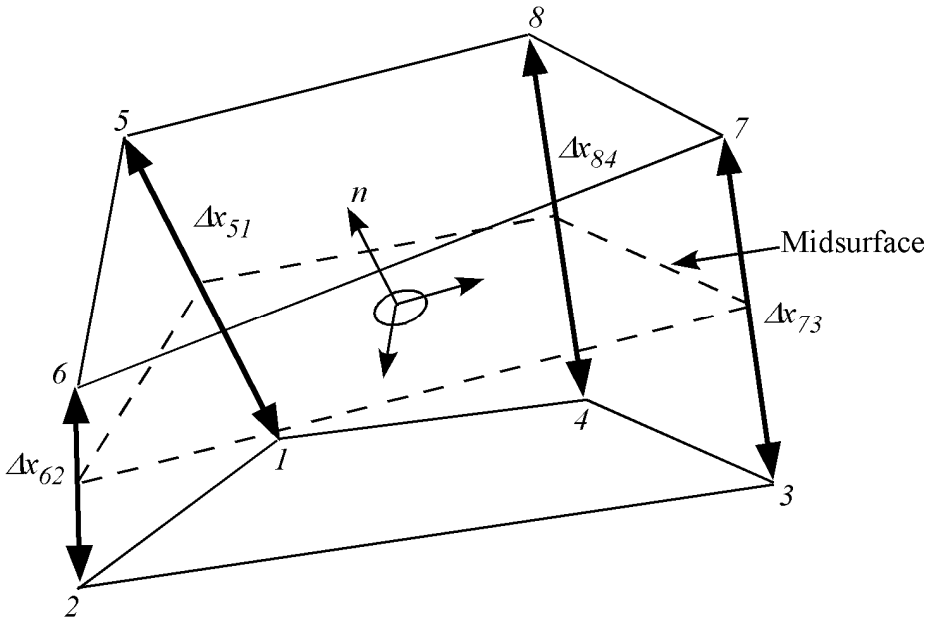


Figure 29.2

13. Formulations 16 and 17 are 10-noded, tetrahedral formulations. The parameter NIPTETS in *CONTROL_SOLID controls the number of integration points for these formulations.

Formulation 17 is generally preferred over formulation 16 because, unlike 16, the nodal weighting factors are equal and thus nodal forces from contact and applied pressures are distributed correctly.

When applying loads to 10-noded tetrahedrons via segments, no load will be applied to the midside nodes if the segments contain only corner nodes. When defining contact, it is recommended that *CONTACT_AUTOMATIC_... be used and the contact surface of the 10-noded tetrahedral part be specified by its part ID. In this manner, midside nodes receive contact forces.

If the 10-noded element connectivity is not defined in accordance with the figure shown in *ELEMENT_SOLID, the order of the nodes can be quickly changed via a permutation vector specified with *CONTROL_SOLID. If *ELEMENT_SOLID defines 4-noded tetrahedrons, you can easily convert to 10-noded tetrahedrons using the command *ELEMENT_SOLID_TET4TOTET10. Because the characteristic length of a 10-noded tetrahedron is half that of a 4-noded tetrahedron, the time step for the tetrahedrons will be smaller by a factor of 2. The parameter TET10 in 971, when set to 1 in *CONTROL_OUTPUT, causes the full 10-node connectivity to be written to the d3plot and d3part databases.

14. Element type 13 is identical with type 10 but with additional averaging of nodal pressures, which significantly lowers volumetric locking. Therefore, it is well suited for applications with incompressible and nearly incompressible material behavior, i.e. rubber materials or ductile metals with isochoric plastic deformations (e.g. bulk forming). Compared to the standard tetrahedron (type 10), a speed penalty of max. 25 % can be observed. Currently, material models *MAT_ 001, 003, 006, 024, 027, 077, 081, 082, 091, 092, 106, 120, 123, 124, 128, 129, 181, 183, 225, and 244 are fully supported in the R5 release of Version 971. For other materials this element behaves like the type 10 tetrahedron.

15. Elements -1 and -2 are identical with 2, the fully integrated solid, but accounted for elements with poor aspect ratio in order to reduce the transverse shear locking effects. Type -1 is an efficient implementation of type -2 but that may suffer from some hourglass tendencies due to inadequate loss of stiffness.

(Note: NODE_SET option is available starting with the R3 release of Version 971.)

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$  *SECTION_SOLID
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  A bolt modeled with solids was found to have excessive hourglassing.
$  Thus, the section (sid = 116) associated with the bolt part was used
$  to specify that a fully integrated Selectively-Reduced solid element
$  formulation be used to totally eliminate the hourglassing (elform = 2).
$
*SECTION_SOLID
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$      sid    elform
       116         2
$
*PART
bolts
$      pid       sid       mid     eosid      hgid    adpopt
        17       116         5
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

***SECTION_SPH_**{*OPTION*}

Available options include:

<BLANK>

TENSOR

USER

such that the keyword cards appear:

***SECTION_SPH**

***SECTION_SPH_TENSOR**

***SECTION_SPH_USER**

Purpose: Define section properties for SPH particles. For the **USER** option, see remark 3.

Card 1 define for all options

Card 1	1	2	3	4	5	6	7	8
Variable	SECID	CSLH	HMIN	HMAX	SPHINI	DEATH	START	IFORM
Type	A8	F	F	F	F	F	F	I
Default	none	1.2	0.2	2.0	0.0	1.e20	0.0	0

Card 2 define only for the TENSOR option.

Card 2	1	2	3	4	5	6	7	8
Variable	HXCSLH	HYCSLH	HZCSLH	HXINI	HYINI	HZINI		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
SECID	Section ID. SECID is referenced on the *PART card. A unique number or label not exceeding 8 characters must be specified.
CSLH	Constant applied to the smoothing length of the particles. The default value applies for most problems. Values between 1.05 and 1.3 are acceptable. Taking a value less than 1 is inadmissible. Values larger than 1.3 will increase the computational time. The default value is recommended.
HMIN	Scale factor for the minimum smoothing length (See Remark 1)
HMAX	Scale factor for the maximum smoothing length (See Remark 1)
SPHINI	Optional initial smoothing length (overrides true smoothing length). This option applies to avoid LS-DYNA to calculate the smoothing length during initialization. In this case, the variable CSLH doesn't apply.
DEATH	Time imposed SPH approximation is stopped.
START	Time imposed SPH approximation is activated.
IFORM	Element formulation option: EQ.0: default formulation, EQ.1: renormalization approximation EQ.2: symmetric formulation, EQ.3: symmetric renormalized approximation EQ.4: tensor formulation, EQ.5: fluid particle approximation EQ.6: fluid particle with renormalization approximation, EQ. 7: Total Lagrangian formulation EQ. 8: Total Lagrangian formulation with renormalization
HXCSLH	Constant applied for the smoothing length in the X direction for the tensor case.
HYCSLH	Constant applied for the smoothing length in the Y direction for the tensor case.
HZCSLH	Constant applied for the smoothing length in the Z direction for the tensor case.
HXINI	Optional initial smoothing length in the X direction for the tensor case (overrides true smoothing length)
HYINI	Optional initial smoothing length in the Y direction for the tensor case (overrides true smoothing length)

HZINI	Optional initial smoothing length in the Z direction for the tensor case (overrides true smoothing length)

Remarks:

1. The SPH processor in LS-DYNA uses a variable smoothing length. LS-DYNA computes the initial smoothing length, h_0, for each SPH part by taking the maximum of the minimum distance between every particle. Every particle has its own smoothing length which varies in time according to the following equation:

$$\frac{d}{dt}(h(t)) = h(t)div(v)$$

h(t) is the smoothing length, div(v) is the divergence of the flow. The smoothing length increases when particles separate from each other and reduces when the concentration of particles is important. It varies to keep the same number of particles in the neighborhood. The smoothing length varies between the minimum and maximum values

$$HMIN * h_0 < h(t) < HMAX * h_0$$

Defining a value of 1 for HMIN and 1 for HMAX will result in a constant smoothing length in time and space.

2. SPH is implemented for explicit applications.

3. The USER option allows the definition of customized subroutine for the variation of the smoothing length. A subroutine called *hdot* is defined in the file dyn21.F (Unix/linux) or lsdyna.f (Windows).

***SECTION_TSHELL**

Purpose: Define section properties for thick shell elements.

Card 1	1	2	3	4	5	6	7	8
Variable	SECID	ELFORM	SHRF	NIP	PROPT	QR	ICOMP	
Type	A8	I	F	F	F	F	I	
Default	none	1	1.0	2	1	0	0	

Optional Section Cards if ICOMP=1 define NIP angles putting 8 on each card.

Card 2…	1	2	3	4	5	6	7	8
Variable	B1	B2	B3	B4	B5	B6	B7	B8
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
SECID	Section ID. SECID is referenced on the *PART card. A unique number or label not exceeding 8 characters must be specified.
ELFORM	Element formulation: EQ.1: one point reduced integration (default), EQ.2: selective reduced 2 x 2 in plane integration. EQ.3: assumed strain 2 x 2 in plane integration, see remark below. EQ.5: assumed strain reduced integration
SHRF	Shear factor. A value of 5/6 is recommended.
NIP	Number of through shell thickness integration points: EQ.0: set to 2 integration points.
PROPT	Printout option: EQ.1.0: average resultants and fiber lengths, EQ.2.0: resultants at plan points and fiber lengths, EQ.3.0: resultants, stresses at all points, fiber lengths.

VARIABLE	DESCRIPTION
QR	Quadrature rule: LT.0.0: absolute value is specified rule number, EQ.0.0: Gauss (up to five points are permitted), EQ.1.0: trapezoidal, not recommended for accuracy reasons.
ICOMP	Flag for layered composite material mode: EQ.1: a material angle is defined for each through thickness integration point. For each layer one integration point is used.
B1	β_1, material angle at first integration point. The same procedure for determining material directions is use for thick shells that is used for the 4 node quadrilateral shell.
B2	β_2, material angle at second integration point
B3	β_3, material angle at third integration point
.	.
.	.
.	.
B8	β_8, material angle at eighth integration point
.	.
Bnip	β_{nip}, material angle at nipth integration point

Define as many cards as necessary until NIP points are defined.

Remarks:

1. Thick shell formulation types 3 and 5 uses a full three-dimensional stress update rather than the two-dimensional plane stress update of types 1 and 2. The type 3 and 5 elements are distortion sensitive and should not be used in situations where the elements are badly shaped. With element types 1 and 2, a single element through the thickness will capture bending response, but with element types 3 and 5 two are recommended to avoid excessive softness.

2. Element formulations 2, and 3 are available for implicit applications. If an element of type 1 is specified in an implicit analysis, it is internally switched to type 2.

*SENSOR

The keyword *SENSOR provides a convenient way of activating and deactivating boundary conditions, airbags, discrete elements, joints, contact, rigid walls, single point constraints, and constrained nodes. The sensor capability is new in the second release of version 971 and will evolve in later releases to encompass many more LS-DYNA capabilities and replace some of the existing capabilities such as the airbag sensor logic. The keyword control cards in this section are defined below in alphabetical order:

***SENSOR_CONTROL**

***SENSOR_DEFINE_CALC-MATH**

***SENSOR_DEFINE_ELEMENT**

***SENSOR_DEFINE_FORCE**

***SENSOR_DEFINE_NODE**

***SENSOR_SWITCH**

***SENSOR_SWITCH_CALC-LOGIC**

To define a sensor, three categories of sensor keyword cards are needed as shown in Figure 30.1.

1. Sensor definitions using the, *SENSOR_DEFINE keywords, which can be combined with the mathematical calculation cards, *SENSOR_DEFINE_CALC-MATH, for more complicated definitions. This category of keyword cards yield a numerical value to be referred by *SENSOR_SWITCH as a switching criterion.

   - *SENSOR_DEFINE
     This card defines the sensor location and types by node ID, element ID, or force-type ID.

   - *SENSOR_DEFINE_CALC-MATH
     This keyword card defines a new sensor ID obtained by performing mathematical calculations on the information from SENSOR_DEFINE definitions.

2. Sensor switching criterion definition using the, *SENSOR_SWITCH, keyword, which can be combined with the logical calculation cards, *SENSOR_SWITCH_CALC-LOGIC, for more complicated definitions. The logic value yielded by this category of cards can be referred by *SENSOR_CONTROL to determine if a status switch condition is met.

   - *SENSOR_SWITCH

     This card compares the numerical value from *SENSOR_DEFINE or *SENSOR_DEFINE_CALC-MATH with the given criterion to see if a switching condition is met.

- *SENSOR_SWITCH_CALC-LOGIC
  This card performs logical calculation on the information from SENSOR_SWITCH.

3. Sensor control definition, *SENSOR_CONTROL. This category of cards determines how and what to switch based on the logical values from *SENSOR_SWICH and/or *SENSOR_SWITCH_CALC-LOGIC.

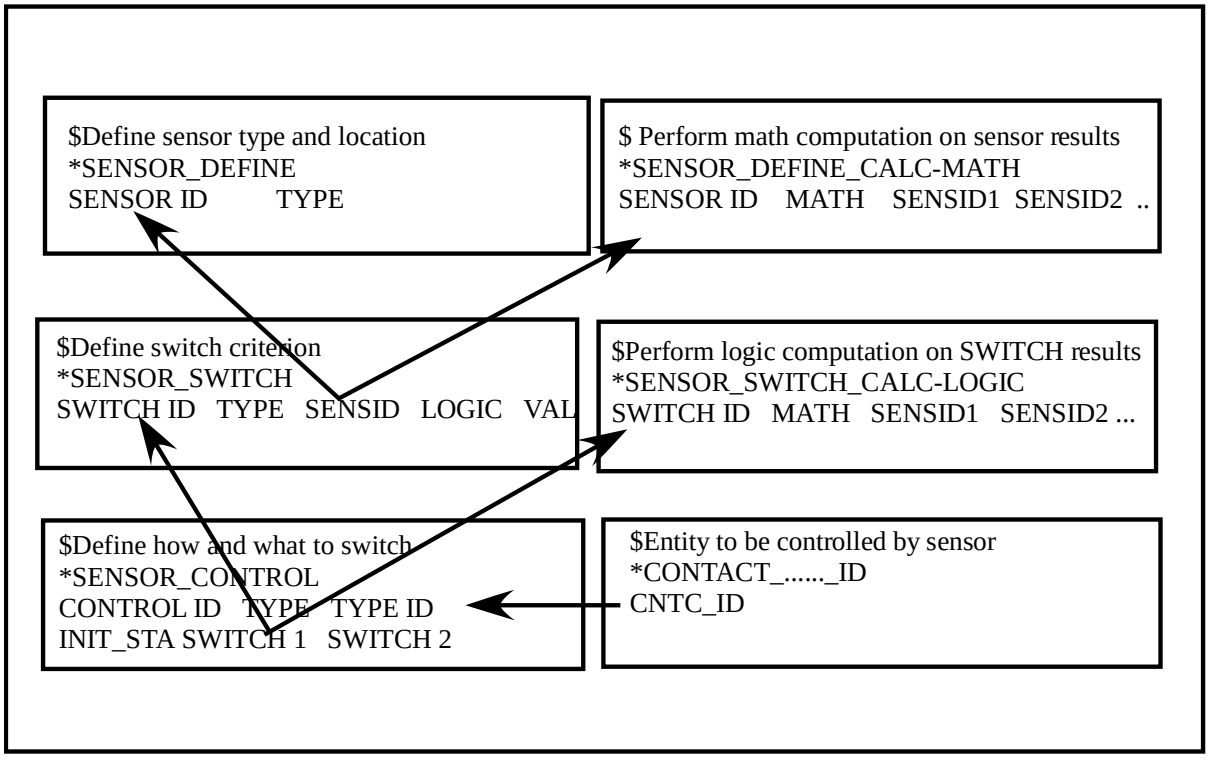


Figure 30.1. Relationship between sensor keyword definitions.

*SENSOR_CONTROL

Purpose: This command, based on the information of *SENSOR_SWITCH, controls the status, on or off, of an entity like *CONTACT, *AIRBAG.

Card 1

Variable	CNTLID	TYPE	TYPEID					
Type	I	A	I					

Card 2

Variable	INITSTT	SWIT1	SWIT2	SWIT3	SWIT4	SWIT5	SWIT6	SWIT7
Type	A	I	I	I	I	I	I	I

VARIABLE **DESCRIPTION**

CNTLID — Control ID.

TYPE — Entity to be controlled:
- EQ.AIRBAG: for *AIRBAG
- EQ.PRESC-MOT: for *BOUNDARY_PRESCRIBED MOTION
- EQ.DISC-ELE: for *ELEMENT_DISCRETE
- EQ.JOINT: for *CONSTRAINED_JOINT
- EQ.JOINTSTIF: for *CONSTRAINED_JOINT_STIFFNESS
- EQ.CONTACT: for *CONTACT
- EQ.RWALL: for *RIGID_WALL
- EQ.SPC: for *BOUNDARY_SPC
- EQ.SPOTWELD: for *CONSTRAINED_POINTS

TYPEID — ID of entity to be controlled.

INITSTT — Initial status:
- EQ.On:
- EQ.Off:

SWIT1 — ID of switch which will change the initial status after its condition is met.

SWITn — ID of nth switch which will change the status set by switch n-1 after its condition is met.

***SENSOR_DEFINE_CALC-MATH**

Purpose: Defines a new sensor with a unique ID. The values associated with this sensor are computed by performing mathematical calculations with the information obtained from sensors defined by the *SENSOR_DEFINE_*OPTION*.

Card	1	2	3	4	5	6	7	8
Variable	SENSID	CALC	SENS1	SENS2	SENS3	SENS4	SENS5	SENS6
Type	I	A	I	I	I	I	I	I

VARIABLE	DESCRIPTION
SENSID	Sensor ID.
CALC	Mathematical calculation, See Table 30.1.
SENSi	*i*th Sensor ID

CALC	DESCRIPTION	MATHEMATICAL EXPRESSION
ABSSUM	Absolute value of the sum of sensor values	|sens1+sens2+…|
MIN	The minimum of sensor values	Min (sens1, sens2, …)
MAX	The maximum of sensor values	Max (sens1, sens2, …)
MAXMAG	The maximum of magnitude of sensor values	Max (|sens1|, |sens2|…)
MINMAG	The minimum of the magnitude of sensor values	Min (|sens1|, |sens2|…)
MULTIPLY	Multiplication of sensor values; negative for division (performed left to right)	sens1* sens2* sens3…
SQRE	Summation of squared values of sensor values	Sens1^2+sens2^2…
SQRTSQRE	Square root of the sum of squared values	SQRT (sens1^2+sens2^2+…)
SQRT	Summation of square root of sensor values; negative for subtracting values	(sens1)**0.5+ (sens2) **0.5…
SUMABS	Summation of absolute sensor values	|sens1|+|sens2|+…
SUM	Summation of sensor values; negative for subtracting values	sens1+sens2+…

Table 30.1. Available mathematical functions.

*SENSOR_DEFINE_ELEMENT

Purpose: Define a strain gage type element sensor that checks the stress, strain, or resultant force of an element.

Card	1	2	3	4	5	6	7	8
Variable	SENSID	ETYPE	ELEMID	COMP	CTYPE	LAYER		
Type	I	A	I	A	A	A		

VARIABLE **DESCRIPTION**

SENSID — Sensor ID.

ETYPE — Element type:
- EQ.BEAM: beam element.
- EQ.SHELL: shell element
- EQ.SOLID: solid element
- EQ.DISC-ELE: discrete element

ELEMID — Element ID

COMP — Element type:
- EQ.XX: x-normal component for shells and solids
- EQ.YY: y-normal component for shells and solids
- EQ.ZZ: z-normal component for shells and solids
- EQ.XY: xy-shear component for shells and solids
- EQ.YZ: yz-shear component for shells and solids
- EQ.ZX: zx-shear component for shells and solids
- EQ:AXIAL: axial
- EQ:SHEARS: local s-direction
- EQ:SHEART: local t-direction
- EQ: : leave blank for discrete elements

CTYPE — Component type:
- EQ.STRAIN: strain component for shells and solids
- EQ.STRESS: stress component for shells and solids
- EQ.FORCE: force resultants for beams
- EQ.MOMENT: moment resultants for beams
- EQ.FORCE: discrete element force
- EQ.DLEN: change in length for discrete element

LAYER — Layer of integration point in shell element
- EQ.BOT: component at lower surface
- EQ.TOP: component at upper surface

*SENSOR_DEFINE_FORCE

Purpose: Define a force transducer type sensor.

Card	1	2	3	4	5	6	7	8
Variable	SENSID	FTYPE	TYPEID	VID	CRD			
Type	I	A	I	A/I	I			

VARIABLE	DESCRIPTION
SENSID	Sensor ID.
FTYPE	Force type. See Table 30.2.
TYPEID	ID defined in the associated KEYWORD command. See Table 30.2.
VID	Vector along which the forces is measured. EQ.X: x-direction in coordinate system CRD. EQ.Y: y-direction in coordinate system CRD. EQ.Z: z-direction in coordinate system CRD. EQ.XMOMENT: x-direction moment for JOINT. EQ.YMOMENT: y-direction moment for JOINT. EQ.ZMOMENT: z-direction moment for JOINT. EQ.n: vector ID n in coordinate system CRD.
CRD	Coordinate system, defined by *DEFINE_COORDINATE_NODES, to which VECT is attached.

FTYPE	TYPEID (Enter ID defined in following KEYWORD commands)	OUTPUT	ASCII FILE
AIRBAG	*AIRBAG	Airbag pressure	ABSTAT
CONTACT	*CONTACT	Contact force	RCFORC
JOINT	*CONSTRAINED_JOINT	Joint force	JNTFORC
JOINTSTIF	*CONSTRAINED_JOINT_STIFFNESS	Joint stiffness force	NA
PRESC-MOT	*BOUNDARY_PRESCRIBED_MOTION	Prescribed motion force	BNDOUT
RWALL	*RIGIDWALL	Rigid wall force	RWFORC
SPC	*BOUNDARY_SPC	SPC reaction force	SPCFORC
SPOTWELD	*CONSTRAINED_POINTS	Spot weld force	SWFORC
X-SECTION	*DATABASE_CROSS_SECTION	Joint force	SECFORC

Table 30.2. Force transducer type sensor

*SENSOR_DEFINE_NODE

Purpose: Define an accelerometer type sensor. This command outputs the relative linear acceleration, velocity, or relative coordinate of node-1 with respect to node-2 along vector VID, which is fixed in coordinate-system CRD.

Card	1	2	3	4	5	6	7	8
Variable	SENSID	NODE1	NODE2	VID	CRD	CTYPE		
Type	I	I	I	A/I	I	A		

VARIABLE	DESCRIPTION
SENSID	Sensor ID.
NODE1,2	Nodes defining the accelerometer.
VID	Vector along which the forces is measured: EQ.X: x-direction in coordinate system CRD. EQ.Y: y-direction in coordinate system CRD. EQ.Z: z-direction in coordinate system CRD. EQ.n: vector ID n in coordinate system CRD.
CRD	Coordinate system, defined by *DEFINE_COORDINATE_NODES, to which VECT is attached.
CTYPE	Output component type: EQ.ACC: acceleration EQ.VEL: velocity EQ.COORD: displacement

*SENSOR_SWITCH

Purpose: This command compares the value of a sensor, *SENSOR_DEFINE or SENSOR_ CALC-MATH, to a given criterion to check if the switch condition is met. It output a logic value of TRUE or FALSE.

Card	1	2	3	4	5	6	7	8
Variable	SWITID	TYPE	SENSID	LOGIC	VALUE	FILTRID	TIMWIN	
Type	I	A	I	A	F	I	F	

VARIABLE	DESCRIPTION
SWITID	Switch ID can be referred directly by *SENSOR_CONTROL to control the status of entities like CONTACT and AIRBAG, or can be referred to by *SENSOR_SWITCH_CALC-LOGIC for logic computation.
TYPE	Type: EQ.Sensor: EQ.Time:
SENSID	ID of the sensor whose value will be compared to the criterion to determine if a switch condition is met.
LOGIC	Logic: EQ.LT: less than EQ.GT: greater than
VALUE	Critical value
FILTER	Filter option. Not yet implemented
TIMWIN	Trigger a status change when the value given by the sensor is less than or greater than (depending on LOGIC) the VALUE for a duration defined by TIMWIN.

*SENSOR_SWITCH_CALC-LOGIC

Purpose: This command performs a logic calculation for the logic output of up to seven *SENSOR_SWITCH or *SENSOR_SWITCH_CALC-LOGIC definitions. The output is a logic value of either TRUE or FALSE.

Card	1	2	3	4	5	6	7	8
Variable	SWITID	SWIT1	SWIT2	SWIT3	SWIT4	SWIT5	SWIT6	SWIT7
Type	I	I	I	I	I	I	I	I

VARIABLE	DESCRIPTION
SWITID	Switch ID can be referred directly by *SENSOR_CONTROL to control the status of entities like CONTACT and AIRBAG, or can be referred to by *SENSOR_SWITCH_CALC-LOGIC for logic computation.
SWITn	Input a positive sensor ID for "AND" and negative ID for "OR".

*SET

The keyword *SET provides a convenient way of defining groups of nodes, parts, elements, and segments. The sets can be used in the definitions of contact interfaces, loading conditions, boundary conditions, and other inputs. Each set type must have a unique numeric identification. The keyword control cards in this section are defined in alphabetical order:

***SET_BEAM_**{*OPTION*}**_**{*OPTION*}

***SET_BEAM_ADD**

***SET_DISCRETE_**{*OPTION*}**_**{*OPTION*}

***SET_DISCRETE_ADD**

***SET_MULTI-MATERIAL_GROUP_LIST**

***SET_NODE_**{*OPTION*}**_**{*OPTION*}

***SET_NODE_ADD_{*OPTION*}**

***SET_PART_{***OPTION*}**_**{*OPTION*}

***SET_PART_ADD**

***SET_SEGMENT_**{*OPTION*}**_**{*OPTION*}

***SET_2D_SEGMENT_**{*OPTION*}**_**{*OPTION*}

***SET_SHELL_**{*OPTION*}**_**{*OPTION*}

***SET_SHELL_ADD**

***SET_SOLID_**{*OPTION*}**_**{*OPTION*}

***SET_SOLID_ADD**

***SET_TSHELL_**{*OPTION*}**_**{*OPTION*}

An additional option **_TITLE** may be appended to all the ***SET** keywords. If this option is used then an addition line is read for each section in 80a format which can be used to describe the set. At present LS-DYNA does make use of the title. Inclusion of titles gives greater clarity to input decks.

The **GENERAL** option is available for set definitions. In this option, the commands are executed in the order defined. For example, the delete option cannot delete a node or element unless the node or element was previously added via a command such as BOX or ALL.

The **COLLECT** option allows for the definition of multiple sets that share the same ID and combines them into one large set whenever this option is found. If two or more like sets definitions share the same IDs, they are combined if and only if the **_COLLECT** option is specified in each definition. If the **_COLLECT** option is not specified for one or more like set definitions that share identical ID's an error termination will occur. For include files using ***INCLUDE_TRANSFORM** where set offsets are specified, the offsets are not applied for the case where the **_COLLECT** option is present.

***SET_BEAM_**{*OPTION*}

Available options include:

<BLANK>

GENERATE

GENERAL

COLLECT

The last option, GENERATE, will generate a block of beam element ID's between a starting ID and an ending ID. An arbitrary number of blocks can be specified to define the set.

Purpose: Define a set of beam elements.

Card 1	1	2	3	4	5	6	7	8
Variable	SID							
Type	I							
Default	none							

Cards 2, 3, 4, ... (*OPTION*=none) (The next "*" card terminates the input.)

Card 2…	1	2	3	4	5	6	7	8
Variable	K1	K2	K3	K4	K5	K6	K7	K8
Type	I	I	I	I	I	I	I	I

Cards 2, 3, 4, ... (*OPTION*=GENERATE) (The next "*" card terminates the input.)

Card 2…	1	2	3	4	5	6	7	8
Variable	B1BEG	B1END	B2BEG	B2END	B3BEG	B3END	B4BEG	B4END
Type	I	I	I	I	I	I	I	I

Cards 2, 3, 4, ... (*OPTION*=GENERAL) (The next "*" card terminates the input.) This set is a combination of a series of options: ALL, ELEM, DELEM, PART, DPART, BOX, and DBOX.

Card 2…	1	2	3	4	5	6	7	8
Variable	OPTION	E1	E2	E3	E4	E5	E6	E7
Type	A	I	I	I	I	I	I	I

VARIABLE	DESCRIPTION
SID	Set ID
K1	First beam element
K2	Second beam element
.	. .
.	. .
KNUM	Last beam element
BNBEG	First beam element ID in block N.
BNEND	Last beam element ID in block N. All defined ID's between and including BNBEG to BNEND are added to the set. These sets are generated after all input is read so that gaps in the element numbering are not a problem. BNBEG and BNEND may simply be limits on the ID's and not element ID's.
OPTION	Option for GENERAL. See table below.
E1,...,E7	Specified entity. Each card must have the option specified. See table below.

OPTION	ENTITY (define up to 7)	FUNCTION
ALL		All beam elements will be included in the set.
ELEM	e1, e2, e3, e4, e5, e6, e7	Elements e1, e2, e3, ... will be included.
DELEM	e1, e2, e3, e4, e5, e6, e7	Elements e1, e2, e3, ... previously added will be excluded.
PART	p1, p2, p3, p4, p5, p6, p7	Elements of parts p1, p2, p3, ... will be included.
DPART	p1, p2, p3, p4, p5, p6, p7	Elements of parts p1, p2, p3, ... previously added will be excluded.
BOX	b1, b2, b3, b4, b5, b6, b7	Elements inside boxes b1, b2, ... will be included.
DBOX	b1, b2, b3, b4, b5, b6, b7	Elements inside boxes b1, b2, ... previously added will be excluded.

***SET_BEAM_ADD**

Purpose: Define a beam set by combining beam sets.

Card 1	1	2	3	4	5	6	7	8
Variable	SID							
Type	I							
Default	none							
Remark								

Card 2, 3, 4 ... (The next "*" card terminates the input.)

Card 2…	1	2	3	4	5	6	7	8
Variable	BSID1	BSID2	BSID3	BSID4	BSID5	BSID6	BSID7	BSID8
Type	I	I	I	I	I	I	I	I

VARIABLE	DESCRIPTION
SID	Set ID of new beam set. All beam sets should have a unique set ID.
BSID*n*	The *n*th beam set ID

***SET_DISCRETE_**{*OPTION*}

Available options include:

<BLANK>

GENERATE

GENERAL

COLLECT

The last option, GENERATE, will generate a block of discrete element ID's between a starting ID and an ending ID. An arbitrary number of blocks can be specified to define the set.

Purpose: Define a set of discrete elements.

Card 1	1	2	3	4	5	6	7	8
Variable	SID							
Type	I							
Default	none							

Cards 2, 3, 4, ... (*OPTION***=none) (The next "*" card terminates the input.)**

Card 2…	1	2	3	4	5	6	7	8
Variable	K1	K2	K3	K4	K5	K6	K7	K8
Type	I	I	I	I	I	I	I	I

Cards 2, 3, 4, ... (*OPTION***=GENERATE) (The next "*" card terminates the input.)**

Card 2…	1	2	3	4	5	6	7	8
Variable	B1BEG	B1END	B2BEG	B2END	B3BEG	B3END	B4BEG	B4END
Type	I	I	I	I	I	I	I	I

Cards 2, 3, 4, ... (*OPTION*=GENERAL) (The next "*" card terminates the input.) This set is a combination of a series of options: ALL, ELEM, DELEM, PART, DPART, BOX, and DBOX.

Card 2…	1	2	3	4	5	6	7	8
Variable	OPTION	E1	E2	E3	E4	E5	E6	E7
Type	A	I	I	I	I	I	I	I

VARIABLE	DESCRIPTION
SID	Set ID
K1	First discrete element
K2	Second discrete element
.	. .
.	. .
KNUM	Last discrete element
BNBEG	First discrete element ID in block N.
BNEND	Last discrete element ID in block N. All defined ID's between and including BNBEG to BNEND are added to the set. These sets are generated after all input is read so that gaps in the element numbering are not a problem. BNBEG and BNEND may simply be limits on the ID's and not element ID's.
OPTION	Option for GENERAL. See table below.
E1,...,E7	Specified entity. Each card must have the option specified. See table below.

OPTION	ENTITY (define up to 7)	FUNCTION
ALL		All discrete elements will be included in the set.
ELEM	e1, e2, e3, e4, e5, e6, e7	Elements e1, e2, e3, ... will be included.
DELEM	e1, e2, e3, e4, e5, e6, e7	Elements e1, e2, e3, ... previously added will be excluded.
PART	p1, p2, p3, p4, p5, p6, p7	Elements of parts p1, p2, p3, ... will be included.
DPART	p1, p2, p3, p4, p5, p6, p7	Elements of parts p1, p2, p3, ... previously added will be excluded.
BOX	b1, b2, b3, b4, b5, b6, b7	Elements inside boxes b1, b2, ... will be included.
DBOX	b1, b2, b3, b4, b5, b6, b7	Elements inside boxes b1, b2, ... previously added will be excluded.

***SET_DISCRETE_ADD**

Purpose: Define a discrete set by combining discrete sets.

Card 1	1	2	3	4	5	6	7	8
Variable	SID							
Type	I							
Default	None							
Remark								

Card 2, 3, 4 ... (The next "*" card terminates the input.)

Card 2…	1	2	3	4	5	6	7	8
Variable	DSID1	DSID2	DSID3	DSID4	DSID5	DSID6	DSID7	DSID8
Type	I	I	I	I	I	I	I	I

VARIABLE	DESCRIPTION
SID	Set ID of new beam set. All beam sets should have a unique set ID.
DSID*n*	The *n*th discrete set ID

*SET_MULTI-MATERIAL_GROUP_LIST

Purpose: This command defines an ALE multi-material set ID (AMMSID) which contains a collection of one or more ALE multi-material group ID(s) (AMMGID). This provides a means for selecting any specific ALE multi-material(s). Application includes, for example, a selection of any particular fluid(s) to be coupled to a fluid-structure interaction.

Card 1

	1	2	3	4	5	6	7	8
Variable	AMSID							
Type	I							
Default	0							

Card 2

Variable	AMGID1	AMGID2	AMGID3	AMGID4	AMGID5	AMGID6	AMGID7	AMGID8
Type	I	I	I	I	I	I	I	I
Default	0	0	0	0	0	0	0	0

VARIABLE	DESCRIPTION
AMSID	An ALE multi-material set ID (AMSID) which contains a collection of one or more ALE multi-material group ID(s) (AMGID).
AMGID1	The 1st ALE multi-material group ID (AMGID=1) defined by the 1st data line of the *ALE_MULTI-MATERIAL_GROUP card.
...	...
AMGID8	The 8th ALE multi-material group ID (AMGID=1) defined by the 8th data line of the *ALE_MULTI-MATERIAL_GROUP card.

Remarks:

1. Refer to an example in the *CONSTRAINED_LAGRANGE_IN_SOLID section.

***SET_NODE_**{*OPTION*}

Available options include:

<BLANK>

LIST

COLUMN

LIST_GENERATE

GENERAL

COLLECT

The option, LIST_GENERATE, will generate a block of node ID's between a starting nodal ID number and an ending nodal ID number. An arbitrary number of blocks can be specified to define the set.

Purpose: Define a nodal set with some identical or unique attributes.

Card 1 1 2 3 4 5 6 7 8

Variable	SID	DA1	DA2	DA3	DA4	SOLVER		
Type	I	F	F	F	F	A		
Default	none	0.	0.	0.	0.	MECH		
Remark		1	1	1	1	3		

Cards 2, 3, 4, ... (*OPTION*=LIST or <BLANK>) (The next "*" card terminates the input.)

Card 2… 1 2 3 4 5 6 7 8

Variable	NID1	NID2	NID3	NID4	NID5	NID6	NID7	NID8
Type	I	I	I	I	I	I	I	I

Cards 2, 3, 4, ... (*OPTION*=COLUMN) (The next "*" card terminates the input.)

Card 2…	1	2	3	4	5	6	7	8
Variable	NID	A1	A2	A3	A4			
Type	I	F	F	F	F			
Remark		2	2	2	2			

Cards 2, 3, 4, ... (*OPTION*=LIST_GENERATE) (The next "*" card terminates the input.)

Card 2…	1	2	3	4	5	6	7	8
Variable	B1BEG	B1END	B2BEG	B2END	B3BEG	B3END	B4BEG	B4END
Type	I	I	I	I	I	I	I	I

Cards 2, 3, 4, ... (*OPTION*=GENERAL) (The next "*" card terminates the input.) This set is a combination of a series of options: ALL, NODE, DNODE, PART, DPART, BOX, and DBOX.

Card 2…	1	2	3	4	5	6	7	8
Variable	OPTION	E1	E2	E3	E4	E5	E6	E7
Type	A	I	I	I	I	I	I	I

VARIABLE	DESCRIPTION
SID	Set identification. All node sets should have a unique set ID.
DA1	First nodal attribute default value, see remark 1 below.
DA2	Second nodal attribute default value
DA3	Third nodal attribute default value
DA4	Fourth nodal attribute default value
NIDN	Node ID n

VARIABLE	DESCRIPTION
NID	Nodal ID
A1	First nodal attribute, see remark 2 below.
A2	Second nodal attribute
A3	Third nodal attribute
A4	Fourth nodal attribute
BNBEG	First node ID in block N.
BNEND	Last node ID in block N. All defined ID's between and including BNBEG to BNEND are added to the set. These sets are generated after all input is read so that gaps in the node numbering are not a problem. BNBEG and BNEND may simply be limits on the ID's and not nodal ID's.
OPTION	Option for GENERAL. See table below.
E1,...,E7	Specified entity. Each card must have the option specified. See table below.

OPTION	ENTITY (define up to 7)	FUNCTION
ALL		All nodes will be included in the set.
NODE	n1, n2, n3, n4, n5, n6, n7	Nodes n1, n2, n3, ... will be included.
DNODE	n1, n2, n3, n4, n5, n6, n7	Nodes n1, n2, n3, ... previously added will be excluded.
PART	p1, p2, p3, p4, p5, p6, p7	Nodes of parts p1, p2, p3, ... will be included.
DPART	p1, p2, p3, p4, p5, p6, p7	Nodes of parts p1, p2, p3, ... previously added will be excluded.
BOX	b1, b2, b3, b4, b5, b6, b7	Nodes inside boxes b1, b2, b3, ... will be included.
DBOX	b1, b2, b3, b4, b5, b6, b7	Nodes inside boxes b1, b2, b3, ... previously added will be excluded.

Remarks:

1. Nodal attributes can be assigned for some input types. For example, for contact option, *CONTACT_TIEBREAK_NODES_TO_SURFACE the attributes are:

   DA1=NFLF Normal failure force,

   DA2=NSFLF Shear failure force,

   DA3=NNEN Exponent for normal force,

   DA4=NMES Exponent for shear force.

2. The default nodal attributes can be overridden on these cards; otherwise, A1=DA1, etc.

3. This field is used by a non-mechanics solver to create a set defined on that solver's mesh. By default, the set refers to the mechanics mesh.

***SET_NODE_ADD_**{*OPTION*}

Available options include:

<BLANK>

ADVANCED

Purpose: Define a node set by combining node sets or for the ADVANCED option by combining, NODE, SHELL, SOLID, BEAM, SEGMENT, DISCRETE and THICK SHELL sets.

Card 1	1	2	3	4	5	6	7	8
Variable	NSID	A1	A2	A3	A4	SOLVER		
Type	I	F	F	F	F	A		
Default	none	none	none	none	none	MECH		
Remark						1		

If the ADVANCED option is inactive:

Card 2, 3, 4, ... (The next "*" card terminates the input.)

Card 2…	1	2	3	4	5	6	7	8
Variable	NSID1	NSID2	NSID3	NSID4	NSID5	NSID6	NSID7	NSID8
Type	I	I	I	I	I	I	I	I

If the ADVANCED option is active:

Card 2, 3, 4 ... (The next "*" card terminates the input.)

Card 2…	1	2	3	4	5	6	7	8
Variable	SID1	TYPE1	SID2	TYPE2	SID3	TYPE3	SID4	TYPE4
Type	I	I	I	I	I	I	I	I

VARIABLE	DESCRIPTION
NSID	Set ID of new node set. All node sets should have a unique set ID.
BSID*n*	The *n*th node set ID
SID*n*	The *n*th set ID
TYPE*n*	Type set for SID*n*: 1: - Node set 2: - Shell set 3: - Beam set 4: - Solid set 5: - Segment set 6: - Discrete set 7: - Thick shell set

Remarks:

1. This field is used by a non-mechanics solver to create a set defined on that solver's mesh. By default, the set refers to the mechanics mesh.

***SET_PART_**{*OPTION*}

Available options include:

<BLANK>

LIST

COLUMN

LIST_GENERATE

COLLECT

The last option will generate a block of part ID's between a starting part ID number and an ending part ID number. An arbitrary number of blocks can be specified to define the part set.

Purpose: Define a set of parts with optional attributes. For the column option, see *AIRBAG or *CONSTRAINED _RIGID_BODY_STOPPERS.

Card 1	1	2	3	4	5	6	7	8
Variable	SID	DA1	DA2	DA3	DA4	SOLVER		
Type	I	F	F	F	F	A		
Default	none	0.				MECH		
Remark		1	1	1	1	3		

Card 2, 3, 4, ... (*OPTION*=LIST or <BLANK>) (The next "*" card terminates the input.)

Card 2…	1	2	3	4	5	6	7	8
Variable	PID1	PID2	PID3	PID4	PID5	PID6	PID7	PID8
Type	I	I	I	I	I	I	I	I

Card 2, 3, 4, ... (*OPTION*=COLUMN) (The next "*" card terminates the input.)

Card 2…	1	2	3	4	5	6	7	8
Variable	PID	A1	A2	A3	A4			
Type	I	F	F	F	F			
Remark		2	2	2	2			

Cards 2, 3, 4, ... (*OPTION*=LIST_GENERATE) (The next "*" card terminates the input.)

Card 2…	1	2	3	4	5	6	7	8
Variable	B1BEG	B1END	B2BEG	B2END	B3BEG	B3END	B4BEG	B4END
Type	I	I	I	I	I	I	I	I

VARIABLE	DESCRIPTION
SID	Set ID. All part sets should have a unique set ID.
DA1	First attribute default value, see remark 1 below.
DA2	Second attribute default value
DA3	Third attribute default value
DA4	Fourth attribute default value
PID	Part ID
PID1	First part ID
PID2	Second part ID
.	. .
A1	First part attribute, see remark 2 below.
A2	Second part attribute
A3	Third part attribute
A4	Fourth part attribute

VARIABLE	DESCRIPTION
BNBEG	First part ID in block N.
BNEND	Last part ID in block N. All defined ID's between and including BNBEG to BNEND are added to the set. These sets are generated after all input is read so that gaps in the part numbering are not a problem. BNBEG and BNEND may simply be limits on the ID's and not part ID's.

Remarks:

1. Part attributes can be assigned for some input types. For example, for airbags a time delay, DA1=T1, can be defined before pressure begins to act along with a time delay, DA2=T2, before full pressure is applied, (default T2=T1), and for the constraint option, *CONSTRAINED_RIGID_BODY_STOPPERS one attribute can be defined: DA1, the closure distance which activates the stopper constraint.

2. The default part attributes can be overridden on the part cards; otherwise, A1=DA1, etc.

3. This field is used by a non-mechanics solver to create a set defined on that solver's mesh. By default, the set refers to the mechanics mesh.

*SET_PART_ADD

Purpose: Define a part set by combining part sets. The attributes, if any, (see *SET_PART above) will be taken from the part sets that are combined.

Card 1	1	2	3	4	5	6	7	8
Variable	SID	SOLVER						
Type	I	A						
Default	none	MECH						
Remark		1						

Card 2, 3, 4, ... (The next "*" card terminates the input.)

Card 2…	1	2	3	4	5	6	7	8
Variable	PSID1	PSID2	PSID3	PSID4	PSID5	PSID6	PSID7	PSID8
Type	I	I	I	I	I	I	I	I

VARIABLE	DESCRIPTION
SID	Set ID. All part sets should have a unique set ID.
PSID*n*	The *n*th part set ID

Remarks:

1. This field is used by a non-mechanics solver to create a set defined on that solver's mesh. By default, the set refers to the mechanics mesh.

***SET_SEGMENT_**{*OPTION*}

Available options include:

<BLANK>

GENERAL

COLLECT

Purpose: Define a set of quadrilateral and triangular segments with optional identical or unique attributes.

Card 1 1 2 3 4 5 6 7 8

Variable	SID	DA1	DA2	DA3	DA4	SOLVER		
Type	I	F	F	F	F	A		
Default	none	0.	0.	0.	0.	MECH		
Remarks		1	1	1	1	4		

Cards 2, 3, 4, ... (No option is specified) (The next "*" card terminates the input.)

Card 2… 1 2 3 4 5 6 7 8

Variable	N1	N2	N3	N4	A1	A2	A3	A4
Type	I	I	I	I	F	F	F	F
Remarks				2	3	3	3	3

Cards 2, 3, 4, ... (*OPTION*=GENERAL) (The next "*" card terminates the input.) This set is a combination of a series of options listed in the table defined below.

Card 2…	1	2	3	4	5	6	7	8
Variable	OPTION	E1	E2	E3	E4	E5	E6	E7
Type	A	I	I	I	I or F	I or F	I or F	I or F

VARIABLE	DESCRIPTION
SID	Set ID. All segment sets should have a unique set ID.
DA1	First segment attribute default value, see remark 1 below.
DA2	Second segment attribute default value
DA3	Third segment attribute default value
DA4	Fourth segment attribute default value
N1	Nodal point n_1
N2	Nodal point n_2
N3	Nodal point n_3
N4	Nodal point n_4, see remark 2 below.
A1	First segment attribute, see remark 3 below.
A2	Second segment attribute
A3	Third segment attribute
A4	Fourth segment attribute
NFLS	Normal failure stress
SFLS	Shear failure stress. Failure criterion:
OPTION	Option for GENERAL. See table below.
E1,...,E7	Specified entity. Each card must have an option specified. See table below.

FORMAT (A10,3I10, 4F10.0)		
OPTION	ENTITIES + ATTRIBUTES	FUNCTION
BOX	b1, b2, b3, a1, a2, a3, a4	Generate segments inside box ID bi, i=1,,2,3. For shell elements one segment per shell is generated. For solid elements only those segments wrapping the solid part and pointing outward from the part will be generated.
BOX_SHELL	b1, b2, b3, a1, a2, a3, a4	Generate segments inside box ID bi, i=1,,2,3. The segments are only generated for shell elements. One segment per shell is generated.
BOX_SLDIO	b1, b2, b3, a1, a2, a3, a4	Generate segments inside box ID bi, i=1,,2,3. Both exterior segments and inter-element segments are generated.
BOX_SOLID	b1, b2, b3, a1, a2, a3, a4	Generate segments inside box ID bi, i=1,,2,3. The segments are only generated for exterior solid elements
PART	p1, p2, p3, a1, a2, a3, a4	Generate segments of parts p1, p2, p3 with attributes a1-a4. For shell elements one segment per shell is generated. For solid elements only those segments wrapping the solid part and pointing outward from the part will be generated.
PART_IO	p1, p2, p3, a1, a2, a3, a4	Generate segments of parts p1, p2, p3 with attributes a1-a4. Same as the PART option above except that inter-element segments inside parts will be generated as well. This option is sometimes useful for single surface contact of solid elements to prevent negative volumes caused be inversion.

FORMAT (A10,7I10)		
DBOX	b1, b2, b3, b4, b5, b6, b7	Segments inside boxes b1, b2, ... previously added will be excluded.
DBOX_SHELL	b1, b2, b3, b4, b5, b6, b7	Shell related segments inside boxes b1, b2, ... previously added will be excluded.
DBOX_SOLID	b1, b2, b3, b4, b5, b6, b7	Solid related segments inside boxes b1, b2, ... previously added will be excluded.
DPART	p1, p2, p3, p4, p5, p6, p7	Segments of parts p1, p2, p3, ... previously added will be excluded.
DSEG	n1, n2, n3, n4	Segments with node ID's n1,n2, n3, and n4 previously added will be deleted. The numbering sequence is irrelevant.
SEG	n1, n2, n3, n4	Create segment with node ID's n1,n2, n3, and n4.t.

Remarks:

1. Segment attributes can be assigned for some input types. For example, for the contact options, the attributes for the SLAVE surface are:

   DA1=NFLS Normal failure stress, *CONTACT_TIEBREAK_SURFACE_contact only,

   DA2=SFLS Shear failure stress, *CONTACT_TIEBREAK_SURFACE_contact only,

   DA3=FSF Coulomb friction scale factor,

   DA4=VSF Viscous friction scale factor,

   and the attributes for the MASTER surface are:

   DA1=FSF Coulomb friction scale factor,

   DA2=VSF Viscous friction scale factor.

   For airbags, see *AIRBAG, a time delay, DA1=T1, can be defined before pressure begins to act on a segment along with a time delay, DA2=T2, before full pressure is applied to the segment, (default T2=T1), and for the constraint option,

2. To define a triangular segment make n_4 equal to n_3.

3. The default segment attributes can be overridden on these cards, otherwise, A1=DA1, etc.

4. This field is used by a non-mechanics solver to create a set defined on that solver's mesh. By default, the set refers to the mechanics mesh.

***SET_2D_SEGMENT _**{*OPTION*}

Available options include:

<BLANK>

SET

COLLECT

Purpose: Define a set of boundary line segments in two dimensional axisymmetric, plane stress, and plane strain geometries with optional identical or unique attributes. This option is recommended for thermal problems which involve adaptivity.

	1	2	3	4	5	6	7	8
Variable	SID	DA1	DA2	DA3	DA4			
Type	I	F	F	F	F			
Default	none	0.	0.	0.	0.			
Remarks		1	1	1	1			

	1	2	3	4	5	6	7	8
Variable	PID/PSID							
Type	I							
Remarks	2							

VARIABLE	DESCRIPTION
SID	Set ID. All segment sets should have a unique set ID.
DA1	First segment attribute default value, see remark 1 below.
DA2	Second segment attribute default value

VARIABLE	DESCRIPTION
DA3	Third segment attribute default value
DA4	Fourth segment attribute default value
PID/PSID	Part ID or part set ID if SET option is specified.

Remarks:

1. The boundary along $r = 0$ isn't included in axisymmetric problems.

2. The common boundary between parts isn't included in the boundary segments.

***SET_SHELL_**{*OPTION*}

Available options include:

<BLANK>

LIST

COLUMN

LIST_GENERATE

GENERAL

COLLECT

The last option will generate a block of shell ID's between a starting shell ID number and an ending ID number. An arbitrary number of blocks can be specified to define the shell set.

Purpose: Define a set of shell elements with optional identical or unique attributes.

Card 1	1	2	3	4	5	6	7	8
Variable	SID	DA1	DA2	DA3	DA4			
Type	I	F	F	F	F			
Default	none	0.	0.	0.	0.			
Remarks		1	1	1	1			

Card 2, 3, 4, ... (*OPTION*=LIST or <BLANK>) (The next "*" card terminates the input.)

Card 2…	1	2	3	4	5	6	7	8
Variable	EID1	EID2	EID3	EID4	EID5	EID6	EID7	EID8
Type	I	I	I	I	I	I	I	I
Remarks	2	2	2	2	2	2	2	2

Card 2, 3, 4, ... (*OPTION*=COLUMN) (The next "*" card terminates the input.)

Card 2…	1	2	3	4	5	6	7	8
Variable	EID	A1	A2	A3	A4			
Type	I	F	F	F	F			
Remarks		3	3	3	3			

Cards 2, 3, 4, ... (*OPTION*=LIST_GENERATE) (The next "*" card terminates the input.)

Card 2…	1	2	3	4	5	6	7	8
Variable	B1BEG	B1END	B2BEG	B2END	B3BEG	B3END	B4BEG	B4END
Type	I	I	I	I	I	I	I	I

Cards 2, 3, 4, ... (*OPTION*=GENERAL) (The next "*" card terminates the input.) This set is a combination of a series of options: ALL, ELEM, DELEM, PART, DPART, BOX, and DBOX.

Card 2…	1	2	3	4	5	6	7	8
Variable	OPTION	E1	E2	E3	E4	E5	E6	E7
Type	A	I	I	I	I	I	I	I

VARIABLE	DESCRIPTION
SID	Set ID. All shell sets should have a unique set ID.
DA1	First attribute default value, see remark 1.
DA2	Second attribute default value
DA3	Third attribute default value
DA4	Fourth attribute default value
EID1	First shell element ID, see remark 2.

VARIABLE	DESCRIPTION
EID2	Second shell element ID
.	. .
.	. .
EID	Element ID
A1	First attribute
A2	Second attribute
A3	Third attribute
A4	Fourth attribute
BNBEG	First shell ID in shell block N.
BNEND	Last shell ID in block N. All defined ID's between and including BNBEG to BNEND are added to the set. These sets are generated after all input is read so that gaps in the element numbering are not a problem. BNBEG and BNEND may simply be limits on the ID's and not element ID's.
OPTION	Option for GENERAL. See table below.
E1,...,E7	Specified entity. Each card must have the option specified. See table below.

OPTION	ENTITY (define up to 7)	FUNCTION
ALL		All shell elements will be included in the set.
ELEM	e1, e2, e3, e4, e5, e6, e7	Elements e1, e2, e3, ... will be included.
DELEM	e1, e2, e3, e4, e5, e6, e7	Elements e1, e2, e3, ... previously added will be excluded.
PART	p1, p2, p3, p4, p5, p6, p7	Elements of parts p1, p2, p3, ... will be included.
DPART	p1, p2, p3, p4, p5, p6, p7	Elements of parts p1, p2, p3, ... previously added will be excluded.
BOX	b1, b2, b3, b4, b5, b6, b7	Elements inside boxes b1, b2, ... will be included.
DBOX	b1, b2, b3, b4, b5, b6, b7	Elements inside boxes b1, b2, ... previously added will be excluded.

Remarks:

1. Shell attributes can be assigned for some input types. For example, for the contact options, the attributes for the SLAVE surface are:

   DA1=NFLS Normal failure stress, *CONTACT_TIEBREAK_SURFACE_contact only,

   DA2=SFLS Shear failure stress, *CONTACT_TIEBREAK_SURFACE_contact only,

   DA3=FSF Coulomb friction scale factor,

   DA4=VSF Viscous friction scale factor,

   and the attributes for the MASTER surface are:

   DA1=FSF Coulomb friction scale factor,

   DA2=VSF Viscous friction scale factor.

2. The default attributes are taken.

3. The default shell attributes can be overridden on these cards; otherwise, A1=DA1, etc.

*SET_SHELL_ADD

Purpose: Define a shell set by combining shell sets.

Card 1	1	2	3	4	5	6	7	8
Variable	SID							
Type	I							
Default	none							
Remark								

Card 2, 3, 4 ... (The next "*" card terminates the input.)

Card 2…	1	2	3	4	5	6	7	8
Variable	SSID1	SSID2	SSID3	SSID4	SSID5	SSID6	SSID7	SSID8
Type	I	I	I	I	I	I	I	I

VARIABLE	DESCRIPTION
SID	Set ID of new shell set. All shell sets should have a unique set ID.
BSID*n*	The *n*th shell set ID

***SET_SOLID_**{*OPTION*}

Available options include:

<BLANK>

GENERATE

GENERAL

COLLECT

The last option, GENERATE, will generate a block of solid element ID's between a starting ID and an ending ID. An arbitrary number of blocks can be specified to define the set.

Purpose: Define a set of solid elements.

Card 1	1	2	3	4	5	6	7	8
Variable	SID	SOLVER						
Type	I	A						
Default	none	MECH						
Remark		1						

Cards 2, 3, 4, ... (*OPTION*=none) (The next "*" card terminates the input.)

Card 2…	1	2	3	4	5	6	7	8
Variable	K1	K2	K3	K4	K5	K6	K7	K8
Type	I	I	I	I	I	I	I	I

Cards 2, 3, 4, ... (*OPTION*=GENERATE) (The next "*" card terminates the input.)

Card 2…	1	2	3	4	5	6	7	8
Variable	B1BEG	B1END	B2BEG	B2END	B3BEG	B3END	B4BEG	B4END

Type	I	I	I	I	I	I	I	I

Cards 2, 3, 4, ... (*OPTION*=GENERAL) (The next "*" card terminates the input.) This set is a combination of a series of options: ALL, ELEM, DELEM, PART, DPART, BOX, and DBOX.

Card 2…	1	2	3	4	5	6	7	8
Variable	OPTION	E1	E2	E3	E4	E5	E6	E7
Type	A	I	I	I	I	I	I	I

VARIABLE	DESCRIPTION
SID	Set ID. All solid sets should have a unique set ID.
K1	First element ID
K2	Second element ID
.	. .
.	. .
K8	Eighth element ID
BNBEG	First solid element ID in block N.
BNEND	Last solid element ID in block N. All defined ID's between and including BNBEG to BNEND are added to the set. These sets are generated after all input is read so that gaps in the element numbering are not a problem. BNBEG and BNEND may simply be limits on the ID's and not element ID's.
OPTION	Option for GENERAL. See table below.
E1,...,E7	Specified entity. Each card must have the option specified. See table below.

OPTION	ENTITY (define up to 7)	FUNCTION
ALL		All solid elements will be included in the set.
ELEM	e1, e2, e3, e4, e5, e6, e7	Elements e1, e2, e3, ... will be included.
DELEM	e1, e2, e3, e4, e5, e6, e7	Elements e1, e2, e3, ... previously added will be excluded.
PART	p1, p2, p3, p4, p5, p6, p7	Elements of parts p1, p2, p3, ... will be included.
DPART	p1, p2, p3, p4, p5, p6, p7	Elements of parts p1, p2, p3, ...previously added will be excluded.
BOX	b1, b2, b3, b4, b5, b6, b7	Elements inside boxes b1, b2, ... will be included.
DBOX	b1, b2, b3, b4, b5, b6, b7	Elements inside boxes b1, b2, ... previously added will be excluded.

Remarks:

1. This field is used by a non-mechanics solver to create a set defined on that solver's mesh. By default, the set refers to the mechanics mesh.

*SET_SOLID_ADD

Purpose: Define a solid set by combining solid sets.

Card 1	1	2	3	4	5	6	7	8
Variable	SID	SOLVER						
Type	I	A						
Default	none	MECH						
Remark		1						

Card 2, 3, 4 ... (The next "*" card terminates the input.)

Card 2…	1	2	3	4	5	6	7	8
Variable	SSID1	SSID2	SSID3	SSID4	SSID5	SSID6	SSID7	SSID8
Type	I	I	I	I	I	I	I	I

VARIABLE	DESCRIPTION
SID	Set ID of new solid set. All solid sets should have a unique set ID.
SSID*n*	The *n*th solid set ID.

Remarks:

1. This field is used by a non-mechanics solver to create a set defined on that solver's mesh. By default, the set refers to the mechanics mesh.

***SET_TSHELL_**{*OPTION*}

Available options include:

<BLANK>

GENERATE

GENERAL

COLLECT

The last option, GENERATE, will generate a block of thick shell element ID's between a starting ID and an ending ID. An arbitrary number of blocks can be specified to define the set.

Purpose: Define a set of thick shell elements.

Card 1	1	2	3	4	5	6	7	8
Variable	SID							
Type	I							
Default	none							

Cards 2, 3, 4, ... (*OPTION*=none) (The next "*" card terminates the input.)

Card 2…	1	2	3	4	5	6	7	8
Variable	K1	K2	K3	K4	K5	K6	K7	K8
Type	I	I	I	I	I	I	I	I

Cards 2, 3, 4, ... (*OPTION*=GENERATE) (The next "*" card terminates the input.)

Card 2…	1	2	3	4	5	6	7	8
Variable	B1BEG	B1END	B2BEG	B2END	B3BEG	B3END	B4BEG	B4END
Type	I	I	I	I	I	I	I	I

Cards 2, 3, 4, ... (*OPTION*=GENERAL) (The next "*" card terminates the input.) This set is a combination of a series of options: ALL, ELEM, DELEM, PART, DPART, BOX, and DBOX.

Card 2…	1	2	3	4	5	6	7	8
Variable	OPTION	E1	E2	E3	E4	E5	E6	E7
Type	A	I	I	I	I	I	I	I

VARIABLE	DESCRIPTION
SID	Set ID. All tshell sets should have a unique set ID.
K1	First thick shell element ID
K2	Second thick shell element ID
.	. .
.	. .
K8	Eighth thick shell element ID
.	. .
.	. .
BNBEG	First thick shell element ID in block N.
BNEND	Last thick shell element ID in block N. All defined ID's between and including BNBEG to BNEND are added to the set. These sets are generated after all input is read so that gaps in the element numbering are not a problem. BNBEG and BNEND may simply be limits on the ID's and not element ID's.
OPTION	Option for GENERAL. See table below.
E1,...,E7	Specified entity. Each card must have the option specified. See table below.

OPTION	ENTITY (define up to 7)	FUNCTION
ALL		All thick shell elements will be included in the set.
ELEM	e1, e2, e3, e4, e5, e6, e7	Elements e1, e2, e3, ... will be included.
DELEM	e1, e2, e3, e4, e5, e6, e7	Elements e1, e2, e3, ... previously added will be excluded.
PART	p1, p2, p3, p4, p5, p6, p7	Elements of parts p1, p2, p3, ... will be included.
DPART	p1, p2, p3, p4, p5, p6, p7	Elements of parts p1, p2, p3, ... previously added will be excluded.
BOX	b1, b2, b3, b4, b5, b6, b7	Elements inside boxes b1, b2, ... will be included.
DBOX	b1, b2, b3, b4, b5, b6, b7	Elements inside boxes b1, b2, ... previously added will be excluded.

*TERMINATION

The keyword *TERMINATION provides an alternative way of stopping the calculation before the termination time is reached. The termination time is specified on the *CONTROL_ TERMINATION input and will terminate the calculation whether or not the options available in this section are active. Different types of termination may be defined:

***TERMINATION_BODY**

***TERMINATION_CONTACT**

***TERMINATION_CURVE**

***TERMINATION_DELETED_SHELLS**_*OPTION*

***TERMINATION_DELETED_SOLIDS**_*OPTION*

***TERMINATION_NODE**

***TERMINATION_SENSOR**

***TERMINATION_BODY**

Purpose: Terminate calculation based on rigid body displacements. For *TERMINATION_ BODY the analysis terminates when the center of mass displacement of the rigid body specified reaches either the maximum or minimum value (stops 1, 2 or 3) or the displacement magnitude of the center of mass is exceeded (stop 4). If more than one condition is input, the analysis stops when any of the conditions is satisfied. Termination by other means than *TERMINATION input is controlled by the *CONTROL_ TERMINATION control card. Note that this type of termination is not active during dynamic relaxation.

Card 1	1	2	3	4	5	6	7	8
Variable	PID	STOP	MAXC	MINC				
Type	I	I	F	F				
Default	none	none	-	-				

VARIABLE	DESCRIPTION
PID	Part ID of rigid body, see *PART_*OPTION*.
STOP	Stop criterion: EQ.1: global x direction, EQ.2: global y direction, EQ.3: global z direction, EQ.4: stop if displacement magnitude is exceeded.
MAXC	Maximum (most positive) displacement, options 1, 2, 3 and 4: EQ.0.0: MAXC set to 1.0e21.
MINC	Minimum (most negative) displacement, options 1, 2 and 3 above only: EQ.0.0: MINC set to -1.0e21.

*TERMINATION_CONTACT

Purpose: The analysis terminates when the magnitude of the contact interface resultant force is zero. If more than one contact condition is input, the analysis stops when any of the conditions is satisfied. Termination by other means than *TERMINATION input is controlled by the *CONTROL_TERMINATION control card. Note that this type of termination is not active during dynamic relaxation.

Card 1	1	2	3	4	5	6	7	8
Variable	CID	ACTIM	DUR	THRES	DOF			
Type	I	F	F	F	I			
Default	none	none	-	0.0	0			

VARIABLE	DESCRIPTION
CID	Contact ID. The contact ID is defined by the ordering of the contact input unless the TITLE option which allows the CID to be defined is used in the *CONTACT section.
ACTIM	Activation time.
DUR	Time duration of null resultant force prior to termination. This time is tracked only after the activation time is reached and the contact resultant forces are zero. EQ.0.0: Immediate termination after null force is detected.
THRES	Any measured force magnitude below or equal to this specified threshold is taken as a null force. Default=0.0
DOF	Option to consider only the force magnitude in the x, y, or z global directions corresponding to DOF=1,2, and 3, respectively.

***TERMINATION_CURVE**

Purpose: Terminate the calculation when the load curve value returns to zero. This termination can be used with the contact option *CONTACT_AUTO_MOVE. In this latter option, the load curve is modified to account for the movement of the master surface.

Card 1	1	2	3	4	5	6	7	8
Variable	LCID	ATIME						
Type	I	F						
Default	none	Remark 1		-				

VARIABLE	DESCRIPTION
LCID	Load curve ID governing termination.
ATIME	Activation time. After this time the load curve is checked. If zero, see remark 1 below.

Remarks:

1. If ATIME=0.0, termination will occur after the load curve value becomes nonzero and then returns to zero.

***TERMINATION_DELETED_SHELLS**_{*OPTION*}

Available options include:

<BLANK>

SET

Purpose: Terminate the calculation when the number of deleted shells for a specified part ID exceeds the value defined here. This input has no effect for a part ID that is left undefined. Generally, this option should be used with the NFAIL1 and NFAIL4 parameters that are defined in the *CONTROL_SHELL control information.

Card 1	1	2	3	4	5	6	7	8
Variable	PID/PSID	NDS						
Type	I	I						
Default	none	1						

VARIABLE	DESCRIPTION
PID/PSID	Part ID or if option SET is active, part set ID.
NDS	Number of elements that must be deleted for the specified part ID's, before an error termination occurs.

***TERMINATION_DELETED_SOLIDS_**{*OPTION*}

Available options include:

<BLANK>

SET

Purpose: Terminate the calculation when the number of deleted solids for a specified part ID exceeds the value defined here. This input has no effect for a part ID that is left undefined.

Card 1	1	2	3	4	5	6	7	8
Variable	PID/PSID	NDS						
Type	I	I						
Default	none	1						

VARIABLE	DESCRIPTION
PID/PSID	Part ID or if option SET is active, part set ID.
NDS	Number of elements that must be deleted for the specified part ID's, before an error termination occurs.

*TERMINATION_NODE

Purpose: Terminate calculation based on nodal point coordinates. The analysis terminates for *TERMINATION_NODE when the current position of the node specified reaches either the maximum or minimum value (stops 1, 2 or 3), or picks up force from any contact surface (stops 4). Termination by other means than *TERMINATION is controlled by the *CONTROL_ TERMINATION control card. Note that this type of termination is not active during dynamic relaxation.

Card 1	1	2	3	4	5	6	7	8
Variable	NID	STOP	MAXC	MINC				
Type	I	I	F	F				
Default	none	none	-	-				

VARIABLE	DESCRIPTION
NID	Node ID, see *NODE_*OPTION*.
STOP	Stop criterion: EQ.1: global x direction, EQ.2: global y direction, EQ.3: global z direction, EQ.4: stop if node touches contact surface.
MAXC	Maximum (most positive) coordinate (options 1, 2 and 3) above only.
MINC	Minimum (most negative) coordinate (options 1, 2 and 3) above only.

***TERMINATION_SENSOR**

Purpose: Terminates the calculation when the switch condition defined in *SENSOR_SWITCH is met.

Card 1	1	2	3	4	5	6	7	8
Variable	SWID							
Type	I							
Default	none							

VARIABLE	DESCRIPTION
SWID	ID of *SENSOR_SWITCH which will terminate the calculation when its condition is met. Only one *TERMINATION_SENSOR is allowed. If more than one *TERMINATION _SENSOR is defined; only the last one is effective.

Remarks:

An example allowing more than one sensor_switch to terminate calculation:

```
*SENSOR_DEFINE_ELMENT
$ Axial force of beam element 1
44,BEAM,1,AXIAL,FORCE
*SENSOR_DEFINE_ELMENT
$ Axial force of beam element 2
55,BEAM,21,AXIAL,FORCE
*SENSOR_SWITCH
$a switch condition is  met when the axial force of beam-1 >5.0
11,SENSOR,44,GT,5.
*SENSOR_SWITCH
$a switch condition is  met when the axial force of beam-2 >10.0
22,SENSOR,55,GT,10.
*SENSOR_SWITCH
$ a switch condition is  met when time >50.
33,TIME, , 50
```

```
*SENSOR_SWITCH_CALC-LOGIC
$ a switch condition is met if both conditions of switch-11 and switch-33 are met, I.e.,
$ axial force of beam-1>5.0 and time>50
44,11,33
*SENSOR_SWITCH_CALC-LOGIC
$ a switch condition is met if both conditions of switch-22 and switch-33 are met, I.e.,
$ axial force of beam-2>10.0 and time>50
55,33,22
*SENSOR_SWITCH_CALC-LOGIC
$ a switch condition is met if the conditions of switch-44 or switch-55 is met, I.e.,
$ axial force of beam-1>5.0 and time>50 or
$ axial force of beam-2>10.0 and time>50
66,44,-55
*TERMINATION_SENSOR
$ job will be terminated when the switch condition of switch-66 is met, I.e.,
$ axial force of beam-1>5.0 and time>50 or
$ axial force of beam-2>10.0 and time>50
66
```

*TITLE

***TITLE**

Purpose: Define job title.

Card	1	2	3	4	5	6	7	8
Variable	TITLE							
Type	C							
Default	LS-DYNA USER INPUT							

VARIABLE	DESCRIPTION
TITLE	Heading to appear on output and in output files.

*USER

USER_INTERFACE_OPTION*

Available options include:

CONTROL

FRICTION

CONDUCTIVITY

Purpose: Define user defined input and allocate storage for user defined subroutines for the contact algorithms. See also *CONTROL_CONTACT. The **CONTROL** option above allows the user to take information from the contact interface for further action, e.g., stopping the analysis. A sample user subroutine is provided in Appendix F.

The **FRICTION** option may be used to modify the Coulomb friction coefficients in contact types 3, 5, or 10 (*CONTACT_SURFACE_TO_SURFACE, *CONTACT_NODES_TO_SURFACE, or *CONTACT_ONE_WAY_SURFACE_TO_SURFACE) according to contact information or to use a friction coefficient database. A sample user-defined friction subroutine is provided in Appendix G. For the subroutine to be called, the static friction coefficient FS on Card 2 of *CONTACT must be any nonzero value, and shell thickness offsets must be invoked in the contact by setting SHLTHK to 1 or 2 using *CONTROL_CONTACT or Opt. Card B in *CONTACT. The array length USRFRC in *CONTROL_CONTACT should be set to a value no less than the sum of the number of history variables NOC and the number of user-defined input parameters in *USER_INTERFACE_FRICTION.

The **CONDUCTIVITY** option is used to define heat transfer contact conductance properties for thermal contacts.

Card 1	1	2	3	4	5	6	7	8
Variable	IFID	NOC	NOCI	NHSV				
Type	I	I	I	I				
Default	none	none	None	O				

(Use as many cards as necessary to define NOCI variables)

Card 2…

1	2	3	4	5	6	7	8
Variable	UC1	UC2	UC3	UC4	UC5	UC6	UC7	UC8
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
IFID	Interface number
NOC	Number of history variables for interface. The number should not exceed the length of the array defined on *CONTROL_CONTACT. See Remarks.
NOCI	Initialize the first NOCI history variables in the input. NOCI must be smaller or equal to NOC.
NHSV	Number of history variables per interface node (only for friction and conductivity interface).
UC1	First user defined input parameter.
UC2	Second user defined input parameter.
.	. .
.	. .
.	. .
UCn	Last user defined input parameter, where n = NOCI.

Remarks:

The (NOC) interface variables (of which NOCI are initialized) are passed as arguments to the user defined subroutine. See Appendix G for the full list of arguments passed to the subroutine.

*USER_LOADING

Purpose: Provide a means of applying pressure and force boundary conditions. The keyword *USER_LOADING activates this option. Input here is optional with the input being read until the next "*" keyword appears. The data read here is to be stored in a common block provided in the user subroutine, LOADUD. This data is stored and retrieved from the restart files.

(Insert as many cards as needed. The next * card terminates input.)

Card 1…

1	2	3	4	5	6	7	8
Variable	PARM1	PARM2	PARM3	PARM4	PARM5	PARM6	PARM7	PARM8
Type	F	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	none	none

VARIABLE	DESCRIPTION
PARMn	This is the nth user input parameter.

RESTART INPUT DATA

In general three categories of restart actions are possible with LS-DYNA and are outlined in the following discussion:

a) A simple restart occurs when LS-DYNA was interactively stopped before reaching the termination time. Then simply defining the R=rtf file on the execution line for LS-DYNA restarts the calculation from the termination point and the calculation will continue to the specified termination time-see INTRODUCTION, Execution Syntax. No additional input deck is required.

b) If minor modifications are desired as, e.g.,

- reset termination time,
- reset output printing interval,
- reset output plotting interval,
- delete contact surfaces,
- delete elements and parts,
- switch deformable bodies to rigid,
- switch rigid bodies to deformable,
- change damping options.

This type of restart is called a small restart and the corresponding input deck a "small restart input deck." All modifications to the problem made with the restart input deck will be reflected in subsequent restart dumps. All the members of the file families are consecutively numbered beginning from the last member. The small input deck replaces the standard input deck on the execution line which has at least the following contents:

LS-DYNA I=*restartinput* R=*D3DUMPnn*

where *D3DUMPnn* (or whatever name is chosen for the family member) is the *n*th restart file from the last run where the data is taken. LS-DYNA automatically detects that a small input deck is used since the I=*restartinput* file may contain the keywords:

***CHANGE**_*OPTION*

***CONTROL_DYNAMIC_RELAXATION**

***CONTROL_SHELL**

***CONTROL_TERMINATION**

***CONTROL_TIMESTEP**

***DAMPING_GLOBAL**

***DATABASE**_*OPTION*

***DATABASE_BINARY**_*OPTION*

***DELETE**_*OPTION*

***INTERFACE_SPRINGBACK**

***RIGID_DEFORMABLE**_*OPTION*

***STRESS_INITIALIZATION**_{*OPTION*}

***TERMINATION**_*OPTION*

***TITLE**

***KEYWORD** (see INTRODUCTION, Execution Syntax)

***CONTROL_CPU**

***DEFINE**_*OPTION*

***SET**_*OPTION*

i.e., the keyword *STRESS_INITIALIZATION may not be used in the small restart. The user has to take care that nonphysical modifications to the input deck are avoided; otherwise, complete nonsense may be the result.

c) If many modifications are desired a so-called full restart may be the appropriate choice. Then the keyword *STRESS_INITIALIZATION has to be provided in the input. As also outlined in the INTRODUCTION, Restart Analysis, either all parts can be initialized with the restart data or some selection of parts can be made for the stress initialization. See *STRESS_INITIALIZATION. In a full deck restart, deleted elements in this section will be deleted in the full deck automatically even though they are defined. Likewise, if it is necessary to change the velocity field, that must also be performed in this section using the CHANGE_VELOCITY_.... options. The velocity field in the full deck part of the input is ignored.

***CHANGE**_*OPTION*

Purpose: Change solution options.

Available options include:

BOUNDARY_CONDITION

CONTACT_SMALL_PENETRATION

CURVE_DEFINITION

RIGID_BODY_CONSTRAINT

RIGID_BODY_INERTIA

RIGID_BODY_STOPPER

STATUS_REPORT_FREQUENCY

THERMAL_PARAMETERS

VELOCITY

VELOCITY_NODE

VELOCITY_RIGID_BODY

VELOCITY_ZERO

For **BOUNDARY_CONDITION** option define an arbitrary number of cards giving the nodal ID and the additional translational displacement boundary condition code. Previous boundary condition codes will continue to be imposed, i.e., a fixed node cannot be freed with this option. This input terminates when the next "*" card is encountered.

Card 1…	1	2	3	4	5	6	7	8
Variable	NID	BCC						
Type	I	I						

VARIABLE	DESCRIPTION
NID	Nodal point ID, see also *NODE.
BCC	New translational boundary condition code: EQ.1: constrained x displacement, EQ.2: constrained y displacement, EQ.3: constrained z displacement, EQ.4: constrained x and y displacements, EQ.5: constrained y and z displacements, EQ.6: constrained z and x displacements, EQ.7: constrained x, y, and z displacements.

For **CONTACT_SMALL_PENETRATION** option define an arbitrary number of cards giving a list of contact surface ID numbers where the small penetration check is to be turned on. This input terminates when the next "*" card is encountered. See the PENCHK variable on the *CONTACT definition.

Card 1…	1	2	3	4	5	6	7	8
Variable	ID1	ID2	ID3	ID4	ID5	ID6	ID7	ID8
Type	I	I	I	I	I	I	I	I

VARIABLE	DESCRIPTION
ID*n*	Contact ID for surface number *n*.

The **CURVE_DEFINITION** option allows a load curve to be redefined. *The new load curve must contain the same number of points as the curve it replaces.* The curve should be defined in the DEFINE_CURVE section of this manual. This input terminates when the next "*" card is encountered. Any offsets and scale factors are ignored.

Card	1	2	3	4	5	6	7	8
Variable	LCID							
Type	I							

VARIABLE	DESCRIPTION
LCID	Load curve ID

The **RIGID_BODY_CONSTRAINT** option allows translational and rotational boundary conditions on a rigid body to be changed. This input terminates when the next "*" card is encountered. Also, see *CONSTRAINED_RIGID_BODIES.

Card	1	2	3	4	5	6	7	8
Variable	PID	TC	RC					
Type	I	I	I					

VARIABLE	DESCRIPTION
PID	Part ID, see *PART.
TC	Translational constraint: EQ.0: no constraints, EQ.1: constrained x displacement, EQ.2: constrained y displacement, EQ.3: constrained z displacement, EQ.4: constrained x and y displacements, EQ.5: constrained y and z displacements, EQ.6: constrained z and x displacements, EQ.7: constrained x, y, and z displacements.
RC	Rotational constraint: EQ.0: no constraints, EQ.1: constrained x rotation, EQ.2: constrained y rotation, EQ.3: constrained z rotation, EQ.4: constrained x and y rotations, EQ.5: constrained y and z rotations, EQ.6: constrained z and x rotations, EQ.7: constrained x, y, and z rotations.

The **RIGID_BODY_INERTIA** option allows the mass and inertia properties of a rigid body to be changed. This input terminates when the next "*" card is encountered. The inertia tensor is defined in the local system defined in *MAT_RIGID at the start of the calculation. This coordinate system, which is fixed in the rigid body, tracks the rigid body rotation.

Card	1	2	3	4	5	6	7	8
Variable	ID	PID	TM					
Type	I	I	F					

Card 2

Variable	IXX	IXY	IXZ	IYY	IYZ	IZZ		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
ID	ID for this change inertia input.
PID	Part ID, see *PART.
TM	Translational mass.
IXX	I_{xx}, xx component of inertia tensor.
IXY	I_{xy}
IXZ	I_{xz}
IYY	I_{yy}, yy component of inertia tensor.
IYZ	I_{yz}
IZZ	I_{zz}, zz component of inertia tensor.

The **RIGID_BODY_STOPPER** option allows existing stoppers to be redefined. This input terminates when the next "*" card is encountered. See *CONSTRAINED_RIGID_BODY_ STOPPERS.

New stopper definitions cannot be introduced in this section. Existing stoppers can be modified.

Card 1

Variable	PID	LCMAX	LCMIN	PSIDMX	PSIDMN	LCVMNX	DIR	VID
Type	I	I	I	I	I	I	I	I
Default	required	0	0	0	0	0	required	0

Card 2

Variable	BIRTH	DEATH						
Type	F	F						
Default	0	10^{28}						

VARIABLE	DESCRIPTION
PID	Part ID of master rigid body, see *PART.
LCMAX	Load curve ID defining the maximum coordinate as a function of time: EQ.0: no limitation of the maximum displacement. New curves can be defined by the *DEFINE_CURVE within the present restart deck.
LCMIN	Load curve ID defining the minimum coordinate as a function of time: EQ.0: no limitation of the minimum displacement. New curves can be defined by the *DEFINE_CURVE within the present restart deck.
PSIDMX	Optional part set ID of rigid bodies that are slaved in the maximum coordinate direction to the master rigid body. This option requires additional input by the *SET_PART definition.

VARIABLE	DESCRIPTION
PSIDMN	Optional part set ID of rigid bodies that are slaved in the minimum coordinate direction to the master rigid body. This option requires additional input by the *SET_PART definition.
LCVMNX	Load curve ID which defines the maximum absolute value of the velocity that is allowed within the stopper: EQ.0: no limitation of the minimum displacement.
DIR	Direction stopper acts in: EQ.1: x-translation, EQ.2: y-translation, EQ.3: z-translation, EQ.4: arbitrary, defined by vector VID, EQ.5: x-axis rotation, EQ.6: y-axis rotation, EQ.7: z-axis rotation, EQ.8: arbitrary, defined by vector VID.
VID	Vector for arbitrary orientation of stopper. The vector must be defined by a *DEFINE_VECTOR within the present restart deck.
BIRTH	Time at which stopper is activated.
DEATH	Time at which stopper is deactivated.

Remarks:

The optional definition of part sets in minimum or maximum coordinate directions allows the motion to be controlled in an arbitrary direction.

The **STATUS_REPORT_FREQUENCY** option allows the output status interval to be changed.

Card	1	2	3	4	5	6	7	8
Variable	IKEDIT							
Type	I							

VARIABLE	DESCRIPTION
IKEDIT	Problem status report interval steps in the D3HSP output file: EQ.0: interval remains unchanged.

The **THERMAL_PARAMETERS** option allows parameters used by a thermal or coupled structural/thermal analysis to be changed. These parameters were initially defined on the *CONTROL_THERMAL cards. Two cards are defined for this option.

Card 1	1	2	3	4	5	6	7	8
Variable	TS	DT	TMIN	TMAX	DTEMP	TSCP		
Type	I	F	F	F	F	F		

Card 2

Variable	REFMAX	TOL						
Type	I	F						

VARIABLE	DESCRIPTION
TS	Thermal time step code: EQ.0: No change, EQ.1: Fixed time step, EQ.2: variable time step.
DT	Thermal time step on restart: EQ.0: No change.
TMIN	Minimum thermal time step: EQ.0: No change.
TMAX	Maximum thermal time step: EQ.0: No change.
DTEMP	Maximum temperature change in a thermal time step: EQ.0: No change.
TSCP	Time step control parameter (0.0 < TSCP < 1.0): EQ.0: No change.
REFMAX	Maximum number of reformations per thermal time step: EQ.0: No change.
TOL	Non-linear convergence tolerance: EQ.0: No change.

The **VELOCITY_NODE** and the **VELOCITY_NODE_ONLY** options allow the velocity of nodal points to be changed at restart. Termination of this input is when the next "*" card is read. Undefined nodes will have their nodal velocities reset to zero if a *CHANGE_VELOCITY_ NODE definition is encountered in the restart deck. However, if any of the *CHANGE_VELOCITY or CHANGE_VELOCITY_NODE definitions have _ONLY appended, then only the specified nodes will have their nodal velocities modified.

Card	1	2	3	4	5	6	7	8
Variable	NID	VX	VY	VZ	VXR	VYR	VZR	
Type	I	F	F	F	F	F	F	
Default	none	0.	0.	0.	0.	0.	0.	

VARIABLE	DESCRIPTION
NID	Node ID
VX	Translational velocity in x-direction.
VY	Translational velocity in y-direction.
VZ	Translational velocity in z-direction.
VXR	Rotational velocity about the x-axis.
VYR	Rotational velocity about the y-axis.
VZR	Rotational velocity about the z-axis.

Remarks:

1. If a node is initialized on more than one input card set, then the last set input will determine its velocity, unless it is specified on a *CHANGE_VELOCITY_NODE card.

2. If both *CHANGE_VELOCITY and *CHANGE_VELOCITY_ZERO cards are defined then all velocities will be reset to zero.

The **VELOCITY** and **VELOCITY_ONLY** options allow a new velocity field to be imposed at restart. Termination of this input is when the next "*" card is read. Undefined nodes will have their nodal velocities reset to zero if a *CHANGE_VELOCITY definition is encountered in the restart deck. However, if any of the *CHANGE_VELOCITY definitions have _ONLY appended, then only the specified nodes will have their nodal velocities modified.

Card 1

Variable	NSID							
Type	I							
Default	none							
Remark	1							

Card 2

Variable	VX	VY	VZ	VXR	VYR	VZR		
Type	F	F	F	F	F	F		
Default	0.	0.	0.	0.	0.	0.		

VARIABLE	DESCRIPTION
NSID	Nodal set ID containing nodes for initial velocity.
VX	Velocity in x-direction.
VY	Velocity in y-direction.
VZ	Velocity in z-direction.
VXR	Rotational velocity about the x-axis.
VYR	Rotational velocity about the y-axis.
VZR	Rotational velocity about the z-axis.

Remarks:

1. If a node is initialized on more than one input card set, then the last set input will determine its velocity, unless it is specified on a *CHANGE_VELOCITY_NODE card.

2. Undefined nodes will have their nodal velocities set to zero if a *CHANGE_VELOCITY definition is encountered in the restart deck.

3. If both *CHANGE_VELOCITY and *CHANGE_VELOCITY_ZERO cards are defined then all velocities will be reset to zero.

The **VELOCITY_RIGID_BODY** option allows the velocity components of a rigid body to be changed at restart. Termination of this input is when the next "*" card is read.

Card	1	2	3	4	5	6	7	8
Variable	PID	VX	VY	VZ	VXR	VYR	VZR	
Type	I	F	F	F	F	F	F	
Default	none	0.	0.	0.	0.	0.	0.	

VARIABLE	DESCRIPTION
PID	Part ID of rigid body.
VX	Translational velocity in x-direction.
VY	Translational velocity in y-direction.
VZ	Translational velocity in z-direction.
VXR	Rotational velocity about the x-axis.
VYR	Rotational velocity about the y-axis.
VZR	Rotational velocity about the z-axis.

Remarks:

1. Rotational velocities are defined about the center of mass of the rigid body.

2. Rigid bodies not defined in this section will not have their velocities modified.

The **VELOCITY_ZERO** option resets the velocities to zero at the start of the restart. Only the *CHANGE_VELOCITY_ZERO card is required for this option without any further input.

*CONTROL_DYNAMIC_RELAXATION

Purpose: Define controls for dynamic relaxation.

Card	1	2	3	4	5	6	7	8
Variable	NRCYCK	DRTOL	DRFCTR	DRTERM	TSSFDR	IRELAL	EDTTL	IDRFLG
Type	I	F	F	F	F	I	F	I
Default	250	0.001	0.995	infinity	TSSFAC	0	0.0	0
Remarks	1	1	1	1	1			1

VARIABLE	DESCRIPTION
NRCYCK	Number of iterations between convergence checks, for dynamic relaxation option (default = 250).
DRTOL	Convergence tolerance for dynamic relaxation option (default = 0.001).
DRFCTR	Dynamic relaxation factor (default = .995).
DRTERM	Optional termination time for dynamic relaxation. Termination occurs at this time or when convergence is attained (default = infinity).
TSSFDR	Scale factor for computed time step during dynamic relaxation. If zero, the value is set to TSSFAC defined on *CONTROL_TERMINATION. After converging, the scale factor is reset to TSSFAC.
IRELAL	Automatic control for dynamic relaxation option based on algorithm of Papadrakakis [1981].
EDTTL	Convergence tolerance on automatic control of dynamic relaxation.
IDRFLG	Dynamic relaxation flag for stress initialization: EQ.0: not active, EQ.1: dynamic relaxation is activated.

Remarks:

1. If a dynamic relaxation relaxation analysis is being restarted at a point before convergence was obtained, then NRCYCK, DRTOL, DRFCTR, DRTERM and TSSFDR will default to their previous values, and IDRFLG will be set to 1.

2. If dynamic relaxation is activated after a restart from a normal transient analysis LS-DYNA continues the output of data as it would without the dynamic relaxation being active. This is unlike the dynamic relaxation phase at the beginning of the calculation when a separate database is not used. Only load curves that are flagged for dynamic relaxation are applied after restarting.

*CONTROL_SHELL

Purpose: Change failure parameters NFAIL1 and NFAIL2 if necessary. These parameters must be nonzero in the initial run.

Card 1

	1	2	3	4	5	6	7	8
Variable								
Type								

Card 2

Variable						NFAIL1	NFAIL4	
Type						I	I	

VARIABLE	DESCRIPTION
NFAIL1	Flag to check for highly distorted under-integrated shell elements, print a message, and delete the element or terminate. Generally, this flag is not needed for one point elements that do not use the warping stiffness. A distorted element is one where a negative jacobian exists within the domain of the shell, not just at integration points. The checks are made away from the integration points to enable the bad elements to be deleted before an instability leading to an error termination occurs. This test will increase CPU requirements for one point elements. EQ.1: print message and delete element. EQ.2: print message, write D3DUMP file, and terminate GT.2: print message and delete element. When NFAIL1 elements are deleted then write D3DUMP file and terminate. These NFAIL1 failed elements also include all shell elements that failed for other reasons than distortion. Before the D3DUMP file is written, NFAIL1 is doubled, so the run can immediately be continued if desired.
NFAIL4	Flag to check for highly distorted fully-integrated shell elements, print a message, and delete the element or terminate. Generally, this flag is recommended. A distorted element is one where a negative jacobian exists within the domain of the shell, not just at integration points.

VARIABLE	DESCRIPTION
	The checks are made away from the integration points to enable the bad elements to be deleted before an instability leading to an error termination occurs. EQ.1: print message and delete element. EQ.2: print message, write D3DUMP file, and terminate GT.2: print message and delete element. When NFAIL4 elements are deleted then write D3DUMP file and terminate. These NFAIL4 failed elements also include all shell elements that failed for other reasons than distortion. Before the D3DUMP file is written, NFAIL4 is doubled, so the run can immediately be continued if desired.

*CONTROL_TERMINATION

Purpose: Stop the job.

Card	1	2	3	4	5	6	7	8
Variable	ENDTIM	ENDCYC						
Type	F	I						

VARIABLE	DESCRIPTION
ENDTIM	Termination time: EQ.0.0: Termination time remains unchanged.
ENDCYC	Termination cycle. The termination cycle is optional and will be used if the specified cycle is reached before the termination time. EQ.0.0: Termination cycle remains unchanged.

This is a reduced version of the *CONTROL_TERMINATION card used in the initial input deck.

***CONTROL_TIMESTEP**

Purpose: Set time step size control using different options.

Card	1	2	3	4	5	6	7	8
Variable	DUMMY	TSSFAC	ISDO	DUMMY	DT2MS	LCTM		
Type	F	F	I	F	F	I		

VARIABLE	DESCRIPTION
DUMMY	Dummy field, see remark 1 below.
TSSFAC	Scale factor for computed time step. EQ.0.0: TSSFAC remains unchanged.
ISDO	Basis of time size calculation for 4-node shell elements, ISDO 3-node shells use the shortest altitude for options 0,1 and the shortest side for option 2. This option has no relevance to solid elements, which use a length based on the element volume divided by the largest surface area: EQ.0: characteristic length=area/(longest side), EQ.1: characteristic length=area/(longest diagonal), EQ.2: based on bar wave speed and MAX **[**shortest side, area/longest side**]**. THIS LAST OPTION CAN GIVE A MUCH LARGER TIME STEP SIZE THAT CAN LEAD TO INSTABILITIES IN SOME APPLICATIONS, ESPECIALLY WHEN TRIANGULAR ELEMENTS ARE USED.
DUMMY	Dummy field, see remark 1 below.
DT2MS	New time step for mass scaled calculations. Mass scaling must be active in the time zero analysis. EQ.0.0: DT2MS remains unchanged.
LCTM	Load curve ID that limits maximum time step size: EQ.0: LCTM remains unchanged.

Remarks:

1. This a reduced version of the *CONTROL_TIMESTEP used in the initial analysis. The dummy fields are included to maintain compatibility. If using free format input then a 0.0 should be entered for the dummy values.

*DAMPING_GLOBAL

Purpose: Define mass weighted nodal damping that applies globally to the deformable nodes.

Card	1	2	3	4	5	6	7	8
Variable	LCID	VALDMP						
Type	I	F						
Default	0	0.0						

VARIABLE	DESCRIPTION
LCID	Load curve ID which specifies node system damping: EQ.n: system damping is given by load curve n. The damping force applied to each node is f=-d(t) mv, where d(t) is defined by load curve n.
VALDMP	System damping constant, d (this option is bypassed if the load curve number defined above is nonzero).

DATABASE_OPTION*

Options for ASCII files include. If a file is not specified in the restart deck then the output interval for the file will remain unchanged.

SECFORC	Cross section forces.
RWFORC	Wall forces.
NODOUT	Nodal point data.
ELOUT	Element data.
GLSTAT	Global data.
DEFORC	Discrete elements.
MATSUM	Material energies.
NCFORC	Nodal interface forces.
RCFORC	Resultant interface forces.
DEFGEO	Deformed geometry file
SPCFORC	Set dt for spc reaction forces.
SWFORC	Nodal constraint reaction forces (spot welds and rivets).
ABSTAT	Set dt for airbag statistics.
NODFOR	Set dt for nodal force groups.
BNDOUT	Boundary condition forces and energy
RBDOUT	Set dt for rigid body data.
GCEOUT	Set dt for geometric contact entities.
SLEOUT	Set dt for sliding interface energy.
JNTFORC	Set dt for joint force file.
SBTOUT	Set dt for seat belt output file.
AVSFLT	Set dt for AVS database.
MOVIE	Set dt for MOVIE.
MPGS	Set dt for MPGS.
TPRINT	Set dt for thermal file.

Card	1	2	3	4	5	6	7	8
Variable	DT							
Type	F							

VARIABLE	DESCRIPTION
DT	Time interval between outputs: EQ.0.0: output interval is unchanged.

To terminate output to a particular file set DT to a high value.

If IACCOP=2 was specified in *CONTROL_OUTPUT, the best results are obtained in the NODOUT file by keeping the same DT on restart. When DT is changed for NODOUT, oscillations may occur around the restart time. If DT is larger than initially specified in the original input file, more memory is required to store the time states for the averaging than was originally allocated. A warning message is printed, and the filtering is applied using the available memory. When DT is smaller than initially specified, more oscillations may appear in the output than earlier in the calculation because the frequency content of the averaged output increases as DT decreases.

DATABASE_BINARY_*OPTION

Options for binary output files with the default names given include:

D3PLOT Dt for complete output states.

D3THDT Dt for time history data for element subsets.

D3DUMP Binary output restart files. Define output frequency in cycles

RUNRSF Binary output restart file. Define output frequency in cycles.

INTFOR Dt for contact surface Interface database.

Card	1	2	3	4	5	6	7	8
Variable	DT/CYCL							
Type	F							

VARIABLE	DESCRIPTION
DT	Time interval between outputs. EQ.0.0: Time interval remains unchanged.
CYCL	Output interval in time steps. EQ.0.0: output interval remains unchanged.

DELETE_OPTION*

Available options are:

CONTACT

CONTACT_2DAUTO

ENTITY

PART

ELEMENT_BEAM

ELEMENT_SHELL

ELEMENT_SOLID

ELEMENT_TSHELL

FSI

Purpose: Delete contact surfaces, ALE FSI couplings, parts, or elements by a list of IDs. There are two contact algorithms for two-dimensional problems: the line-to-line contact and the automatic contact defined by part ID's. Each uses their own numbering.

For **CONTACT**, **CONTACT_2DAUTO**, **ENTITY**, **FSI**, or **PART** option.

Card	1	2	3	4	5	6	7	8
Variable	ID1	ID2	ID3	ID4	ID5	ID6	ID7	ID8
Type	I	I	I	I	I	I	I	I

VARIABLE	DESCRIPTION
IDI	Contact ID/Coupling ID/Part ID

For *DELETE_CONTACT/*DELETE_FSI a negative ID implies that the absolute value gives the contact surface/FSI coupling which is to be activated.

For the four **ELEMENT** options. Termination of input is when the next "*" card is read.

Card	1	2	3	4	5	6	7	8
Variable	ESID							
Type	I							

VARIABLE	DESCRIPTION
ESID	Element set ID, see *SET_SOLID, *SET_BEAM, *SET_SHELL, *SET_TSHELL.

*INTERFACE_SPRINGBACK

Purpose: Define a material subset for an implicit springback calculation in LS-NIKE3D and any nodal constraints to eliminate rigid body degrees-of-freedom. Generally, only the materials that make up the original blank are included in the springback calculation. After termination of the LS-DYNA3D computation, an input deck for LS-NIKE3D and a stress initialization file for LS-NIKE3D are written.

Card	1	2	3	4	5	6	7	8
Variable	PSID							
Type	I							

VARIABLE	DESCRIPTION
PSID	Part set ID for springback, see *SET_PART.

Define a list of nodal points that are constrained for the springback. This section is terminated by an "*" indicating the next input section.

Card	1	2	3	4	5	6	7	8
Variable	NID	TC	RC					
Type	I	F	F					
Default	none	0.	0.					

VARIABLE	DESCRIPTION
NID	Node ID
TC	Translational constraint: EQ.0: no constraints, EQ.1: constrained x displacement, EQ.2: constrained y displacement, EQ.3: constrained z displacement, EQ.4: constrained x and y displacements,

VARIABLE	DESCRIPTION
	EQ.5: constrained y and z displacements, EQ.6: constrained z and x displacements, EQ.7: constrained x, y, and z displacements.
RC	Rotational constraint: EQ.0: no constraints, EQ.1: constrained x rotation, EQ.2: constrained y rotation, EQ.3: constrained z rotation, EQ.4: constrained x and y rotations, EQ.5: constrained y and z rotations, EQ.6: constrained z and x rotations, EQ.7: constrained x, y, and z rotations.

RIGID_DEFORMABLE_OPTION*

Available options include:

CONTROL

D2R (Deformable to rigid part switch)

R2D (Rigid to deformable part switch)

Purpose: Define parts to be switched from rigid to deformable and deformable to rigid in a restart. It is only possible to switch parts on a restart if part switching was activated in the time zero analysis. See *DEFORMABLE_TO_RIGID for details of part switching.

*RIGID_DEFORMABLE_CONTROL

Card	1	2	3	4	5	6	7	8
Variable	NRBF	NCSF	RWF	DTMAX				
Type	I	I	I	F				
Default	0	0	0	none				

VARIABLE	DESCRIPTION
NRBF	Flag to delete or activate nodal rigid bodies. If nodal rigid bodies or generalized, weld definitions are active in the deformable bodies that are switched to rigid, then the definitions should be deleted to avoid instabilities: EQ.0: no change, EQ.1: delete, EQ.2: activate.
NCSF	Flag to delete or activate nodal constraint set. If nodal constraint/spot weld definitions are active in the deformable bodies that are switched to rigid, then the definitions should be deleted to avoid instabilities: EQ.0: no change, EQ.1: delete, EQ.2: activate.
RWF	Flag to delete or activate rigid walls: EQ.0: no change, EQ.1: delete, EQ.2: activate.
DTMAX	Maximum permitted time step size after restart.

*RIGID_DEFORMABLE_D2R

Termination of this input is when the next "*" card is read.

Card	1	2	3	4	5	6	7	8
Variable	PID	MRB						
Type	I	I						
Default	none	0						

VARIABLE	DESCRIPTION
PID	Part ID of the part which is switched to a rigid material.
MRB	Part ID of the master rigid body to which the part is merged. If zero, the part becomes either an independent or master rigid body.

***RIGID_DEFORMABLE_R2D**

Termination of this input is when the next "*" card is read.

Card	1	2	3	4	5	6	7	8
Variable	PID							
Type	I							
Default	none							

VARIABLE	DESCRIPTION
PID	Part ID of the part which is switched to a deformable material.

***STRESS_INITIALIZATION_**{*OPTION*}

This keyword allows a full deck restart to be performed in LS-DYNA. For a full deck restart a complete input deck has to be included in the restart deck. The stress initialization feature allows all or selected parts to be initialized on restart, using data from the d3dump or runrsf database.

The options that are available with this keyword are:

<BLANK>

DISCRETE

SEATBELT

***STRESS_INITIALIZATION**

If this card is specified without further input as described below then all parts in the new input deck that existed in the previous input deck (with or without the same part IDs) are initialized from the d3dump or runrsf database. Further all seatbelt and discrete parts are initialized.

If only a subset of parts is to be initialized in the new analysis then define as many of the following cards as necessary. Termination of this input is when the next "*" card is read.

Card 1…	1	2	3	4	5	6	7	8
Variable	PIDO	PIDN						
Type	I	I						
Default	none	PIDO						

VARIABLE	DESCRIPTION
PIDO	Old part ID, see *PART.
PIDN	New part ID, see *PART: EQ.0: New part ID is the same as the old part ID.

Remarks:

If one or more of the above cards are defined then discrete and seatbelt elements will not be initialized unless the additional option cards *STRESS_INITIALIZATION_DISCRETE and *STRESS_INITIALIZATION_SEATBELT are defined.

*STRESS_INITIALIZATION_DISCRETE

Initialize all discrete parts from the old parts. No further input is required with this card. This card is not required if *STRESS_INITIALIZATION is specified without further input.

*STRESS_INITIALIZATION_SEATBELT

Initialize all seatbelt parts from the old parts. No further input is required with this card. This card is not required if *STRESS_INITIALIZATION is specified without further input.

TERMINATION_OPTION*

Purpose: Stops the job depending on some displacement conditions.

Available options include:

NODE

BODY

Caution: The inputs are different for the nodal and rigid body stop conditions. The nodal stop condition works on the global coordinate position, while the body stop condition works on the relative global translation. The number of termination conditions cannot exceed the maximum of 10 or the number specified in the original analysis.

The analysis terminates for *TERMINATION_NODE when the current position of the node specified reaches either the maximum or minimum value (stops 1, 2 or 3), or picks up force from any contact surface (stop 4). For *TERMINATION_BODY the analysis terminates when the center of mass displacement of the rigid body specified reaches either the maximum or minimum value (stops 1, 2 or 3) or the displacement magnitude of the center of mass is exceeded (stop 4). If more than one condition is input, the analysis stops when any of the conditions is satisfied. ***This input completely overrides the existing termination conditions defined in the time zero run.***

Termination by other means is controlled by the *CONTROL_TERMINATION control card.

For both options, the input is identical:

Card	1	2	3	4	5	6	7	8
Variable	NID/PID	STOP	MAXC	MINC				
Type	I	I	F	F				
Default	none	none	-	-				

For the **NODE** option:

VARIABLE	DESCRIPTION
NID	Node ID
STOP	Stop criterion: EQ.1: global x direction, EQ.2: global y direction, EQ.3: global z direction, EQ.4: stop if node touches contact surface.
MAXC	Maximum (most positive) coordinate, options 1, 2 and 3 above only.
MINC	Minimum (most negative) coordinate, options 1, 2 and 3 above only.

For the **BODY** option:

VARIABLE	DESCRIPTION
PID	Part ID of rigid body
STOP	Stop criterion: EQ.1: global x direction, EQ.2: global y direction, EQ.3: global z direction, EQ.4: stop if displacement magnitude is exceeded.
MAXC	Maximum (most positive) displacement, options 1, 2, 3 and 4: EQ.0.0: MAXC set to 1.0e21
MINC	Minimum (most negative) displacement, options 1, 2 and 3 above only: EQ.0.0: MINC set to -1.0e21

*TITLE

Purpose: Define job title.

Card	1	2	3	4	5	6	7	8
Variable	TITLE							
Type	C							
Default	LS-DYNA USER INPUT							

VARIABLE	DESCRIPTION
TITLE	Heading to appear on output.

REFERENCES

Abbo, A.J., and S.W. Sloan, “A Smooth Hyperbolic Approximation to the Mohr-Coulomb Yield Criterioin,” *Computers and Structures*, Vol. 54, No. 1, (1995).

Allen, D.J., Rule, W.K., Jones, S.E., “Optimizing Material Strength Constants Numerically Extracted from Taylor Impact Data”, *Experimental Mechanics*, Vol. 37, No 3, September (1997).

Allman, D.J., “A Compatible Triangular Element Including Vertex Rotations for Plane Elasticity Analysis,” *Computers and Structures*, 19, 1-8, (1984).

Anagonye, A.U. and J.T. Wang, “A Semi-Empirical Method for Estimating the Effective Leak and Vent Areas of an Airbag”, AMD-Vol. 237/BED-Vol. 45, pp. 195-217, (1999).

Anand, L. and M.E. Gurtin, “A theory of amorphous solids undergoing large deformations, with application to polymeric glasses,” *International Journal of Solids and Structures*, 40, pp. 1465-1487 (2003)

Aretz, H. “Applications of a New Plane Stress Yield Function to Orthotropic Steel and Aluminum Sheet Metals,” *Modeling and Simulation in Materials Science and Engineering*, 12, 491-509 (2004).

Argon, AS., “A theory for the low-temperature plastic deformation of glassy polymers”, *Philosophical Magazine*, 28, 839-865 (1973).

Armstrong, P.J., and Frederick, C.O., “A Mathematical Representation of the Multiaxial Bauschinger Effect,” CEGB Report, RD/B/N731, Berkeley Nuclear Laboratories (1966).

Arruda, E. and M. Boyce, "A Three-Dimensional Constitutive Model for the Large Stretch Behavior of Rubber Elastic Materials," *Journal of the Mechanics and Physics of Solids*, Vol. 41, No. 2, pp. 389-412, (1993).

Auricchio, F., R.L. Taylor and J. Lubliner, "Shape-memory alloys: macromodeling and numerical simulations of the superelastic behavior", *Computer Methods in Applied Mechanics and Engineering*, vol. 146, pp. 281-312, (1997).

Auricchio, F. and R.L. Taylor, "Shape-memory alloys: modeling and numerical simulations of the finite-strain superelastic behavior", *Computer Methods in Applied Mechanics and Engineering*, vol. 143, pp. 175-194, (1997).

Bahler AS: The series elastic element of mammalian skeletal muscle. *Am J Physiol* 213:1560-1564, (1967).

Baker, E.L., "An Explosives Products Thermodynamic Equation of State Appropriate for Material Acceleration and Overdriven Detonation: Theoretical Background and

Fourmulation," Technical Report ARAED-TR-911013, U.S. Army Armament Research, Development and Engineering Center, Picatinney Arsenal, New Jersey, 1991).

Baker, E.L. and J. Orosz, J., "Advanced Warheads Concepts: An Advanced Equation of State for Overdriven Detonation," Technical Report ARAED-TR-911007, U.S. Army Armament Research, Development and Engineering Center, Picatinney Arsenal, New Jersey, (1991).

Baker, E.L. and L.I. Stiel, "Improved Quantitative Explosive Performance Prediction Using Jaguar," 1997 Insensitive Munitions and Energetic Materials Technology Symposium, Tampa, FL, (1997).

Bammann, D.J. and E.C. Aifantis, "A Model for Finite-Deformation Plasticity," *Acta Mechanica*, 70, 1-13 (1987).

Bammann, D.J. and G. Johnson, "On the Kinematics of Finite-Deformation Plasticity," *Acta Mechanica, 69,* 97-117 (1987).

Bammann, D.J., "Modeling the Temperature and Strain Rate Dependent Large Deformation of Metals," Proceedings of the 11th US National Congress of Applied Mechanics, Tuscon, AZ, (1989).

Bammann, D.J., M.L. Chiesa, A. McDonald, W.A. Kawahara, J.J. Dike, and V.D. Revelli, "Predictions of Ductile Failure in Metal Structures," in AMD-Vol. 107, Failure Criteria and Analysis in Dynamic Response, Edited by. H.E. Lindberg, 7-12, (1990).

Bandak, F.A., private communications, U.S. Dept. of Trans., Division of Biomechanics Research, 400 7th St., S.W. Washington, D.C. 20590 (1991).

Barlat, F. and J. Lian, "Plastic Behavior and Stretchability of Sheet Metals. Part I: A Yield Function for Orthotropic Sheets Under Plane Stress Conditions," *Int. J. of Plasticity*, Vol. 5, pp. 51-66 (1989).

Barlat, F., D.J. Lege, and J.C. Brem, "A Six-Component Yield Function for Anisotropic Materials," *Int. J. of Plasticity*, 7, 693-712, (1991).

Barlat, F., Y. Maeda, K. Chung, M. Yanagawa, J.C. Brem, Y. Hayashida, D.J. Lege, K. Matsui, S.J. Murtha, S. Hattori, R.C. Becker, and S. Makosey, "Yield Function Development for Aluminum Alloy Sheets", J. Mech. Phys. Solids, Vol. 45, No. 11-12, 1727-1763, (1997).

Barlat, F., Brem, J.C., Yoon, J.W., Chung, K., Dick, R.E., Lege, D.J., Pourboghrat, F., Choi, S.H., Chu, E., "Plane Stress Yield Function for Aluminum Alloy Sheets – Part 1: Theory, Int. J. Plast. 19, 1-23, (2003).

Basu, U., "Explicit finite element perfectly matched layer for transient three-dimensional elastic waves," *International Journal for Numerical Methods in Engineering*, vol. 77, pp. 151–176, (2009).

Basu, U. and Chopra, A.K., "Perfectly matched layers for time-harmonic elastodynamics of unbounded domains theory and finite-element implementation," *Computer Methods in Applied Mechanics and Engineering*, vol. 192, pp. 1337–1375, (2003).

REFERENCES

Basu, U. and Chopra, A.K., "Perfectly matched layers for transient elastodynamics of unbounded domains," *International Journal for Numerical Methods in Engineering*, vol. 59, pp. 1039–1074, (2004). Erratum: Ibid. vol. 61, pp. 156–157, (2004).

Bathe, K.-J. and Dvorkin, E.N. A four node plate bending element based on Mindlin-Reissner plate theory and a mixed interpolation, *Int. J. Num. Meth. Eng.*, 21, 367-383 (1985).

Batoz, J.L. and Ben Tahar, M. Evaluation of a new quadrilateral thin plate bending element, Int. *J. Num. Meth. Eng.*, 18, 1644-1677 (1982).

Batoz, J.-L. and M. Ben Tahar, Evaluation of a new quadrilateral thin plate bending element, *International Journal for Numerical Methods in Engineering*, 18, (1982), 1655-1677.

Bazeley, G.P., W.K. Cheung, R.M. Irons, and O.C. Zienkiewicz, "Triangular Elements in Plate Bending-Confirming and Nonconforming Solutions in Matrix Methods and Structural Mechanics," Proc. Conf. on Matrix Methods in Structural Analysis, Rept. AFFDL-R-66-80, Wright Patterson AFB, 547-576 (1965).

Belytschko, T. and Bindeman, L. P. "Assumed Strain Stabilization of the Eight Node Hexahedral Element," Comp. Meth. Appl. Mech. Eng. **105**, 225-260 (1993).

Belytschko, T.B. and A.H. Marchertas, "Nonlinear Finite Element Method for Plates and its Application to the Dynamic Response of Reactor Fuel Subassemblies," *Trans, ASME J. Pressure Vessel Tech.*, 251-257 (1974).

Belytschko, T.B. and C.S. Tsay, "Explicit Algorithms for Nonlinear Dynamics of Shells," AMD-Vol.48, ASME, 209-231 (1981).

Belytschko, T.B. and C.S. Tsay, "Explicit Algorithms for Nonlinear Dynamics of Shells," *Comp. Meth. Appl. Mech. Eng.*, 43, 251-276, (1984).

Belytschko, T.B. and C.S. Tsay, "A Stabilization Procedure for the Quadrilateral Plate Element with One-Point Quadrature," *Int. J. Num. Method. Eng.*, 19, 405-419 (1983).

Belytschko, T.B., H. Stolarski, and N. Carpenter, "A C^o Triangular Plate Element with One-Point Quadrature," *Int. J. Num. Meth. Eng.*, 20, 787-802 (1984).

Belytschko, T.B., L. Schwer, and M.J. Klein, "Large Displacement Transient Analysis of Space Frames," *Int. J. Num. Eng.*, 11, 65-84 (1977).

Benson, D.J. and J.O. Hallquist, "A Simple Rigid Body Algorithm for Structural Dynamics Programs," *Int. J. Numer. Meth. Eng.*, 22, (1986).

Benson, D.J. and J.O. Hallquist, "A Single Surface Contact Algorithm for the Postbuckling Analysis of Shell Structures," Comp. Meths. Appl. Mech. Eng., 78, 141-163 (1990).

Benzeggagh, M.L. and Kenane, M., "Measurement of Mixed-mode Delamination Fracture Toughness of Unidirectional Glass/Epoxy Composites with Mixed-mode Bending Apparatus," Composites Science and Technology, 56, 439-449 (1996).

REFERENCES

Berstad, T., "Material Modeling of Aluminium for Crashworthiness Analysis", Dr.Ing. Dissertation, Department of Structural Engineering, Norwegian University of Science and Technology, Trondheim, Norway, (1996).

Berstad, T., Hopperstad, O.S., Lademo, O.-G. and Malo, K.A., "Computational Model of Ductile Damage and Fracture in Shell Analysis", Second European LS-DYNA Conference, Gothenburg, Sweden, (1999).

Berstad, T., Lademo, O.-G., Pedersen, K.O. and Hopperstad, O.S., *"Formability modeling with LS-DYNA"*, 8th International LS-DYNA User's Conference, Detroit, May 3-5, 2004.

Berstad, T., Langseth, M. and Hopperstad, O.S., "Elasto-viscoplastic Constitutive Models in the Explicit Finite Element Code LS-DYNA3D," Second International LS-DYNA3D conference, San Francisco, (1994).

Bielak, J. and Christiano, P., "On the effective seismic input for non-linear soil-structure interaction systems," *Earthquake Engineering and Structural Dynamics*, vol. 12, pp. 107–119, (1984).

Bilkhu, S.S., M. Founas, and G.S. Nasholtz, "Material Modeling of Structural Foams in Finite Element Analysis Using Compressive Uniaxial and Triaxial Data," SAE (Nat. Conf.) Detroit 1993, pp. 4-34.

Blatz, P.J., and Ko, W.L., "Application of Finite Element Theory to the Deformation of Rubbery Materials," *Trans. Soc. of Rheology*, 6, 223-251 (1962).

Boyce, M.C., Parks, D.M., and Argon, A.S., "Large inelastic deformation of glassy polymers. Part I: Rate dependent constitutive model". *Mechanics of Materials*, 7, 15-33 (1988).

Boyce, M.C., Socrate, C. and Llana, P.G., "Constitutive model for the finite deformation stress-strain behavior of poly(ethylene terephthalate) above the glass transition". *Polymer*, 41, 2183-2201 (2000).

Brekelmans, W.A.M., Scheurs,P.J.G., and de Vree, J.H.P., 1991, "Continuum damage mechanics for softening of brittle materials", *Acta Mechanica*, vol 93, pp 133-143

Broadhouse, B.J., "The Winfrith Concrete Model in LS-DYNA3D," Report: SPD/D(95)363, Structural Performance Department, AEA Technology, Winfrith Technology Centre, U.K. (1995).

Broadhouse, B.J. and Neilson, A.J., "Modelling Reinforced Concrete Structures in DYNA3D", Safety and Engineering Division, United Kingdom Atomic Energy Authority, Winfrith, AEEW-M 2465, 1987.

Brode, H.L., "Height of Burst Effects at High Overpressure," RAND, RM-6301-DASA, DASA 2506, (1970).

REFERENCES

Brown, B.E. and J.O. Hallquist, "TAURUS: An Interactive Post-Processor for the Analysis Codes NIKE3D, DYNA3D, TACO3D, and GEMINI," University of California, Lawrence Livermore National Laboratory, Rept. UCID-19392 (1982) Rev. 1 (1984).

Bruneau, M., Uang, C.M., Whittaker, A., Ductile Design of Steel Structures, McGraw Hill, (1998).

Burton, D.E. et al. "Physics and Numerics of the TENSOR Code," Lawrence Livermore National Laboratory, Internal Document UCID-19428, (July 1982).

CEB Code 1993, Comite euro-international du beton, *CEB-FIP Model Code 1990*, Thomas Telford, London, (1993).

Chang, F.K. and K.Y. Chang, "A Progressive Damage Model for Laminated Composites Containing Stress Concentration," *J. of Composite Materials*, 21, 834-855 (1987a).

Chang, F.K. and K.Y. Chang, "Post-Failure Analysis of Bolted Composite Joints in Tension or Shear-Out Mode Failure," *J. of Composite Materials*, 21 809-823 (1987b).

Chang, F.S., "Constitutive Equation Development of Foam Materials," Ph.D. Dissertation, submitted to the Graduate School, Wayne State University, Detroit, Michigan (1995).

Chen, W.F., and Baladi, G.Y., Soil Plasticity: Theory and Implementation, Elesvier, New York, (1985).

Cheng, H., Obergefell, L.A., and Rizer, A., March 1994, "Generator of Body (GEBOD) Manual," Report No. AL/CF-TR-1994-0051.

Chowdhury, S.R. and Narasimhan R., "A Cohesive Finite Element Formulation for Modeling Fracture and Delamination in Solids," *Sadhana*, 25(6), 561-587, (2000).

Christensen, R.M. "A Nonlinear Theory of Viscoelasticity for Application to Elastomers," *Journal of Applied Mechanics*, Volume 47, American Society of Mechanical Engineers, pages 762-768, December 1980.

Chu, C.C. and A. Needleman, "Void Nucleation Effects in Biaxially Stretched Sheets", ASME Journal of Engineering Materials and Technology, 102, 249-256 (1980).

Chung, K. and K. Shah, "Finite Element Simulation of Sheet Metal Forming for Planar Anisotropic Metals," *Int. J. of Plasticity*, 8, 453-476, (1992).

Cochran, S.G. and J. Chan, "Shock Initiation and Detonation Models in One and Two Dimensions," University of California, Lawrence Livermore National Laboratory, Rept. UCID-18024 (1979).

Cook, R. D., Concepts and Applications of Finite Element Analysis, John Wiley and Sons, Inc. (1974).

Couch, R., E. Albright, and N. Alexander, The Joy Computer Code, Lawrence Livermore National Laboratory, Internal Document Rept. UCID-19688, (January, 1983).

REFERENCES

Cowper, G.R. and P.S. Symonds, Strain Hardening and Strain Rate Effects in the Impact Loading of Cantilever Beams, Brown University, Applied Mathematics Report, 1958.

CRAY-1 Computer System CFT Reference Manual, Cray Research Incorporated, Bloomington, NM., Publication No. 2240009 (1978).

DeRuntz, J.A. Jr., "Reference Material for USA, The Underwater Shock Analysis Code, USA-STAGS, and USA-STAGS-CFA," Report LMSC-P032568, Computational Mechanics Laboratory, Lockheed Palo Alto Research Laboratory, Palo Alto, CA. (1993).

Desai, C.S., and H.J. Siriwardane, Constitutive Laws for Engineering Materials with Emphasis On Geologic Materials, Prentice-Hall, Chapter 10, (1984).

Deshpande, V.S. and N.A. Fleck, "Isotropic Models for Metallic Foams," *Journal of the Mechanics and Physics of Solids*, 48, 1253-1283, (2000).

Dick, R.E., and W.H. Harris, "Full Automated Rezoning of Evolving Geometry Problems," Numerical Methods in Industrial Forming Processes, Chenot, Wood, and Zienkiewicz, Editors, Bulkema, Rotterdam, 243-248, (1992).

Dilger, W.H., R. Koch, and R. Kowalczyk, "Ductility of Plain and Confined Concrete Under Different Strain Rates," *ACI Journal*, January-February, (1984).

Dobratz, B.M., "LLNL Explosives Handbook, Properties of Chemical Explosives and Explosive Simulants," University of California, Lawrence Livermore National Laboratory, Rept. UCRL-52997 (1981).

Du Bois, P.A., "Numerical Simulation of Strandfoam" Daimler-Chrysler AG Abt. EP/CSV, Report (2001).

Dufailly, J., and Lemaitre, J., "Modeling very low cycle fatigue", *International Journal of damage mechanics*, 4, pp. 153-170 (1995).

Englemann, B. E., R.G. Whirley, and G.L. Goudreau, "A Simple Shell Element Formulation for Large-Scale Elastoplastic Analysis," CED-Vol. 3. Analytical and Computational Models of Shells, A.K. Noor, T. Belytschko, and J.C. Simo, Editors, 1989, pp. 399-416.

Faßnacht, W., "Simulation der Rißbildung in Aluminiumgußbauteilen," Dissertation, Technishe Universität Darmstadt, (1999).

Feng, W.W. and Hallquist, J.O., "On Constitutive Equations for Elastomers and Elstomeric Foams", The 4th European LS-DYNA Conference, D-II-15, Ulm, Germany, May 2003.

Feucht, M., "Ein gradientenabhängiges Gursonmodell zur Beshreibung duktiler Schädigung mit Entfestigung," Dissertation, Technishe Universität Darmstadt, (1998).

Fiolka, M. and Matzenmiller, A., "Delaminationsberechnung von Faserverbundstrukturen", *PAMM Proc. Appl. Math. Mech.* 5, S.393-394 (2005).

Flanagan, D.P. and T. Belytschko, "A Uniform Strain Hexahedron and Quadrilateral and Orthogonal Hourglass Control," *Int. J. Numer. Meths. Eng.*, 17, 679-706 (1981).

Fung, Y.C., Biomechanics, Springer, New York, 1993.

Fung, Y.C., Foundations of Solid Mechanics, Prentice Hall, Inc., Englewood Cliffs, New Jersey, 1965.

Gerlach, S., Fiolka M. and Matzenmiller, A., Modelling and analysis of adhesively bonded joints with interface elements for crash analysis, 4. LS-DYNA Forum, 20-21, (2005) Bamberg, DYNAmore GmbH, Stuttgart.

Ginsberg, M. and J. Johnson, "Benchmarking the Performance of Physical Impact Simulation Software on Vector and Parallel Computers," Applications Track of Supercomputing, IEEE monograph, Computer Society Press, March, 1989.

Giroux, E.D. HEMP User's Manual, University of California, Lawrence Livermore National Laboratory, Rept. UCRL-51079 (1973).

Goudreau, G.L. and J.O. Hallquist, "Recent Developments in Large Scale Finite Element Lagrangian Hydrocode Technology," *J. Comp. Meths. Appl. Mechs. Eng.*, 30 (1982).

Goldak, J., Chakravarti, A., and Bibby, M., "A New Finite Element Model for Welding Heat Sources," *Metallurgical Transactions B*, vol. 15B, pp. 299-305, June, 1984.

Govindjee, S., Kay, J.G., and Simo, J.C. [1994], Anisotropic Modeling and Numerical Simulation of Brittle Damage in Concrete, Report No. UCB/SEMM-94/18, Department of Civil Engineering, University of California, Berkeley, CA 94720.

Govindjee, S., Kay, J.G., and Simo, J.C. [1995], "Anisotropic Modeling and Numerical Simulation of Brittle Damage in Concrete," *Int. J. Numer. Meth. Engng*, **38**, 3611-3633.

Graefe, H., W. Krummheuer, and V. Siejak, "Computer Simulation of Static Deployment Tests for Airbags, Air Permeability of Uncoated Fabrics and Steady State Measurements of the Rate of Volume Flow Through Airbags," SAE Technical Paper Series, 901750, Passenger Car Meeting and Expositition, Dearborn, Michigan, September 17-20, 1990.

Gran, J.K. and P.E. Senseny, "Compression Bending of Scale-Model Reinforced-Concrete Walls," *ASCE Journal of Engineering Mechanics*, Volume 122, Number 7, pages 660-668, July (1996).

Guccione, J., A. McCulloch, and L. Waldman, "Passive Material Properties of Intact Ventricular Myocardium Determined from a Cylindrical Model", *ASME Journal of Biomechanical Engineering*, Vol. 113, pages 42-55, (1991).

Gurson, A.L., Plastic Flow and Fracture Behavior of Ductile Materials Incorporating Void Nucleation, Growth, and Interaction, Ph.D. Thesis, Brown University, (1975).

REFERENCES

Gurson, A.L., "Continuum Theory of Ductile Rupture by Void Nucleation and Growth: Part I - Yield Criteria and Flow Rules for Porous Ductile Media", *J. of Eng. Materials and Technology*, (1977).

Hallquist, J.O., Preliminary User's Manuals for DYNA3D and DYNAP (Nonlinear Dynamic Analysis of Solids in Three Dimension), University of California, Lawrence Livermore National Laboratory, Rept. UCID-17268 (1976) and Rev. 1 (1979).[**a**]

Hallquist, J.O., A Procedure for the Solution of Finite Deformation Contact-Impact Problems by the Finite Element Method, University of California, Lawrence Livermore National Laboratory, Rept. UCRL-52066 (1976).

Hallquist, J.O., "A Numerical Procedure for Three-Dimensional Impact Problems," *American Society of Civil Engineering*, Preprint 2956 (1977).

Hallquist, J.O., "A Numerical Treatment of Sliding Interfaces and Impact," in: K.C. Park and D.K. Gartling (eds.) *Computational Techniques for Interface Problems*, AMD Vol. 30, ASME, New York (1978).

Hallquist, J.O., NIKE2D: An Implicit, Finite-Element Code for Analyzing the Static and Dynamic Response of Two-Dimensional Solids, University of California, Lawrence Livermore National Laboratory, Rept. UCRL-52678 (1979).[**b**]

Hallquist, J.O., User's Manual for DYNA2D – An Explicit Two-Dimensional Hydrodynamic Finite Element Code with Interactive Rezoning, University of California, Lawrence Livermore National Laboratory, Rept. UCID-18756 (1980).

Hallquist, J.O., User's Manual for DYNA3D and DYNAP (Nonlinear Dynamic Analysis of Solids in Three Dimensions), University of California, Lawrence Livermore National Laboratory, Rept. UCID-19156 (1981).[**a**]

Hallquist, J. O., NIKE3D: An Implicit, Finite-Deformation, Finite-Element Code for Analyzing the Static and Dynamic Response of Three-Dimensional Solids, University of California, Lawrence Livermore National Laboratory, Rept. UCID-18822 (1981).[**b**]

Hallquist, J.O., DYNA3D User's Manual (Nonlinear Dynamic Analysis of Solids in Three Dimensions), University of California, Lawrence Livermore National Laboratory, Rept. UCID-19156 (1982; Rev. 1: 1984; Rev. 2: 1986).

Hallquist, J.O., Theoretical Manual for DYNA3D, University of California, Lawrence Livermore National Laboratory, Rept. UCID-19501 (March, 1983).

Hallquist, J.O., DYNA3D User's Manual (Nonlinear Dynamic Analysis of Solids in Three Dimensions), University of California, Lawrence Livermore National Laboratory, Rept. UCID-19156 (1988, Rev. 4).

Hallquist, J.O., LS-DYNA User's Manual (Nonlinear Dynamic Analysis of Solids in Three Dimensions), Livermore Software Technology Corporation, Rept. 1007 (1990).

Hallquist, J.O., D.J. Benson, and G.L. Goudreau, “Implementation of a Modified Hughes-Liu Shell into a Fully Vectorized Explicit Finite Element Code,” Proceedings of the International Symposium on Finite Element Methods for Nonlinear Problems, University of Trondheim, Trondheim, Norway (1985).

Hallquist, J.O. and D.J. Benson, “A Comparison of an Implicit and Explicit Implementation of the Hughes-Liu Shell,” Finite Element Methods for Plate and Shell Structures, T.J.R. Hughes and E. Hinton, Editors, 394-431, Pineridge Press Int., Swanea, U.K. (1986).

Hallquist, J.O. and D.J. Benson, DYNA3D User’s Manual (Nonlinear Dynamic Analysis of Solids in Three Dimensions), University of California, Lawrence Livermore National Laboratory, Rept. UCID-19156 (Rev. 2: 1986; Rev. 3: 1987).

Hallquist, J.O., D.W. Stillman, T.J.R. Hughes, C. and Tarver,”Modeling of Airbags Using MVMA/DYNA3D,” LSTC Report (1990).

Hashin, Z, “Failure Criteria for Unidirectional Fiber Composites,” *Journal of Applied Mechanics*, 47, 329 (1980).

Hänsel, C., P. Hora, and J. Reissner, "Model for the Kinetics of Strain-Induced Martensitic Phase Transformation at Isothermal Conditions for the Simulation of Sheet Metal Forming Processes with Metastable Austenitic Steels," Simulation of Materials Processing: Theory, Methods, and Applications, Huétink and Baaijens (eds), Balkema, Rotterdam, (1998).

Haward, R.N., and Thackray, G., “The use of a mathematical model to describe isothermal stress-strain curves in glassy thermoplastics”. *Proc Roy Soc A*, 302, 453-472 (1968).

Herrmann, L.R. and F.E. Peterson, “A Numerical Procedure for Viscoelastic Stress Analysis,” Seventh Meeting of ICRPG Mechanical Behavior Working Group, Orlando, FL, CPIA Publication No. 177, 1968.

Hill A.V., "The heat of shortening and the dynamic constants of muscle," *Proc Roy Soc* B126:136-195, (1938).

Hill, R., “A Theory of the Yielding and Plastic Flow of Anisotropic Metals,” *Proceedings of the Royal Society of London,* Series A., Vol. 193, pp. 281-197 (1948).

Hill, R., “Aspects of Invariance in Solid Mechanics,” *Advances in Applied Mechanics*, Vol. 18, pp. 1-75 (1978).

Hill, R., “Constitutive Modeling of Orthotropic Plasticity in Sheet Metals,” *J. Mech. Phys. Solids*, Vol. 38, No. 3, 1989, pp. 405-417.

Hirth, A., P. Du Bois, and K. Weimar, "Improvement of LS-DYNA Material Law 83 (Fu Chang) for the Industrial Simulation of Reversible Energy-Absorbing Foams," CAD-FEM User's Meeting, Bad Neuenahr - Ahrweiler, Germany, October 7-9, Paper 2-40, (1998).

Holmquist, T.J., G.R. Johnson, and W.H. Cook, "A Computational Constitutive Model for Concrete Subjected to Large Strains, High Strain Rates, and High Pressures",

Proceedings 14th International Symposium on Ballistics, Quebec, Canada, pp. 591-600, (1993).

Hopperstad, O.S. and Remseth, S.," A return Mapping Algorithm for a Class of Cyclic Plasticity Models", *International Journal for Numerical Methods in Engineering*, Vol. 38, pp. 549-564, (1995).

Huang, Yuli, private communication, Livermore, (2006).

Hughes, T.J.R. and E. Carnoy, "Nonlinear Finite Element Shell Formulation Accounting for Large Membrane strains," *AMD*-Vol.48, ASME, 193-208 (1981).

Hughes, T.J.R. and W.K. Liu, "Nonlinear Finite Element Analysis of Shells: Part I. Three-Dimensional Shells." *Comp. Meths. Appl. Mechs.*, 27, 331-362 (1981a).

Hughes, T.J.R. and W.K. Liu, "Nonlinear Finite Element Analysis of Shells: Part II. Two-Dimensional Shells." *Comp. Meths. Appl. Mechs.*, 27, 167-181 (1981b).

Hughes, T.J.R., W.K. Liu, and I. Levit, "Nonlinear Dynamics Finite Element Analysis of Shells." Nonlinear Finite Element Analysis in Struct. Mech., Eds. W. Wunderlich, E. Stein, and K.J. Bathe, Springer-Verlag, Berlin, 151- 168 (1981c).

Huh, H. and Kang, W.J., "Crash-Worthiness Assessment of Thin-Walled Structures with the High-Strength Steel Sheet", *Int. Journal of Vehicle Design*, Vol. 30, Nos. 1/2 (2002).

Ibrahimbegovic, A. and Wilson, E.L. "A unified formulation for triangular and quadrilateral flat shell finite elements with six nodal degrees of freedom", *Comm. Applied Num. Meth*, 7, 1-9 (1991).

Isenberg, J., Vaughan, D.K., Sandler, I.S., Nonlinear Soil-Structure Interaction, Electric Power Research Institute report EPRI NP-945, Weidlinger Associates (1978).

Ivanov, I., and A. Tabiei, "Loosely Woven Fabric Model With Viscoelastic Crimped Fibers for Ballistic Impact Simulations", IJNME, 57, (2004).

Johnson, G.C. and D.J. Bammann, "A discussion of stress rates in finite deformation problems," *Int. J. Solids Struct*, 20, 725-737 (1984).

Johnson, G.R. and W.H. Cook, "A Constitutive Model and Data for Metals Subjected to Large Strains, High Strain Rates and High Temperatures." Presented at the Seventh International Symposium on Ballistics, The Hague, The Netherlands, April 1983.

Johnson, G.R. and T.J. Holmquist, "An Improved Computational Model for Brittle Materials" in High-Pressure Science and Technology - 1993 American Institute of Physics Conference Proceedings 309 (c 1994) pp.981-984 ISBN 1-56396-219-5.

Jones, R.M., Mechanics of Composite Materials, Hemisphere Publishing Corporation, New York, (1975).

Kenchington, G.J., "A Non-Linear Elastic Material Model for DYNA3D," Proceedings of the DYNA3D Users Group Conference, published by Boeing Computer Services (Europe) Limited (1988).

Key, S.W. HONDO – A Finite Element Computer Program for the Large Deformation Dynamic Response of Axisymmetric Solids, Sandia National Laboratories, Albuquerque, N.M., Rept. 74-0039 (1974).

Kolling, S., Haufe, A., Feucht, M., DuBois, P. A. "SAMP-1: A Semi-Analytical Model for the Simulation of Polymers", 4. LS-DYNA Anwenderforum, October 20-21, Bamberg, Germany, (2005).

Kolling, S., Hirth, A., Erhart, and Du Bois P.A., Private Communication, Livermore, California (2006).

Krieg, R.D.,A Simple Constitutive Description for Cellular Concrete, Sandia National Laboratories, Albuquerque, NM, Rept. SC-DR-72-0883 (1972).

Krieg, R.D. and S.W. Key, "Implementation of a Time Dependent Plasticity Theory into Structural Computer Programs," Vol. 20 of Constitutive Equations in Viscoplasticity: Computational and Engineering Aspects (American Society of Mechanical Engineers, New York, N.Y., pp. 125-137 (1976).

Lademo, O.G., Berstad, T., Tryland, T., Furu, T., Hopperstad, O.S. and Langseth, M., "*A model for process-based crash simulation*", 8th International LS-DYNA User's Conference, Detroit, May 3-5, 2004.

Lademo, O.G., Hopperstad, O.S., Berstad, T. and Langseth M., "Prediction of Plastic Instability in Extruded Aluminum Alloys Using Shell Analysis and a Coupled Model of Elasto-plasticity and Damage," Journal of Materials Processing Technology, 2002 (Article in Press).

Lademo, O.G., Hopperstad, O.S., Malo, K.A. and Pedersen, K.O., "Modelling of Plastic Anisotropy in Heat-Treated Aluminum Extrusions", Journal of Materials Processing Technology **125-126**, pp. 84-88 (2002).

Lee, E.L. and C.M. Tarver, "Phenomenological Model of Shock Initiation in Heterogenous Explosives," PHYS. Fluids, Vol. 23, p. 2362 (1980).

Lemaitre, J., A Course on Damage Mechanics, Springer-Verlag, (1992).

Lemaitre, J., and Chaboche, J.L., Mechanics of Solid Materials, Cambridge University Press, (1990).

Lemmen, P. P. M. and Meijer, G. J., "Failure Prediction Tool Theory and User Manual," TNO Report 2000-CMC-R0018, (2001).

Lewis, B.A., "Developing and Implementing a Road Side Safety Soil Model into LS-DYNA," FHWA Research and Development Turner-Fairbank Highway Research Center, (1999).

REFERENCES

Li, Y.H. and Sellars, C.M., "Modeling Deformation Behavior of Oxide Scales and their Effects on Interfacial Heat Transfer and Friction during Hot Steel Rolling", Proc. Of the 2nd Int. Conf. Modeling of Metals Rolling Processes, The Insitute of Materials, Londong, UK, 192-201 (1996).

Lian, W., Memo entitled: "LS-DYNA Airbag Module Improvement Request", General Motors Corporation (2000).

MADYMO3D USER'S MANUAL, Version 4.3, TNO Road-Vehicles Research Institute, Department of Injury Prevention, The Hague, The Netherlands, (1990).

Maker, B.N., Private communication Lawrence Livermore National Laboratory, Dr. Maker programmed and implemented the compressible Mooney Rivlin rubber model (1987).

Makris N. and Zhang, J., "Time-domain visco-elastic analysis of earth structures," *Earthquake Engineering and Structural Dynamics*, vol. 29, pp. 745–768, (2000).

Malvar, L.J., Crawford, J.E., Morrill, K.B., K&C Concrete Material Model Release III — Automated Generation of Material Model Input, K&C Technical Report TR-99-24-B1, 18 August 2000 (*Limited Distribution*).

Malvar, L.J., Crawford, J.E., Wesevich, J.W., Simons, D., "A Plasticity Concrete Material Model for DYNA3D," *International Journal of Impact Engineering*, Volume 19, Numbers 9/10, pages 847-873, December 1997.

Malvar, L.J., and Ross, C.A., "Review of Static and Dynamic Properties of Concrete in Tension," *ACI Materials Journal*, Volume 95, Number 6, pages 735-739, November-December 1998.

Malvar, L.J., and Simons,D., "Concrete Material Modeling in Explicit Computations," Proceedings, Workshop on Recent Advances in Computational Structural Dynamics and High Performance Computing, USAE Waterways Experiment Station, Vicksburg, MS, pages 165-194, April 1996. (LSTC may provide this reference upon request.)

Malvar, H.S., Sullivan, G.S., and Wornell, G.W., "Lapped Orthogonal Vector Quantization", in Proc. Data Compression Conference, Snowbird, Utah, 1996.

Marin, E.B., unpublished paper, Sandia National Laboratory, CA (2005).

Matzenmiller, A., Lubliner, J., and Taylor, R.L., "A Constitutive Model for Anisotropic Damage in Fiber-Composites," Mechanics of Materials, Vol. 20, pp. 125-152 (1995).

Matzenmiller, A. and J. K. Schweizerhof, "Crashworthiness Considerations of Composite Structures – A First Step with Explicit Time Integration in Nonlinear Computational Mechanics–State-of-the-Art," Ed. P. Wriggers, W. Wagner, Springer Verlay, (1991).

Mauldin, P.J., R.F. Davidson, and R.J. Henninger, "Implementation and Assessment of the Mechanical-Threshold-Stress Model Using the EPIC2 and PINON Computer Codes," Report LA-11895-MS, Los Alamos National Laboratory (1990).

McCormick, P.G., "Theory of flow localization due to dynamic strain ageing," *Acta Metallurgica*, 36, 3061-3067 (1988).

Mi Y., Crisfield, M.A., Davies, A.O. Progressive delamination using interface elements. *J Compos Mater*, 32(14)1246-72 (1998).

Moran, B., Ortiz, M. and Shih, C.F., "Formulation of implicit finite element methods for multiplicative finite deformation plasticity". *Int J for Num Methods in Engineering*, 29, 483-514 (1990).

de Moura MFSF, Gonçalves, J.P., Marques, A.T., and de Castro, P.T., Elemento finito isoparamétrico de interface para problemas tridimensionais. *Revista Internacional de Métodos Numéricos Para Cálculo e Diseño en Ingeniería*, 14:447-66 (1996).

Murray, Y.D., Users Manual for Transversely Isotropic Wood Model APTEK, Inc., Technical Report to the FHWA (to be published) (2002).

Murray, Y.D. and Lewis, B.A., Numerical Simulation of Damage in Concrete APTEK, Inc., Technical Report DNA-TR-94-190, Contract DNA 001-91-C-0075, Defense Nuclear Agency, Alexandria VA 22310.

Murray, Y.D., Users Manual for LS-DYNA Concrete Material Model 159, Report No. FHWA-HRT-05-062, Federal Highway Administration, (2007).

Murray, Y.D., A. Abu-Odeh, and R. Bligh, Evaluation of Concrete Material Model 159, Report No. FHWA-HRT-05-063, Federal Highway Administration, (2007).

Muscolini, G., Palmeri, A. and Ricciardelli, F., "Time-domain response of linear hysteretic systems to deterministic and random excitations," *Earthquake Engineering and Stuctrual Dynamics*, vol. 34, pp. 1129–1147, (2005).

Nagararaiah, Reinhorn, & Constantinou, "Nonlinear Dynamic Analysis of 3-D Base-Isolated Structures", *Jounal of Structural Engineering* Vol 117, No 7, (1991).

Nahshon, K. and Hutchinson, J.W., "Modification of the Gurson Model for shear failure", *European Journal of Mechanics A/Solids*, Vol. 27, 1-17, (2008).

Neal, M.O., C-H Lin, and J. T. Wang, "Aliasing Effects on Nodal Acceleration Output from Nonlinear Finite Element Simulations," ASME 2000 International Mechanical Engineering Congress and Exposition, Orlando, Florida, November 5-10, (2000).

Neilsen, M.K., H.S. Morgan, and R.D. Krieg, "A Phenomenological Constitutive Model for Low Density Polyurethane Foams," Rept. SAND86-2927, Sandia National Laboratories, Albuquerque, N.M., (1987).

Nusholtz, G., W. Fong, and J. Wu, "Air Bag Wind Blast Phenomena Evaluation," *Experimental Techniques*, Nov.-Dec. (1991).

Nusholtz, G., D. Wang, and E.B. Wylie, "Air Bag Momentum Force Including Aspiration," Preprint, Chrysler Corporation, (1996).

REFERENCES

Nusholz, private communication, (1996).

Ogden, R.W., Non-Linear Elastic Deformations, Ellis Horwood Ltd., Chichester, Great Britian (1984).

Oliver, J., "A Consistent Characteristic Length of Smeared Cracking Models," *International Journal for Numerical Methods in Engineering*, **28**, 461-474 (1989).

Papadrakakis, M., "A Method for the Automatic Evaluation of the Dynamic Relaxation Parameters," *Comp. Meth. Appl. Mech. Eng.*, Vol. 25, pp. 35-48 (1981).

Park, R. and Paulay, T., (1975) Reinforced Concrete Structures, J. Wiley and Sons, New York.

Park, Y.J., Wen, Y.K, and Ang, A.H-S, "Random Vibration of Hysteretic Systems Under Bi-directional Ground Motions", *Earthquake Engineering and Structural Dynamics*, Vol. 14, pp. 543-557 (1986).

Penelis, G.G. and Kappos, A.J., Earthquake-Resistant Concrete Structures, E&FN Spon., (1997).

Pijaudier-Cabot, G., and Bazant, Z.P., "Nonlocal Damage Theory," *Journal of Engineering Mechanics*, ASCE, Vol. 113, No. 10, 1512-1533 (1987).

Porcaro, R., A.G. Hanssen, A. Aalberg and M. Langseth, "The behaviour of aself-piercing riveted connection under quasi-static loading conditions," *Int. J. Solids and Structures*, Vol. 43/17, pp. 5110-5131 (2006).

Porcaro, R., A.G. Hanssen, A. Aalberg and M. Langseth, "Self-piercing riveting process, an experimental and numerical investigation," *Journal of Materials processing Technology*, Vol. 171/1, pp. 10-20 (2006).

Porcaro, R., M. Langseth, A.G. Hanssen, H. Zhao, S. Weyer and H. Hooputra, "Crashworthiness of self-piercing riveted connections," *International Journal of Impact Engineering*, In press, Accepted manuscript (2007).

Puso, M.A., "A Highly Efficient Enhanced Assumed Strain Physically Stabilized Hexahedral Element", *Int. J. Numer. Meth. Eng.*, Vol. 49, 1029-1064 (2000).

Puso, M.A. and Weiss, J.A., "Finite Element Implementation of Anisotropic Quasilinear Viscoelasticity Using a Discrete Spectrum Approximation", *ASME J. Biomech. Engng.*, 120, 62-70 (1998).

Pelessone, D., Private communication, GA Technologies, P.O. Box 85608, San Diego, CA., Telephone No. 619-455-2501 (1986).

Quapp, K.M. and Weiss, J.A., "Material Characterization of Human Medial Collateral Ligament", *ASME J. Biomech Engng.*, 120, 757-763 (1998).

Reyes, A., O.S. Hopperstad, T. Berstad, and M. Langseth, Implementation of a Material Model for Aluminium Foam in LS-DYNA, Report R-01-02, Restricted, Department of Structural Engineering, Norwegian University of Science and Technology, (2002).

Randers-Pehrson, G. and K. A. Bannister, Airblast Loading Model for DYNA2D and DYNA3D, Army Research Laboratory, Rept. ARL-TR-1310, publicly released with unlimited distribution, (1997).

Richards, G.T., Derivation of a Generalized Von Neuman Psuedo-Viscosity with Directional Properties, University of California, Lawrence Livermore National Laboratory, Rept. UCRL-14244 (1965).

Roussis, P.C., and Constantinou, M.C., "Uplift-restraining Friction Pendulum seismic isolation system", *Earthquake Engineering and Structural Dynamics*, 35 (5), 577-593, (2006).

Rupp, A., Grubisic, V., and Buxbaum, O., Ermittlung ertragbarer Beanspruchungen am Schweisspunkt auf Basis der ubertragbaren Schnittgrossen, FAT Schriftenreihe 111, Frankfurt (1994).

Sala, M.O. Neal, and J.T. Wang, Private Communication, General Motors, May, 2004.

Sackett, S.J., "Geological/Concrete Model Development," Private Communication (1987).

Sandler, I.S. and D. Rubin, "An Algorithm and a Modular Subroutine for the Cap Model," *Int. J. Numer. Analy. Meth. Geomech.*, 3, pp. 173-186 (1979).

Schedin, E., Prentzas, L. and Hilding D., "Finite Element Simulation of the TRIP-effect in Austenitic Stainless Steel," presented at SAE 2004, SAE Technical paper 2004-01-0885, (2004).

Schwer, L.E., "A Viscoplastic Augmentation of the Smooth Cap Model," *Nuclear Engineering and Design*, Vol. 150, pp. 215-223, (1994).

Schwer, L.E., "Demonstration of the Continuous Surface Cap Model with Damage: Concrete Unconfined Compression Test Calibration," LS-DYNA Geomaterial Modeling Short Course Notes, July (2001).

Schwer, L.E., W. Cheva, and J.O. Hallquist, "A Simple Viscoelastic Model for Energy Absorbers Used in Vehicle-Barrier Impact," in Computation Aspects of Contact, Impact, and Penetration, Edited by R.F. Kulak and L.E. Schwer, Elmepress International, Lausanne, Switzerland, pp. 99-117 (1991).

Schwer, L.E. and Y.D. Murray, "A Three-Invariant Smooth Cap Model with Mixed Hardening," *International Journal for Numerical and Analytical Methods in Geomechanics*, Volume 18, pp. 657-688, (1994).

Seeger, F., M. Feucht, T. Frank (DaimlerChrysler AG), and A. Haufe, B. Keding (DYNAmore GmbH), "An Investigation on Spotweld Modeling for Crash Simulation with LS-DYNA", 4th LS-DYNA-Forum, Bamburg, Germany, October (2005), ISBN 3-9809901-1-7.

REFERENCES

Sheppard, S.D., Estimations of Fatigue Propagation Life in Resistance Spot Welds, ASTM STP 1211, pp. 169-185, (1993).

Sheppard, T. and Wright, D.S., "Determination of flow stress: Part 1 constitutive equation for aluminum alloys at elevated temperatures", *Metals Technology*, p. 215, June 1979.

Shvets, I.T. and Dyban, E., P., "Contact Heat Transfer between Plane Metal Surfaces", *Int. Chem. Eng.*, Vol. 4, No. 4, 621 (1964).

Simo, J.C., J.W. Ju, K.S. Pister, and R.L. Taylor, "An Assessment of the Cap Model: Consistent Return Algorithms and Rate-Dependent Extension," *J. Eng. Mech.*, Vol. 114, No. 2, 191-218 (1988a).

Simo, J.C., J.W. Ju, K.S. Pister, and R.L. Taylor, "Softening Response, Completeness Condition, and Numerical Algorithms for the Cap Model," *Int. J. Numer. Analy. Meth. Eng.*, (in press) (1988b).

Simo, J. C., J.W. Ju, K.S. Pister, and R.L. Taylor, "Softening Response, Completeness Condition, and Numerical Algorithms for the Cap Model," *Int. J. Numer. Analy. Meth. Eng.* (1990).

Solberg, J.M., and C.M. Noble, "Contact Algorithm for Small-Scale Surface Features with Application to Finite Element Analysis of Concrete Arch Dams with Beveled Contraction Joints", Lawrence Livermore National Laboratory (2002).

Spanos, P.D. and Tsavachidis, S., "Deterministic and stochastic analyses of a nonlinear system with a Biot visco-elastic element," *Earthquake Engineering and Structural Dynamics*, vol. 30, pp. 595–612, (2001).

Steinberg, D.J. and M.W. Guinan, A High-Strain-Rate Constitutive Model for Metals, University of California, Lawrence Livermore National Laboratory, Rept. UCRL-80465 (1978).

Steinberg, D.J. and C.M. Lund, "A Constitutive Model for Strain Rates form 10^{-4} to 10^{6} S^{-1}," *J. Appl. Phys.*, 65, p. 1528 (1989).

Stillman, D.W. and J.O. Hallquist, INGRID: A Three-Dimensional Mesh Generator for Modeling Nonlinear Systems, University of California, Lawrence Livermore National Laboratory, Rept. UCID-20506. (1985).

Stojko, S., privated communication, NNC Limited, Engineering Development Center (1990).

Storakers, B., "On material representation and constitutive branching in finite compressible elasticity", J. Mech. Phy. Solids, 34 No. 2, 125-145 (1986).

Stouffer and Dame, Inelastic Deformation of Metals, Wiley, (1996).

Stout, M.G., D.E. Helling, T.L. Martin, and G.R. Canova, *Int. J. Plasticity*, Vol. 1, pp. 163-174, (1985).

Structural Engineers Association of California, Tentative Lateral Force Requirements, Seismology Committee, SEAOC, 1974, 1990, 1996.

Sussman, T. and Bathe, K.J., "A Finite Element Formulation for Nonlinear Incompressible Elastic and Inelastic Analysis," *Computers & Structures*, **26**, Number 1/2, 357-409 (1987).

Tabiei, A. and I. Ivanov, "Computational micro-mechanical Model of Flexible Woven Fabric for Finite Element Impact Simulation," IJNME, 53, (6), 1259-1276, (2002).

Tahoe User Guide, Sandia National Laboratory, can be downloaded from: www.sandia.gov, Input version 3.4.1, (2003).

Taylor, L.M. and D.P. Flanagan, PRONTO3D A Three-Dimensional Transient Solid Dynamics Program, Sandia Report: SAND87-1912, UC-32, (1989).

Taylor, R.L. Finite element analysis of linear shell problems, *in Whiteman, J.R. (ed.)*, Proceedings of the Mathematics in Finite Elements and Applications, Academic Press, New York, 191-203, (1987).

Taylor, R.L. and Simo, J.C. Bending and membrane elements for the analysis of thick and thin shells, *Proc. of NUMETA Conference*, Swansea (1985).

Tsai, S.W. and E.M. Wu, "A General Theory of Strength for Anisotropic Materials," J. Composite Materials, 5, pp. 73-96 (1971).

Tuler, F.R. and B.M. Butcher, "A Criterion for the Time Dependence of Dynamic Fracture," *The International Journal of Fracture Mechanics*, Vol. 4, No. 4, (1968).

Tvergaard, V. and J.W. Hutchinson, "The relation between crack growth resistance and fracture process parameters in elastic-plastic solids," *J. of the Mech. And Phy. of Solids*, 40, pp1377-1397, (1992)

Tvergaard, V. and Needleman, A., "Analysis of the cup-cone fracture in a round tensile bar", *Acta Metallurgica*, 32, 157-169 (1984).

Vawter, D., "A Finite Element Model for Macroscopic Deformation of the Lung," published in the *Journal of Biomechanical Engineering*, Vol.102, pp. 1-7 (1980).

VDA Richtlinier (Surface Interfaces), Version 20, Verband der Automobilindustrie e.v., Frankfurt, Main, Germany, (1987).

Vegter, H., and Boogaard, A.H. van den, "A plane stress yield function for anisotropic sheet material by interpolation of biaxial stress states", *International Journal of Plasticity* 22, 557-580 (2006).

Walker, J.C., Ratcliffe M.B., Zhang P., Wallace A.W., Fata, B., Hsu E., Saloner D., and Guccione J.M. "MRI-based finite-element analysis of left ventricular aneurysm", Am J Physiol Heart Circ Physiol 289(2): H692:700 (2005).

REFERENCES

Wang, J.T. and O.J. Nefske, “A New CAL3D Airbag Inflation Model,” SAE paper 880654, 1988.

Wang, J.T., "An Analytical Model for an Airbag with a Hybrid Inflator", Publication R&D 8332, General Motors Development Center, Warren, Mi. (1995).

Wang, J.T., "An Analytical Model for an Airbag with a Hybrid Inflator", *AMD*-Vol. 210, *BED*-Vol. 30, ASME, pp 467-497, (1995).

Weiss, J.A., Maker, B.N. and Govindjee, S., "Finite Element Implementation of Incompressible, Transversely Isotropic Hyperelasticity", *Comp. Meth. Appl. Mech. Eng.*, 135, 107-128 (1996).

Wen, T.K. “Method for Random Vibration of Hysteretic Systems”, *J. Engrg. Mech., ASCE*, Vol. 102, No. EM2, Proc. Paper 12073, pp.249-263 (1976).

Whirley, R. G., and J. O. Hallquist, DYNA3D, A Nonlinear, Explicit, Three-Dimensional Finite Element Code for Solid and Structural Mechanics-Users Manual, Report No.UCRL-MA-107254 , Lawrence Livermore National Laboratory, (1991).

Whirley, R. G., and G.A. Henshall, "Creep Deformation Structural Analysis Using An Efficient Numerical Algorithm," *IJNME*, Vol. 35, pp. 1427-1442, (1992).

Wilkins, M.L., “Calculations of Elastic Plastic Flow,” *Meth. Comp. Phys.*, **3**, (Academic Press), 211-263 (1964).

Wilkins, M.L., Calculation of Elastic-Plastic Flow, University of California, Lawrence Livermore National Laboratory, Rept. UCRL-7322, Rev. I (1969).

Wilkins, M.L., The Use of Artificial Viscosity in Multidimensional Fluid Dynamics Calculations, University of California, Lawrence Livermore National Laboratory, Rept. UCRL-78348 (1976)

Wilkins, M.L., R.E. Blum, E. Cronshagen, and P. Grantham, A Method for Computer Simulation of Problems in Solid Mechanics and Gas Dynamics in Three Dimensions and Time, University of California, Lawrence Livermore National Laboratory, Rept. UCRL-51574 (1974).

Wilkins, M.L., J.E. Reaugh, B. Moran, J.K. Scudder, D.F. Quinones, M.E. Prado, Fundamental Study of Crack Initiation and Propagation Annual Progress Report, Report UCRL-52296, Lawrence Livermore National Laboratory, Livermore, CA. (1977).

Wilson, E.L. *Three Dimensional Static and Dynamic Analysis of Structures*, Computers and Structures, Inc., Berkeley CA, (2000).

Winters, J.M., "Hill-based muscle models: A systems engineering perspective," In Multiple Muscle Systems: Biomechanics and Movement Organization, JM Winters and SL-Y Woo eds, Springer-Verlag (1990).

Winters J.M. and Stark L., "Estimated mechanical properties of synergistic muscles involved in movements of a variety of human joints," *J Biomechanics* 21:1027-1042, (1988).

Woodruff, J.P., KOVEC User's Manual, University of California, Lawrence Livermore National Laboratory, Report UCRL-51079, (1973).

Worswick, M.J., and Xavier Lalbin, Private communication, Livermore, California, (1999).

Yamasaki, H., M. Ogura, R. Nishimura, and K. Nakamura, "Development of Material Model for Crack Propagation of Casting Aluminum", Presented at the 2006 JSAE Annual Congress, Paper Number 20065077, (2006).

Yen, C.F., "Ballistic Impact Modeling of Composite Materials," Proceedings of the 7th International LS-DYNA Users Conference, Dearborn, MI, May 19-21, 2002, 6.15-6.25.

Yoshida, F. and Uemori, T., "A Model of Large-Strain Cyclic Plasticity and its Application to Springback Simulation," *International Journal of Mechanical Sciences*, Vol. 45, 1687-1702, (2003).

Zajac F.E., "Muscle and tendon: Properties, models, scaling, and application to biomechanics and motor control", *CRC Critical Reviews in Biomedical Engineering* 17(4):359-411, (1989).

Zayas, V.A., Low, S.S. and Mahin, S.A., "A Simple Pendulum Technique for Achieving Seismic Isolation", *J. Earthquake Spectra*, Vol. 6, No. 2, pp. 317-334 (1990).

Zhang, S., Approximate Stress Intensity Factors and Notch Stresses for Common Spot-Welded Specimens, Welding Research Supplement, pp. 173s-179s, (1999).

Zhang, S., McCormick, P.G., Estrin, Y., "The morphology of Portevin-Le Chatelier bands: Finite element simulation for Al-Mg-Si", *Acta Materialia* 49, 1087-1094, (2001).

APPENDIX A: User Defined Materials

The user can supply his/her own subroutines defining material models in LS-DYNA. To invoke a user-defined material, one must

1. Write a user material subroutine that is called by the LS-DYNA user material interface.
2. Create a custom executable which includes the material subroutine.
3. Invoke that subroutine by defining a part in the keyword input deck that uses `*MAT_USER_DEFINED_MATERIAL_MODELS` with appropriate input parameters.

All subroutines, including interface, for the user-defined materials are collected in the file `dyn21.F` (Unix/Linux) or `lsdyna.f` (Windows). Up to ten user subroutines can currently be implemented simultaneously to update the stresses in solids, shells, beams, discrete beams and truss beams. This text serves as an introductory guide to implement such a model. Note that names of variables and subroutines below may differ from the actual ones depending on platform and current version of LS-DYNA.

General overview

When the keyword `*MAT_USER_DEFINED_MATERIAL_MODELS` is defined for a part in the keyword deck, LS-DYNA calls the subroutine `usrmat` with appropriate input data for the constitutive update. This routine in turn calls `urmathn` for 2D and 3D solid elements, `urmats` for 2D plane stress and 3D shell elements, `urmatb` for beam elements, `urmatd` for discrete beam elements and `urmatt` for truss beam elements. In these routines, which may be modified by the user if necessary, the following data structures are initialized for the purpose of being supplied to a specific *scalar* material subroutine.

`sig(6)` – stresses in previous time step
`eps(6)` – strain increments
`epsp` – effective plastic strain in previous time step
`hsv(*)` – history variables in previous time step excluding plastic strain
`dt1` – current time step size
`temper` – current temperature
`failel` – flag indicating failure of element

If the *vectorization* flag is active (`IVECT=1`) on the material card, variables are in general stored in vector blocks of length `nlq`, with vector indexes ranging from `lft` to `llt` , which allows for a more efficient execution of the material routine. As an example, the data structures mentioned above are for the vectorized case exchanged for

`sigX(nlq)` – stresses in previous time step
`dX(nlq)` – strain increments
`epsps(nlq)` – effective plastic strains in previous time step
`hsvs(nlq,*)` – history variables in previous time step
`dt1siz(nlq)` - current time step sizes
`temps(nlq)` – current temperatures
`failels(nlq)` – flags indicating failure of elements

where *x* ranges from 1 to 6 for the different components. Each entry in a vector block is associated with an integration point in the finite element mesh.

The number of entries in the history variables array (indicated by * in the above) matches the number of history variables requested on the material card (NHV). Hence the number NHV should equal to the number of history variables excluding the effective plastic strain since this variable is given a special treatment. All history variables, including the effective plastic strain, are initially zero. Furthermore, all user-defined material models require a bulk modulus and shear modulus for transmitting boundaries, contact interfaces, rigid body constraints, and time step calculations. This generally means that the length of material constants array LMC must be increased by 2 for the storage of these parameters. In addition to the variables mentioned above, the following data can be supplied to the user material routines, regardless of whether vectorization is used or not.

cm(*) – material constants array
capa – transverse shear correction factor for shell elements
tt – current time
crv(101,2,*) – array representation of curves defined in the keyword deck

A specific material routine, umat*XX* in the scalar case or umat*XX*v in the vector case, is now called with any necessary parameters of the ones above, and possibly others as well. The letters *XX* stands for a number between 41 and 50 and matches the number MT on the material card. This subroutine is written by the user, and should update the stresses and history variables to the current time. For shells and beams it is also necessary to determine the strain increments in the directions of constrained zero stress. To be able to write different stress updates for different elements, the following character string is passed to the user-defined subroutine

etype – character string that equals solid, shell, beam, dbeam or tbeam

A sample user subroutine of a hypo-elastic material in the scalar case is provided below.

Sample user subroutine 41

```
      subroutine umat41 (cm,eps,sig,epsp,hsv,dt1,capa,etype,
     .    tt,temper,failel,crv)
c*************************************************************
c|  livermore software technology corporation  (lstc)         |
c|  ------------------------------------------------------    |
c|  copyright 1987-2003                                       |
c|  all rights reserved                                       |
c*************************************************************
c
c     Isotropic elastic material (sample user subroutine)
c
c     Variables
c
c     cm(1)=first material constant, here young's modulus
c     cm(2)=second material constant, here poisson's ratio
c        .
c        .
c        .
c     cm(n)=nth material constant
c
c     eps(1)=local x  strain increment
c     eps(2)=local y  strain increment
c     eps(3)=local z  strain increment
```

```
c     eps(4)=local xy strain increment
c     eps(5)=local yz strain increment
c     eps(6)=local zx strain increment
c
c     sig(1)=local x  stress
c     sig(2)=local y  stress
c     sig(3)=local z  stress
c     sig(4)=local xy stress
c     sig(5)=local yz stress
c     sig(6)=local zx stress
c
c     epsp=effective plastic strain
c
c     hsv(1)=1st history variable
c     hsv(2)=2nd history variable
c        .
c        .
c        .
c        .
c     hsv(n)=nth history variable
c
c     capa=reduction factor for transverse shear
c
c     etype:
c       eq."brick" for solid elements
c       eq."shell" for all shell elements
c       eq."beam"  for all beam elements
c       eq."dbeam" for all discrete beam elements
c
c     temper=current temperature
c
c     dt1=current time step size
c
c     tt=current problem time.
c
c     crv=array representation of curves defined in keyword deck
c
c     failel=flag for failure, set to .true. to fail an element
c
c     All transformations into the element local system are
c     performed prior to entering this subroutine.  Transformations
c     back to the global system are performed after exiting this
c     routine.
c
c     All history variables are initialized to zero in the input
c     phase.   Initialization of history variables to nonzero values
c     may be done during the first call to this subroutine for each
c     element.
c
c     Energy calculations for the energy balance are done
c     outside this subroutine.
c
c
      character*(*) etype
      dimension  cm(*),eps(*),sig(*),hsv(*),crv(101,2,*)
      logical failel
c
c     compute shear modulus, g
      g2 =cm(1)/(1.+cm(2))
      g  =.5*g2
c
      if (etype.eq.'brick') then
c
```

```
        davg=(-eps(1)-eps(2)-eps(3))/3.
        p=-davg*cm(1)/((1.-2.*cm(2)))
        sig(1)=sig(1)+p+g2*(eps(1)+davg)
        sig(2)=sig(2)+p+g2*(eps(2)+davg)
        sig(3)=sig(3)+p+g2*(eps(3)+davg)
        sig(4)=sig(4)+g*eps(4)
        sig(5)=sig(5)+g*eps(5)
        sig(6)=sig(6)+g*eps(6)
c
      elseif (etype.eq.'shell') then
c
        gc    =capa*g
        q1    =cm(1)*cm(2)/((1.0+cm(2))*(1.0-2.0*cm(2)))
        q3    =1./(q1+g2)
        eps(3)=-q1*(eps(1)+eps(2))*q3
        davg  =(-eps(1)-eps(2)-eps(3))/3.
        p     =-davg*cm(1)/((1.-2.*cm(2)))
        sig(1)=sig(1)+p+g2*(eps(1)+davg)
        sig(2)=sig(2)+p+g2*(eps(2)+davg)
        sig(3)=0.0
        sig(4)=sig(4)+g *eps(4)
        sig(5)=sig(5)+gc*eps(5)
        sig(6)=sig(6)+gc*eps(6)
c
      elseif (etype.eq.'beam' ) then
c
        q1    =cm(1)*cm(2)/((1.0+cm(2))*(1.0-2.0*cm(2)))
        q3    =q1+2.0*g
        gc    =capa*g
        deti  =1./(q3*q3-q1*q1)
        c22i  = q3*deti
        c23i  =-q1*deti
        fac   =(c22i+c23i)*q1
        eps(2)=-eps(1)*fac-sig(2)*c22i-sig(3)*c23i
        eps(3)=-eps(1)*fac-sig(2)*c23i-sig(3)*c22i
        davg  =(-eps(1)-eps(2)-eps(3))/3.
        p     =-davg*cm(1)/((1.-2.*cm(2)))
        sig(1)=sig(1)+p+g2*(eps(1)+davg)
        sig(2)=0.0
        sig(3)=0.0
        sig(4)=sig(4)+gc*eps(4)
        sig(5)=0.0
        sig(6)=sig(6)+gc*eps(6)
      elseif (etype.eq.'tbeam' ) then
c
        q1    =cm(1)*cm(2)/((1.0+cm(2))*(1.0-2.0*cm(2)))
        q3    =q1+2.0*g
        deti  =1./(q3*q3-q1*q1)
        c22i  = q3*deti
        c23i  =-q1*deti
        fac   =(c22i+c23i)*q1
        eps(2)=-eps(1)*fac
        eps(3)=-eps(1)*fac
        davg  =(-eps(1)-eps(2)-eps(3))/3.
        p     =-davg*cm(1)/((1.-2.*cm(2)))
        sig(1)=sig(1)+p+g2*(eps(1)+davg)
        sig(2)=0.0
        sig(3)=0.0
      endif
c
      return
      end
```

Additional features

Load curves and tables

If the material of interest should require load curves, for instance a curve defining yield stress as a function of effective plastic strain, the variable `crv` should be used. Each curve defined in the keyword deck is represented by points (x_i, y_i), $i = 1,...,100$, stored in the array `crv` together with a number defining the increments Δx stored in position 101. To be more precise, the first x value is stored in `crv(1,1,*)`, the first y value in `crv(1,2,*)`, the second x value in `crv(2,1,*)`, the second y value in `crv(2,2,*)`, and so on. The increment Δx is stored in `crv(101,1,*)`. The third index in the `crv` array represents the internal load curve id. There are two ways to extract the values from a load curve from a user defined materials routine.

First, there are two subroutines that can be called from within the user defined routine, these are

```
      subroutine crvval(crv,eid,xval,yval,slope)
```

and

```
      subroutine crvval_v(crv,eid,xval,yval,slope,lft,llt)
```

where the former routine is used in the scalar context and the latter for vectorized umat. The arguments are the following

`crv`	-	the load curve array
`eid`	-	external load curve ID, i.e., the load curve ID taken from the keyword deck
`xval`	-	abscissa value
`yval`	-	ordinate value (output from routine)
`slope`	-	slope of curve (output from routine)
`lft`	-	first index of vector
`llt`	-	final index of vector

where `xval`, `yval` and `slope` are scalars in the scalar routine and vectors of length `nlq` in the vectorized routine. Note that `eid` should be passed as float.

Second, for efficiency considerations the user may extract values on his/her own. The following few lines of code shows how to extract the ordinate value `y` at the abscissa `x` for a curve with external curve id (in the keyword deck) given by `crvid_ext`.

```
      integer crvid_int
c
c     obtain internal curve id
c
      crvid_int=lcids(nint(crvid_ext))
c
c     proceed if curve id is valid
c
      if (crvid_int.gt.0) then
c
c     obtain increment in x and first x value
c
         xinc=crv(101,1,crvid_int)
```

```
         xbgn=crv(1,1,crvid_int)
c
c     find interval in which x is situated
c
         ind=aint((x-xbgn)/xinc)+1
         ind=min(ind,99)
         ind=max(ind,1)
c
c     find slope of that particular segment
c
         slope=(crv(ind+1,2,crvid_int)-crv(ind,2,crvid_int))/
     1         (crv(ind+1,1,crvid_int)-crv(ind,1,crvid_int))
c
c     evaluate ordinate value y
c
         y=crv(ind,2,crvid_int)+slope*(x-crv(ind,1,crvid_int))
c
      endif
```

For tables, two subroutines are available for extracting values. A scalar version is

```
subroutine tabval(crv,eid,dxval,yval,dslope,xval,slope)
```

and a vector version is

```
subroutine tabval_v(crv,eid,dxval,yval,dslope,lft,llt,xval,slope)
```

where

crv - curve array
eid - external curve id, i.e., curve id taken from keyword deck
dxval - abscissa value (x2-axis)
yval - ordinate value (y-axis, output from routine)
dslope- slope of curve (dy/dx2, output from routine)
xval - abscissa value (x1-axis)
slope - slope of curve (dy/dx1, output from routine)
lft - vector index
llt - vector index

In the scalar routine, dxval, yval, dslope, xval and slope are all scalars whereas in the vector routine they are vectors of length nlq.

Local coordinate system

If the material model has directional properties, such as composites and anisotropic plasticity models, the local coordinate system option can be invoked. This is done by putting `IORTHO` equal to `1` on the material card. This also requires two additional cards with values for how the coordinate system is formed and updated. When this option is used, all data passed to the constitutive routine `umat`*XX* or `umat`*XX*`v` is in the local system and the transformation back to the global system is done outside this user-defined routine. There is one exception however, see the section on the deformation gradient.

Temperature
For a material with thermal properties, temperatures are made available by putting the flag `ITHERMAL` equal to `1` on the material card. The temperatures in the elements are then available in the `temper` variable for a scalar and `temps` array for the vectorized implementation. For a coupled thermal structural analysis, the thermal problem is solved first and temperatures at the current time are available in the user-defined subroutine. Calculation of dissipated heat in the presence of plastic deformation is taken care of by LS-DYNA and needs not be considered by the user. If the time derivative of the temperature is needed for the stress update, a history variable that contains the temperature in the previous time step should be requested. The time derivative can then be obtained by a backward finite difference estimate.

Failure
It is possible to include failure in the material model, resulting in the deletion of elements that fulfill a certain failure criterion. To accomplish this, the flag `IFAIL` must be set to `1` on the material card. For a scalar implementation, the variable `failel` is set to `.true.` when a failure criterion is met. For a vectorized implementation, the corresponding entry in the `failels` array is set to `.true.`

Deformation gradient
For some materials, the stresses are not obtained from incremental strains, but are expressed in terms of the deformation gradient $\mathbf{F}$. This is the case for hyper-elastic(-plastic) materials. To make the deformation gradient available for bricks and shells in the user-defined material subroutines, the variable `IHYPER` on the material card should be set to `1`. The deformation gradient components F_{11}, F_{21}, F_{31}, F_{12}, F_{22}, F_{32}, F_{13}, F_{23} and F_{33} can then be found in the history variables array in positions `NHV+1` to `NHV+9`, i.e., the positions coming right after the requested number of history variables.

For shell elements, the components of the deformation gradient are with respect to the co-rotational system for the element currently used. In this case the third row of the deformation gradient, i.e., the components F_{31}, F_{32} and F_{33}, will not be properly updated when entering the user-defined material routine. These components depend on the thickness strain increment which in turn must be determined so that the normal stress in the shell vanishes. For a given thickness strain increment `d3`, these three components, `f31`, `f32` and `f33`, can be determined by calling the subroutine

```
subroutine compute_f3s(f31,f32,f33,d3)
```

for a scalar implementation and

```
subroutine compute_f3(f31,f32,f33,d3,lft,llt)
```

for a vector implementation. The first four arguments are arrays of length `nlq` for the vector routine and scalars for the scalar routine.

For hyper-elastic materials there are push forward operations that can be called from within the user defined subroutines. These are

```
subroutine push_forward_2(sig1,sig2,sig3,sig4,sig5,sig6,
      f11,f21,f31,f12,f22,f32,f13,f23,f33,lft,llt)
```

which performs a push forward operation on the stress tensor, and the corresponding scalar routine

```
subroutine push_forward_2s(sig1,sig2,sig3,sig4,sig5,sig6,
      f11,f21,f31,f12,f22,f32,f13,f23,f33)
```

In the latter subroutine all arguments are scalars whereas the corresponding entries in the vectorized routine are vectors of length `nlq`. The `sig1` to `sig6` are components of the stress tensor and `f11` to `f33` are components of the deformation gradient.

If the local coordinate system option is invoked (`IORTHO=1`), then the deformation gradient is transformed to this local system prior to entering the user-defined material routine according to

$$\bar{F}_{ij} = Q^s_{ki} F_{kj}$$

where Q^s_{ij} refers to a transformation between the current global and material frames. For `IORTHO` equal to `1` one can choose to put `IHYPER` equal to `-1` which results in that the deformation gradient is transformed according to

$$\bar{F}_{ij} = F_{ik} Q^r_{kj}$$

where Q^r_{ij} is the transformation between the reference global and material and frames. For this latter option the spatial frame remains the global one so the stresses should be expressed in this frame of reference upon exiting the user defined routines. The suitable choice of `IHYPER` depends on the formulation of the material model.

In the following, a Neo-Hookean material is used as an example of the usage of the deformation gradient in user-defined materials. With λ and μ being the Lame parameters in the linearized theory, the strain energy density for this material is given by

$$\psi = \frac{1}{2}\lambda(\ln(\det \mathbf{F}))^2 - \mu \ln(\det \mathbf{F}) + \frac{1}{2}\mu(\mathrm{tr}(\mathbf{F}^T\mathbf{F}) - 3)$$

meaning that the Cauchy stress can be expressed as

$$\boldsymbol{\sigma} = \frac{1}{\det \mathbf{F}}(\lambda \ln(\det \mathbf{F})\mathbf{I} + \mu(\mathbf{F}\mathbf{F}^T - \mathbf{I})).$$

Sample user subroutine 42

```
      subroutine umat42 (cm,eps,sig,epsp,hsv,dt1,capa,
     .     etype,tt,temper,failel,crv)
c************************************************************
c|  livermore software technology corporation  (lstc)       |
c|  ------------------------------------------------------  |
c|  copyright 1987-2003                                     |
c|  all rights reserved                                     |
c************************************************************
c
c     Neo-Hookean material (sample user subroutine)
c
c     Variables
c
c     cm(1)=first material constant, here young's modulus
c     cm(2)=second material constant, here poisson's ratio
c        .
c        .
c        .
c     cm(n)=nth material constant
c
c     eps(1)=local x  strain increment
c     eps(2)=local y  strain increment
c     eps(3)=local z  strain increment
c     eps(4)=local xy strain increment
c     eps(5)=local yz strain increment
c     eps(6)=local zx strain increment
c
c     sig(1)=local x  stress
c     sig(2)=local y  stress
c     sig(3)=local z  stress
c     sig(4)=local xy stress
c     sig(5)=local yz stress
c     sig(6)=local zx stress
c
c     epsp=effective plastic strain
c
c     hsv(1)=1st history variable
c     hsv(2)=2nd history variable
c        .
c        .
c        .
c        .
c     hsv(n)=nth history variable
c
c     dt1=current time step size
c     capa=reduction factor for transverse shear
c     etype:
c        eq."brick" for solid elements
c        eq."shell" for all shell elements
c        eq."beam"  for all beam elements
c        eq."dbeam" for all discrete beam elements
c
c     tt=current problem time.
c     temper=current temperature
c
c     crv=array representation of curves defined in keyword deck
c
c     failel=flag for failure, set to .true. to fail an element
c
c     All transformations into the element local system are
```

```
c     performed prior to entering this subroutine.  Transformations
c     back to the global system are performed after exiting this
c     routine.
c
c     All history variables are initialized to zero in the input
c     phase.   Initialization of history variables to nonzero values
c     may be done during the first call to this subroutine for each
c     element.
c
c     Energy calculations for the energy balance are done
c     outside this subroutine.
c
      character*(*) etype
      dimension cm(*),eps(*),sig(*),hsv(*),crv(101,2,*)
      logical failel
c
c     compute lame parameters
c
      xlambda=cm(1)*cm(2)/((1.+cm(2))*(1.-2.*cm(2)))
      xmu=.5*cm(1)/(1.+cm(2))
c
      if (etype.eq.'brick') then
c
c     no history variables, NHV=0
c     deformation gradient stored in hsv(1),...,hsv(9)
c
c     compute jacobian
c
         detf=hsv(1)*(hsv(5)*hsv(9)-hsv(6)*hsv(8))
     1        -hsv(2)*(hsv(4)*hsv(9)-hsv(6)*hsv(7))
     2        +hsv(3)*(hsv(4)*hsv(8)-hsv(5)*hsv(7))
c
c     compute left cauchy-green tensor
c
         b1=hsv(1)*hsv(1)+hsv(4)*hsv(4)+hsv(7)*hsv(7)
         b2=hsv(2)*hsv(2)+hsv(5)*hsv(5)+hsv(8)*hsv(8)
         b3=hsv(3)*hsv(3)+hsv(6)*hsv(6)+hsv(9)*hsv(9)
         b4=hsv(1)*hsv(2)+hsv(4)*hsv(5)+hsv(7)*hsv(8)
         b5=hsv(2)*hsv(3)+hsv(5)*hsv(6)+hsv(8)*hsv(9)
         b6=hsv(1)*hsv(3)+hsv(4)*hsv(6)+hsv(7)*hsv(9)
c
c     compute cauchy stress
c
         detfinv=1./detf
         dmu=xmu-xlambda*log(detf)
         sig(1)=detfinv*(xmu*b1-dmu)
         sig(2)=detfinv*(xmu*b2-dmu)
         sig(3)=detfinv*(xmu*b3-dmu)
         sig(4)=detfinv*xmu*b4
         sig(5)=detfinv*xmu*b5
         sig(6)=detfinv*xmu*b6
c
      else if (etype.eq.'shell') then
c
c     no history variables, NHV=0
c     deformation gradient stored in hsv(1),...,hsv(9)
c
c     compute part of left cauchy-green tensor
c     independent of thickness strain increment
c
         b1=hsv(1)*hsv(1)+hsv(4)*hsv(4)+hsv(7)*hsv(7)
         b2=hsv(2)*hsv(2)+hsv(5)*hsv(5)+hsv(8)*hsv(8)
         b4=hsv(1)*hsv(2)+hsv(4)*hsv(5)+hsv(7)*hsv(8)
```

```
c
c     secant iterations for zero normal stress
c
         do iter=1,5
c
            if (iter.eq.1) then
c
c     first thickness strain increment initial guess
c     assuming Poisson's ratio different from zero
c
               eps(3)=-xlambda*(eps(1)+eps(2))/(xlambda+2.*xmu)
c
            else if (iter.eq.2) then
c
c     second thickness strain increment initial guess
c
               sigold=sig(3)
               epsold=eps(3)
               eps(3)=0.
c
            else if (abs(sig(3)-sigold).gt.0.0) then
c
c     secant update of thickness strain increment
c
               deps=-(eps(3)-epsold)/(sig(3)-sigold)*sig(3)
               sigold=sig(3)
               epsold=eps(3)
               eps(3)=eps(3)+deps
c
            endif
c
c     compute last row of deformation gradient
c
            call compute_f3s(hsv(3),hsv(6),hsv(9),eps(3))
c
c     compute jacobian
c
            detf=hsv(1)*(hsv(5)*hsv(9)-hsv(6)*hsv(8))
     1           -hsv(2)*(hsv(4)*hsv(9)-hsv(6)*hsv(7))
     2           +hsv(3)*(hsv(4)*hsv(8)-hsv(5)*hsv(7))
c
c     compute normal component of left cauchy-green tensor
c
            b3=hsv(3)*hsv(3)+hsv(6)*hsv(6)+hsv(9)*hsv(9)
c
c     compute normal stress
c
            detfinv=1./detf
            dmu=xmu-xlambda*log(detf)
            sig(1)=detfinv*(xmu*b1-dmu)
            sig(2)=detfinv*(xmu*b2-dmu)
            sig(3)=detfinv*(xmu*b3-dmu)
            sig(4)=detfinv*xmu*b4
c
c     exit loop if normal stress is sufficiently small
c
            if (abs(sig(3)).le.1.e-4*
     1           (abs(sig(1))+abs(sig(2))+abs(sig(4)))) goto 10
c
         enddo
c
c     compute remaining components of left cauchy-green tensor
c
```

```
 10      b5=hsv(2)*hsv(3)+hsv(5)*hsv(6)+hsv(8)*hsv(9)
         b6=hsv(1)*hsv(3)+hsv(4)*hsv(6)+hsv(7)*hsv(9)
c
c     compute remaining stress components
c
         sig(5)=detfinv*xmu*b5
         sig(6)=detfinv*xmu*b6
c
      else
c
c     material model only available for bricks and shells
c
         write ( *,20) etype
         write (13,20) etype
         write (59,20) etype
         call adios(2)
c
      endif
c
 20   format(/
     1     '***  error element type ',a,' can not be'
     2     'run with the current material model. ***')
c
      return
      end
```

Implicit analysis

For brick and shell elements, a user-defined material model can also be run with implicit analysis. When an implicit analysis is requested in the input keyword deck, LS-DYNA calls the subroutine `urtanh` for bricks and `urtans` for shells with appropriate input data for the calculation of the material tangent modulus. For a scalar implementation, this routine in turn calls `utan`*XX* with all necessary input parameters including

`es(6,6)` – material tangent modulus

Again, *XX* is the number that matches `MT` on the material card. For a vectorized implementation, the routine `utan`*XX*`v` is called, this time with the corresponding vector block

`dsave(nlq,6,6)` – material tangent modulus

This subroutine builds the tangent modulus to be used for assembling the tangent stiffness matrix and must be provided by the user. This matrix is equal to the zero matrix when entering the user-defined routine, it must be symmetric and if the local coordinate system option is invoked for bricks, then it should be expressed in this local system. For shell elements, it should be expressed in the co-rotational system defined for the current shell element. All transformations back to the global system are made after exiting the user-defined routine.
If the material is hyper-elastic, there are push forward operations of tangent modulus tensor available in

```
      subroutine push_forward_4(dsave,
     .      f11,f21,f31,f12,f22,f32,f13,f23,f33,lft,llt)
```

which performs a push forward operation on the tangent modulus tensor, and the corresponding scalar routine

```
      subroutine push_forward_4s(es,
     .      f11,f21,f31,f12,f22,f32,f13,f23,f33)
```

In the latter subroutine all arguments are scalars whereas the corresponding entries in the vectorized routine are vectors of length `nlq`. The `f11` to `f33` are components of the deformation gradient.

The following sample user subroutine illustrates how to implement the tangent stiffness modulus for the Neo-Hookean material above. The material tangent modulus is for this material given by

$$\mathbf{C} = \frac{1}{\det \mathbf{F}}(\lambda \mathbf{I} \otimes \mathbf{I} + 2(\mu - \lambda \ln(\det \mathbf{F}))\mathbf{I}) .$$

Sample user subroutine 42, tangent modulus

```
      subroutine utan42(cm,eps,sig,epsp,hsv,dt1,capa,
     .     etype,tt,temper,es,crv)
c*****************************************************************
c|  livermore software technology corporation  (lstc)             |
c|  ---------------------------------------------------------  |
c|  copyright 1987-1999                                          |
c|  all rights reserved                                          |
c*****************************************************************
c
c     Neo-Hookean material tangent modulus (sample user subroutine)
c
c     Variables
c
c     cm(1)=first material constant, here young's modulus
c     cm(2)=second material constant, here poisson's ratio
c        .
c        .
c        .
c     cm(n)=nth material constant
c
c     eps(1)=local x  strain increment
c     eps(2)=local y  strain increment
c     eps(3)=local z  strain increment
c     eps(4)=local xy strain increment
c     eps(5)=local yz strain increment
c     eps(6)=local zx strain increment
c
c     sig(1)=local x  stress
c     sig(2)=local y  stress
c     sig(3)=local z  stress
c     sig(4)=local xy stress
c     sig(5)=local yz stress
c     sig(6)=local zx stress
c
c     epsp=effective plastic strain
c
c     hsv(1)=1st history variable
c     hsv(2)=2nd history variable
c        .
c        .
c        .
```

```
c        .
c     hsv(n)=nth history variable
c
c     dt1=current time step size
c     capa=reduction factor for transverse shear
c     etype:
c        eq."brick" for solid elements
c        eq."shell" for all shell elements
c        eq."beam"  for all beam elements
c        eq."dbeam"  for all discrete beam elements
c
c     tt=current problem time.
c
c     temper=current temperature
c
c     es=material tangent modulus
c
c     crv=array representation of curves in keyword deck
c
c     The material tangent modulus is set to 0 prior to entering
c     this routine. It should be expressed in the local system
c     upon exiting this routine. All transformations back to the
c     global system is made outside this routine.
c
      character*(*) etype
      dimension cm(*),eps(*),sig(*),hsv(*),crv(101,2,*)
      dimension es(6,*)
c
c     no history variables, NHV=0
c     deformation gradient stored in hsv(1),...,hsv(9)
c
c     compute jacobian
c
      detf=hsv(1)*(hsv(5)*hsv(9)-hsv(6)*hsv(8))
     1     -hsv(2)*(hsv(4)*hsv(9)-hsv(6)*hsv(7))
     2     +hsv(3)*(hsv(4)*hsv(8)-hsv(5)*hsv(7))
c
c     compute lame parameters
c
      xlambda=cm(1)*cm(2)/((1.+cm(2))*(1.-2.*cm(2)))
      xmu=.5*cm(1)/(1.+cm(2))
c
c     compute tangent stiffness
c     same for both shells and bricks
c
      detfinv=1./detf
      dmu=xmu-xlambda*log(detf)
      es(1,1)=detfinv*(xlambda+2.*dmu)
      es(2,2)=detfinv*(xlambda+2.*dmu)
      es(3,3)=detfinv*(xlambda+2.*dmu)
      es(4,4)=detfinv*dmu
      es(5,5)=detfinv*dmu
      es(6,6)=detfinv*dmu
      es(2,1)=detfinv*xlambda
      es(3,2)=detfinv*xlambda
      es(3,1)=detfinv*xlambda
      es(1,2)=es(2,1)
      es(2,3)=es(3,2)
      es(1,3)=es(3,1)
c
      return
      end
```

User-Defined Materials with Equations of State

The following example umat44v is set up to be used with an equation of state (EOS). Unlike standard models, it updates only the deviatoric stress and it assigns a value to *PC*, the pressure cut-off. The pressure cut-off limits the amount of hydrostatic pressure that can be carried in tension (i.e., when the pressure is negative). The default value is zero, and a large negative number will allow the material to carry an unlimited pressure load in tension. It is calculated within the material model because it is typically a function of the current state of the material and varies with time. In this example, however, it is a constant value for simplicity. The pressure cut-off array is passed through the named common block `eosdloc`. Depending on the computing environment, compiler directives may be required (e.g., the task common directive in the example) for correct SMP execution.

In addition, the number of history variables, NHV, must be increased by 4 in the input file to allocate the extra storage required for the EOS. The storage is the last 4 variables in `hsvs`, and it must not be altered by the user-defined material model.

```
      subroutine umat44v(cm,d1,d2,d3,d4,d5,d6,sig1,sig2,
     . sig3,sig4,sig5,sig6,eps,hsvs,lft,llt,dt1siz,capa,
     . etype,tt,temps,failels,nlqa,crv)
      parameter (third=1.0/3.0)
      include 'nlqparm'
c
c***  isotropic plasticity with linear hardening
c
c***  updates only the deviatoric stress so that it can be used with
c     an equation of state
c
      character*5 etype
      logical failels
c
C_TASKCOMMON (eosdloc)
      common/eosdloc/pc(nlq)
c
      dimension cm(*),d1(*),d2(*),d3(*),d4(*),d5(*),d6(*),
     & sig1(*),sig2(*),sig3(*),sig4(*),sig5(*),sig6(*),
     & eps(*),hsvs(nlqa,*),dt1siz(*),temps(*),crv(lq1,2,*),
     & failels(*)
c
c***  shear modulus, initial yield stress, hardening, and pressure cut-off
      g   =cm(1)
      sy0 =cm(2)
      h   =cm(3)
      pcut=cm(4)
c
      ofac=1.0/(3.0*g+h)
      twog=2.0*g
c
      do i=lft,llt
c
c***    trial elastic deviatoric stress
        davg=third*(d1(i)+d2(i)+d3(i))
        savg=third*(sig1(i)+sig2(i)+sig3(i))
        sig1(i)=sig1(i)-savg+twog*(d1(i)-davg)
        sig2(i)=sig2(i)-savg+twog*(d2(i)-davg)
        sig3(i)=sig3(i)-savg+twog*(d3(i)-davg)
        sig4(i)=sig4(i)+g*d4(i)
        sig5(i)=sig5(i)+g*d5(i)
```

```
        sig6(i)=sig6(i)+g*d6(i)
c
c***    radial return
        aj2=sqrt(1.5*(sig1(i)**2+sig2(i)**2+sig3(i)**2)+
     &           3.0*(sig4(i)**2+sig5(i)**2+sig6(i)**2))
        sy=sy0+h*eps(i)
        eps(i)=eps(i)+ofac*max(0.0,aj2-sy)
        synew=sy0+h*eps(i)
        scale=synew/max(synew,aj2)
c
c***    scaling for radial return. note that the stress is now deviatoric.
        sig1(i)=scale*sig1(i)
        sig2(i)=scale*sig2(i)
        sig3(i)=scale*sig3(i)
        sig4(i)=scale*sig4(i)
        sig5(i)=scale*sig5(i)
        sig6(i)=scale*sig6(i)
c
c***    set pressure cut-off
        pc(i)=pcut
c
      enddo
c
      return
      end
```

Post-processing a user-defined material

Post-processing a user-defined material is very similar to post-processing a regular LS-DYNA material. There are however some things that are worth being stressed, all dealing with how to post-process history variables.

First, the effective plastic strain is always written to the d3plot database and thus need not be requested by the user. It is in LS-PRE/POST treated just as it is for any other LS-DYNA material.

The number of additional history variables written to the d3plot database must be requested as the parameter NEIPH (for bricks) or NEIPS (for shells) on *DATABASE_EXTENT_BINARY. For instance, if NEIPH (NEIPS) equals 2 the first two history variables in the history variables array are obtained as history var#1 and history var#2 in the d3plot database. By putting NEIPH (NEIPS) equal to NHV, all history variables are written to the d3plot database. Furthermore, if the material uses the deformation gradient (IHYPER=1) an additional 9 variables must be requested to make this available for post-processing, i.e., put NEIPH (NEIPS) equal to NHV+9. This makes the deformation gradient available in the d3plot database as history variables NHV+1 to NHV+9, note however that for shells it is expressed in the co-rotational system. If the local coordinate system option (IORTHO=1) is used, then the deformation gradient is expressed in this local system. To make the deformation gradient in the global system for bricks and co-rotational system for shells available and stored as history variables NHV+10 to NHV+18, NEIPH (NEIPS) is put equal to NHV+9+9(=NHV+18).

APPENDIX B: User Defined Equation of State

The user can supply his/her own subroutines defining equation of state (EOS) models in LS-DYNA. To invoke a user-defined EOS, one must

1. Write a user EOS subroutine that is called by the LS-DYNA user EOS interface.
2. Create a custom executable which includes the EOS subroutine.
3. Invoke that subroutine by defining a part in the keyword input deck that uses `*EOS_USER_DEFINED` with the appropriate input parameters.

All subroutines, including the interface, for the user-defined materials are collected in the file `dyn21.F` (Unix/Linux) or `lsdyna.f` (Windows). Up to ten user subroutines can currently be implemented simultaneously to update the pressure in solids and shells. This text serves as an introductory guide to implementing such a model. Note that names of variables and subroutines below may differ from the actual ones depending on platform and current version of LS-DYNA.

General overview

When the keyword `*EOS_USER_DEFINED` is defined for a part in the keyword deck, LS-DYNA calls the subroutine `ueoslib` with the appropriate input data for the EOS update. This subroutine is called twice for each integration point in each element. The first call requires the EOS to calculate the bulk modulus, and the second updates the pressure and internal energy. In these routines, which may be modified by the user if necessary, the following data structures are initialized for the purpose of being supplied to a specific *scalar* material subroutine.

`iflag` – =0 for calculating the bulk modulus, =1 for the pressure and energy update
`cb` – bulk modulus
`pnew` – the new pressure
`rho0` – reference density
`hist` – array of user-defined history variables `NHV` in length
`specen` – internal energy per unit reference volume
`df` – volume ratio, V/V0
`v0` – the initial volume.
`dvol` – volume increment
`pc` – pressure cut-off

If the *vectorization* flag is active (`IVECT=1`) on the EOS card, variables are, in general, stored in vector blocks of length `nlq`, with vector indices ranging from `lft` to `llt` , which allows for a more efficient execution of the EOS routine. As an example, the data structures mentioned above for the vectorized case are

`cb(nlq)` – bulk modulus
`Pnew(nlq)` – the new pressure
`hist(nlq,*)` – array of user-defined history variables with NHV columns
`specen(nlq)` – internal energy per unit reference volume
`df(nlq)` – volume ratio, V/V0
`v0(nlq)` – the initial volume
`dvol(nlq)` – volume increment
`pc(nlq)` – pressure cut-off

The value of nlq is set as a parameter in the include file nlqparm, included at the top of the subroutine, and varies between machines and operating systems. Each entry in a vector block is associated with an integration point in the finite element mesh. The number of entries in the history variables array (indicated by * in the above) matches the number of history variables requested on the material card (NHV). All history variables are initially zero and are initialized within the EOS on the first time step, when the logical variable first, passed through the argument list, is .TRUE. Furthermore, all user-defined EOS models require a bulk modulus, cb, for transmitting boundaries, contact interfaces, rigid body constraints, and time step calculations. In addition to the variables mentioned above, the following data can be supplied to the user material routines, regardless of whether vectorization is used or not.

eosp(*) – array of material constants from the input file
tt – current time
crv(101,2,*) – array representation of curves defined in the keyword deck.

A specific material routine, ueos*XX*s in the scalar case or ueos*XX*v in the vector case, is now called with any necessary parameters of the ones above, and possibly others as well. The use of curves is discussed in Appendix A. The letters *XX* stands for a number between 21 and 30 and matches the number EOST on the EOS card. This subroutine is written by the user, and should calculate the bulk modulus when iflag=0, and update the pressure, internal energy and history variables to the current time when iflag=1. During the input phase, the EOS is called with iflag=-1 to permit the initialization of any desired constants in eosp. Although fewer than 48 constants may be read into eosp during the input, the user may use all 48 within the EOS subroutines.

A sample scalar user subroutine for a Gruneisen EOS is provided below and it is immediately followed by its vector counterpart.

Sample user subroutine 21

```
      subroutine ueos21s(iflag,cb,pnew,hist,rho0,eosp,specen,
     &                   df,dvol,v0,pc,dt,tt,crv,first)
      include 'nlqparm'
c
c***  example scalar user implementation of the Gruneisen EOS
c
c***  variables
c          iflag ----- =0 calculate bulk modulus
c                      =1 update pressure and energy
c          cb -------- bulk modulus
c          pnew ------ new pressure
c          hist ------ history variables
c          rho0 ------ reference density
c          eosp ------ EOS constants
c          specen ---- energy/reference volume
c          df -------- volume ratio, v/v0 = rho0/rho
c          dvol ------ change in volume over time step
c          v0 -------- reference volume
c          pc -------- pressure cut-off
c          dt -------- time step size
c          tt -------- current time
c          crv ------- curve array
c          first ----- logical .true. for tt,crv,first time step
```

```
c                       (for initialization of the history variables)
c
      logical first
c
      dimension hist(*),eosp(*),crv(101,2,*)
c
      c  =eosp(1)
      s1 =eosp(2)
      s2 =eosp(3)
      s3 =eosp(4)
      g0 =eosp(5)
      sa =eosp(6)
      s11=s1-1.
      s22=2.*s2
      s33=3.*s3
      s32=2.*s3
      sad2=.5*sa
      g0d2=1.-.5*g0
      roc2=rho0*c**2
c
c***  calculate the bulk modulus for the EOS contribution to the sound speed
      if (iflag.eq.0) then
        xmu=1.0/df-1.
        dfmu=df*xmu
        facp=.5*(1.+sign(1.,xmu))
        facn=1.-facp
        xnum=1.+xmu*(+g0d2-sad2*xmu)
        xdem=1.-xmu*(s11+dfmu*(s2+s3*dfmu))
        tmp=facp/(xdem*xdem)
        a=roc2*xmu*(facn+tmp*xnum)
        b=g0+sa*xmu
        pnum=roc2*(facn+facp*(xnum+xmu*(g0d2-sa*xmu)))
        pden=2.*xdem*(-s11 +dfmu*(-s22+dfmu*(s2-s33+s32*dfmu)))
        cb=pnum*(facn+tmp)-tmp*a*pden+sa*specen+
     &         b*df**2*max(pc,(a+b*specen))
c
c***  update the pressure and internal energy
      else
        xmu=1.0/df-1.
        dfmu=df*xmu
        facp=.5*(1.+sign(1.,xmu))
        facn=1.-facp
        xnum=1.+xmu*(+g0d2-sad2*xmu)
        xdem=1.-xmu*(s11+dfmu*(s2+s3*dfmu))
        tmp=facp/(xdem*xdem)
        a=roc2*xmu*(facn+tmp*xnum)
        b=g0+sa*xmu
        dvov0=0.5*dvol/v0
        denom=1.+ b*dvov0
        pnew=(a+specen*b)/max(1.e-6,denom)
        pnew=max(pnew,pc)
        specen=specen-pnew*dvov0
      endif
c
      return
      end
      subroutine ueos21v(lft,llt,iflag,cb,pnew,hist,rho0,eosp,specen,
     &                   df,dvol,v0,pc,dt,tt,crv,first)
      include 'nlqparm'
c
c***  example vectorized user implementation of the Gruneisen EOS
c
c***  variables
```

```
c          lft,llt --- tt,crv,first and last indices into arrays
c          iflag ----- =0 calculate bulk modulus
c                      =1 update pressure and energy
c          cb -------- bulk modulus
c          pnew ------ new pressure
c          hist ------ history variables
c          rho0 ------ reference density
c          eosp ------ EOS constants
c          specen ---- energy/reference volume
c          df -------- volume ratio, v/v0 = rho0/rho
c          dvol ------ change in volume over time step
c          v0 -------- reference volume
c          pc -------- pressure cut-off
c          dt -------- time step size
c          tt -------- current time
c          crv ------- curve array
c          first ----- logical .true. for tt,crv,first time step
c                      (for initialization of the history variables)
c
      logical first
c
      dimension cb(*),pnew(*),hist(nlq,*),eosp(*),
     &          specen(*),df(*),dvol(*),pc(*),v0(*)
c
      c  =eosp(1)
      s1 =eosp(2)
      s2 =eosp(3)
      s3 =eosp(4)
      g0 =eosp(5)
      sa =eosp(6)
      s11=s1-1.
      s22=2.*s2
      s33=3.*s3
      s32=2.*s3
      sad2=.5*sa
      g0d2=1.-.5*g0
      roc2=rho0*c**2
c
c***  calculate the bulk modulus for the EOS contribution to the sound speed
      if (iflag.eq.0) then
        do i=lft,llt
          xmu=1.0/df(i)-1.
          dfmu=df(i)*xmu
          facp=.5*(1.+sign(1.,xmu))
          facn=1.-facp
          xnum=1.+xmu*(+g0d2-sad2*xmu)
          xdem=1.-xmu*(s11+dfmu*(s2+s3*dfmu))
          tmp=facp/(xdem*xdem)
          a=roc2*xmu*(facn+tmp*xnum)
          b=g0+sa*xmu
          pnum=roc2*(facn+facp*(xnum+xmu*(g0d2-sa*xmu)))
          pden=2.*xdem*(-s11 +dfmu*(-s22+dfmu*(s2-s33+s32*dfmu)))
          cb(i)=pnum*(facn+tmp)-tmp*a*pden+sa*specen(i)+
     &          b*df(i)**2*max(pc(i),(a+b*specen(i)))
        enddo
c
c***  update the pressure and internal energy
      else
        do i=lft,llt
          xmu=1.0/df(i)-1.
          dfmu=df(i)*xmu
          facp=.5*(1.+sign(1.,xmu))
          facn=1.-facp
```

```
            xnum=1.+xmu*(+g0d2-sad2*xmu)
            xdem=1.-xmu*(s11+dfmu*(s2+s3*dfmu))
            tmp=facp/(xdem*xdem)
            a=roc2*xmu*(facn+tmp*xnum)
            b=g0+sa*xmu
            dvov0=0.5*dvol(i)/v0(i)
            denom=1.+b*dvov0
            pnew(i)=(a+specen(i)*b)/max(1.e-6,denom)
            pnew(i)=max(pnew(i),pc(i))
            specen(i)=specen(i)-pnew(i)*dvov0
          enddo
        endif
c
        return
        end
```

The Gruneisen EOS implemented in the example subroutines has the same form as *EOS_GRUNEISEN, EOS Form 4. Its update of the pressure and the internal energy are typical for an EOS that is linear in the internal energy,

$$P = A(\rho) + B(\rho)E$$

where A and B correspond to the variables `a` and `b` in the example subroutines, and E is `specen`. Integrating the energy equation with the trapezoidal rule gives

$$E^{n+1} = E^n + \frac{1}{2}(\sigma'^n + \sigma'^{n+1})\Delta\varepsilon - \frac{1}{2}(P^n + q^n + P^{n+1} + q^{n+1})\frac{\Delta V}{V_0}$$

where the superscripts refer to the time step, ΔV is the change in the volume associated with the Gauss point and V_0 is the reference volume. Collecting all the energy contributions on the right hand side except for the contribution from the new pressure gives a simple linear relationship between the new internal energy and pressure,

$$E^{n+1} = \tilde{E} - \frac{P^{n+1}\Delta V}{2V_0}.$$

The value of `specen` passed to `ueosXX` for the pressure and energy update corresponds to $\tilde{E}$. Substituting this relation into the EOS and solving for the new pressure gives

$$P^{n+1} = \frac{A(\rho^{n+1}) + B(\rho^{n+1})\tilde{E}}{1 + \dfrac{B\Delta V}{2V_0}}.$$

The final update of the new energy is calculated using the new pressure. For a more general EOS, the nonlinear equation in the new pressure,

$$P^{n+1} = P(\rho^{n+1}, \tilde{E} - \frac{P^{n+1}\Delta V}{2V_0})$$

is solved iteratively using Newton iteration or successive substitution.

The pressure cut-off, `pc`, is used to limited the amount of pressure that can be generated by tensile loading, `pnew=max(pnew,pc)`. Its value is usually specified in the *MAT input, e.g., *MAT_JOHNSON_COOK. It is not enforced outside of the EOS subroutines, and it is up to the user to determine whether or not to enforce the pressure cut-off in `ueosXX`. If the user does enforce it, the pressure cut-off should be applied before the final update to the internal energy otherwise the energy will be incorrect.

Many of the calculations performed to calculate the bulk modulus are the same as those for updating the pressure and energy. Since the bulk modulus calculation always precedes the pressure update, the values may be saved in a `common` block during the bulk modulus calculation to reduce the cost of the pressure update. The arrays used to store the values in the vectorized subroutines should be dimensioned by `nlq`.

One of the most common errors in implementing an EOS from a paper or book is the use of the wrong internal energy. There are three internal energies in common use: the energy per unit mass, e_M , the energy per unit current volume, e_V , and the energy per unit reference volume, *E*. LS-DYNA always uses the energy per unit reference volume. Some useful relations for converting between EOS in the literature and the variables in LS-DYNA are

$$e_V = E\frac{V_0}{V} = \texttt{specen/df}$$

$$e_M = E\frac{V_0}{M} = \texttt{specen/rho0}$$

$$\rho = \rho_0\frac{V_0}{V} = \texttt{rho0/df}$$

APPENDIX C: User Defined Element Interface for Solids and Shells

In this appendix the user-defined element interface for solids and shells is described. The interface can accommodate either an integrated or a resultant element. For the integrated element, the user needs to supply two matrices defining the kinematical properties of the element, and choose between using standard LS-DYNA hourglass stabilization, a user-defined stabilization, or no stabilization when zero energy modes are not present. The number and location of the integration points is arbitrary, i.e., user-defined. For the resultant/discrete element formulations, the force and stiffness assembly must also be implemented. History variables can be associated with the user defined elements. If desired, the element may utilize more than the conventional 3 (for bricks) and 6 (for shells) degrees-of-freedom per node.

USER DEFINED ELEMENTS

The user element is implemented according to how standard elements are implemented in LS-DYNA with the exception that two user routines are called for setting up the matrices of interest. In the end, the gradient-displacement matrix B_{ijkK} is constructed with the property that

$$B_{ijkK} u_{kK} = \frac{\partial v_i}{\partial x_j}$$

where u_{kK} is the vector of velocity nodal degrees of freedom and the right hand side is the velocity gradient. Moreover, the determinant J of the jacobian matrix determining the mapping from the isoparametric to physical domain is needed for numerical integration. From these expressions, the strains are determined as the symmetric part of the velocity gradient and the spin as the corresponding antisymmetric part. The stresses are evaluated using the constitutive models in LS-DYNA and the internal forces are obtained from

$$f_{kK} = \int \sigma_{ij} B_{ijkK} dV$$

where σ_{ij} are the stresses. Furthermore, the geometric and material tangent stiffnesses are obtained through

$$K^{mat}_{iIjJ} = \int C_{klmn} B_{kliI} B_{mnjJ} dV$$

and

$$K^{geo}_{iIjJ} = \int \sigma_{mn} B_{kmiI} B_{knjJ} dV$$

where C_{klmn} is the tangent modulus for the material. The integrals are evaluated using user-defined quadrature using the determinant J.

For user-defined hourglass control, the user must provide the corresponding internal force and stiffness contribution in a separate user routine. There is also the option to provide the force and stiffness matrix directly for the entire element.

To invoke a user-defined element one must do the following:

1. Write user element subroutine that defines the kinematics or kinetics of the element.
2. Create a custom executable which includes these subroutines.
3. Invoke the element by specifying this on the corresponding `*SECTION` card.

The dummy subroutines for the user defined elements are provided to the user in a FORTRAN source file for you to modify along with the necessary object files to compile a new executable. Contact LSTC or your local distributor for information about how to obtain these files as well as what compiler/version to use for your specific platform. Up to five user elements can simultaneously be used for bricks and shells (i.e. a total of ten). This text serves as an introductory guide on how to implement such an element.

General overview

To activate a user-defined element, it is necessary to set `ELFORM` to a number between `101` and `105` on the `*SECTION` definition. By doing so, the kinematics of the elements in the corresponding part will be determined from calling the subroutine

```
subroutine uXXX_bYYY(bmtrx,gmtrx,gjac,…
dimension bmtrx(nlq,3,3,*),gmtrx(nlq,3,3),gjac(*)
```

where *XXX* is substituted for `shl` for a shell-section and `sld` for a solid-section and *YYY* is the number specified in position `ELFORM`. Depending on the choice of `ITAJ` in the input, the user should set the matrices as follows.

If `ITAJ=0`, then set the isoparametric gradient-displacement matrix, represented by the array `bmtrx` , and jacobian matrix, represented by the array `gmtrx`. Here, the first index corresponds to the LS-DYNA block loop index where `nlq` is the block size. For a more convenient notation in the following, we assign a correspondence between the arrays `gmtrx` and `bmtrx` in the subroutines to matrices/tensors as follows

`gmtrx(*,i,j)` - g_{ij}

`bmtrx(*,i,j,k)` - b_{ijk}

These matrices should be determined so that at the current integration point:

$$g_{ij} = \frac{\partial x_i}{\partial \xi_j}$$

$$b_{ijk} u_k = \frac{\partial v_i}{\partial \xi_j} \Delta t$$

In the above, summation over repeated indices is assumed. We use the following notation

$x_i(\xi_1, \xi_2, \xi_3, t) =$ ith component of the current position vector at isoparametric coordinate (ξ_1, ξ_2, ξ_3) and time t.

$v_i(\xi_1,\xi_2,\xi_3,t) =$ ith component of the velocity vector
at isoparametric coordinate (ξ_1,ξ_2,ξ_3) and time t.

$\Delta t =$ current time step

$u_k =$ kth component of the generalized local displacements

$\xi_i =$ ith component of the isoparametric coordinate ranging from -1 to 1,

For shells, there is an option to get all variables in either the LS-DYNA local coordinate system (`ILOC=0`) or in the global coordinate system (`ILOC=1`). The matrix for the coordinate system transformation is also passed to the user routines where the columns represent the local unit base vectors. The resulting strains must always be in the local coordinate system for the constitutive evaluations. For no extra degrees of freedom (see below), the index k in the displacement expression is determined from the formula

$$k = n(m-1) + d$$

where $n = 3$ if only translational degrees of freedom are present (typical for solids) and $n = 6$ if rotational degrees of freedom are present (typical for shells), m is the local node number ($m = 1,2,...$) and d is the degree of freedom. The translational degrees of freedom correspond to $d \le 3$ and the rotational degrees of freedom to $4 \le d \le 6$.

If `ITAJ=1`, the user should set up the physical gradient-displacement matrix, represented by the array `bmtrx`, and jacobian determinant, represented by the array `gjac`. Again, we assign a correspondence between the arrays `gjac` and `bmtrx` in the subroutines to matrices/tensors as follows

`gjac(*)`	-	J
`bmtrx(*,i,j,k)`	-	b_{ijk}

These matrices should be determined so that at the current integration point:

$$J = \det \frac{\partial x_i}{\partial \xi_j}$$

$$b_{ijk} u_k = \frac{\partial v_i}{\partial x_j} \Delta t$$

To be able to set up these matrices, a set of additional auxiliary variables are passed to the user element subroutines. These include the isoparametric coordinate, the element thickness, and the shape function values, and derivatives. Again, for shells these are expressed in either the local or global coordinate system depending on the user's choice. For more information on these variables, the user is referred to the comments in the subroutines.

The integrated elements can use up to a total of 100 integration points (in the plane for shells) at arbitrary locations. These must be specified in terms of isoparametric coordinates and weights following the first of the user-defined cards in the `*SECTION_...` input. The isoparametric coordinates should range from –1 to 1 and the weights should sum up to 4 for shells and 8 for solids.

It may be necessary to incorporate hourglass stabilization to suppress zero energy modes, this is done by putting `IHGF.GT.0` in the input. For `IHGF.EQ.1`, the LS-DYNA hourglass routines are used automatically and for `IHGF.EQ.2` or `IHGF.EQ.3` the user must provide hourglass force and stiffness in a specific user-defined routine. If `IHGF.EQ.3`, physical stabilization becomes available since the resultant material tangent moduli are passed to the hourglass routine to provide the current membrane, bending and coupled membrane-bending stiffness of the material. With C_{ij} denoting the material tangent modulus in matrix form, the resultant tangent moduli are expressed as

$$\bar{C}_{ij}^{0} = \int C_{ij} dV \qquad \text{(membrane)}$$

$$\bar{C}_{ij}^{1} = \int z^1 C_{ij} dV \qquad \text{(membrane-bending)}$$

$$\bar{C}_{ij}^{2} = \int z^2 C_{ij} dV \qquad \text{(bending)}$$

where z is the thickness coordinate for shells. For solids, only the first resultant modulus is passed. In this case the array has 21 entries that correspond to the subdiagonal terms of the 6 by 6 resultant matrix. For the matrix index (i, j) in the material tangent modulus matrix, where $i \geq j$, the index I of the array passed to the routine is given by

$$I = i(i-1)/2 + j$$

i.e., the subdiagonal terms are stored row-wise in the array. For shells, all three moduli are passed in the local coordinate system where each array has 15 entries corresponding to the subdiagonal terms of the 5 by 5 resultant matrices. The through thickness direction is here eliminated from the plane stress assumption. The formula for the array indices transformation above holds. This subroutine is called

```
subroutine uXXX_eYYY(force,stiff,ndtot,…
dimension force(nlq,*),stiff(nlq,ndtot,*)
```

where again *XXX* and *YYY* should be substituted as described for the other subroutines in the above. The variables in the subroutine corresponds to the force and stiffness as

`force(*,i)` - f_i

`stiff(*,i,j)` - K_{ij}

where the indices corresponds to node and degree of freedom numbers exactly as for the displacements. For shells the force and stiffness is set up in the local element system (`ILOC=0`) or global system (`ILOC=1`). The variable `ndtot` is the total number of degrees of freedom for the element. Passed to this subroutine are also the property parameters and history variables

associated with the element. The values of the property parameters are defined in the input of a user-defined element. No more than 40 property parameters and 100 history variables can be used for each user-defined element. The history variables must be updated in this routine by the user.

Resultant/discrete elements
By putting NIP(P) equal to 0 in the input, a resultant/discrete element is assumed understood. For this option (which is incompatible with IHGF.GT.0) the user must provide force and stiffness in the same user-defined routine as for the user-defined hourglass control. This means that no material routine is called to update stresses and history variables, but all that must be accounted for in the user element routine. Nevertheless, the user should define *MAT_ELASTIC as the material for the corresponding part with suitable values of the Young's modulus and Poisson's ratio. These material properties are used for time step calculations and for contact stiffnesses. Again, property parameters and history variables are passed to the routine, and for shells also the thicknesses of the elements. For the shell thickness update option (ISTUPD.GT.0 on *CONTROL_SHELL) it is up to the user to update the thicknesses in this routine.

In what follows, a short description of the additional features associated with the user elements is given.

Nodal fiber vectors
If a user-defined shell element formulation uses the nodal fiber vectors, this must be specified by putting IUNF=1 on the *SECTION_SHELL card. With this option the nodal fiber vectors are processed in the element routines and can be used as input for determining the b_{ijk}, g_{ij}/J, f_i and K_{ij} tensors/matrices in the user routines. If not, it is assumed that the fiber direction is normal to the plane of the shell at all times. These are expressed in either the local or global system depending on the user's choice. See comments in the subroutines for more information.

Extra degrees of freedom
Exotic element formulations may require extra degrees-of-freedom per node besides the translational (and rotational) degrees-of-freedom. Currently, up to 3 extra degrees of freedom per node can be used for user-defined elements. To use extra degrees of freedom, a scalar node must be defined for each node that makes up the connectivity of the user element. A scalar node is defined using the keyword *NODE_SCALAR_VALUE, in which the user also prescribe initial and boundary conditions associated with the extra variables. The connectivity of the user elements must then be specified with the option *ELEMENT_SOLID_DOF or *ELEMENT_SHELL_DOF, where an extra line is used to connect the scalar nodes to the element. As an example:

```
*NODE_SCALAR_VALUE
$     NID              V1              V2              V3      NDF
       11             1.0                                      1
       12             1.0                                      1
       13             1.0                                      1
       14             1.0                                      1
*ELEMENT_SHELL_DOF
$     EID     PID       N1       N2       N3       N4
        1       1        1        2        3        4
$                     NS1      NS2      NS3      NS4
                       11       12       13       14
```

defines an element with one extra degree of freedom. The initial value of the corresponding variable is `1.0` and it is unconstrained. Finally, the user sets the parameter `NXDOF` on the `*SECTION_…` card to `1`, `2` or `3` depending on how many extra degrees of freedom that should be used in the user-defined element. An array `xdof` containing the current values of these extra variables are passed to the user routines for setting up the correct kinematical properties, see comments in the routines for more information. The formula for the displacement index changes to

$$k = (n + n_{xdof})(m-1) + d$$

where n_{xdof} is the number of extra degrees of freedom. The extra degrees of freedom for each node corresponds to $n+1 \le d \le n+n_{xdof}$. For dynamic simulations, the mass corresponding to these extra nodes are defined using `*ELEMENT_INERTIA` or `*ELEMENT_MASS`.

Related keywords

The following is a list of keywords that apply to the user defined elements

The *SECTION_SHELL card

A third card with accompanying optional cards of the `*SECTION_SHELL` keyword must be added if the user defined element option is invoked

Card 3 and accompanying cards
Define if and only if ELFORM=101,102,103,104 or 105

Variable	NIPP	NXDOF	IUNF	IHGF	ITAJ	LMC	NHSV	ILOC
Type	I	I	I	I	I	I	I	I
Default	0	0	0	0	0	0	0	0

Define NIPP cards according to the following format.

Variable	XI	ETA	WGT					
Type	F	F	F					

Define LMC property parameters using 8 parameters per card.

Variable	P1	P2	P3	P4	P5	P6	P7	P8
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
ELFORM	… GT.100.AND.LT.106: User-defined shell
NIPP	Number of in-plane integration points for user-defined shell (0 if resultant element)
NXDOF	Number of extra degrees of freedom per node for user-defined shell
IUNF	Flag for using nodal fiber vectors in user-defined shell EQ.0: Nodal fiber vectors are not used. EQ.1: Nodal fiber vectors are used
IHFG	Flag for using hourglass stabilization (NIPP.GT.0) EQ.0: Hourglass stabilization is not used EQ.1: LS-DYNA hourglass stabilization is used EQ.2: User-defined hourglass stabilization is used
ITAJ	Flag for setting up finite element matrices (NIPP.GT.0) EQ.0: Set up matrices wrt isoparametric domain EQ.1: Set up matrices wrt physical domain
LMC	Number of property parameters
NHSV	Number of history variables
ILOC	Local coordinate system option EQ.0: All variables are passed in the local element system EQ.1: All variables are passed in the global system
XI	First isoparametric coordinate
ETA	Second isoparametric coordinate
WGT	Isoparametric weight
PI	Ith property parameter

For more information on the variables the user may consult the previous sections in this appendix.

The *SECTION_SOLID card

A second card with accompanying optional cards of the `*SECTION_SOLID` keyword must be added if the user defined elements option is invoked.

Card 2

Define if and only if ELFORM=101,102,103,104 or 105

Variable	NIP	NXDOF	IHGF	ITAJ	LMC	NHSV		
Type	I	I	I	I	I	I		
Default	0	0	0	0	0	0		

Define NIP cards according to the following format.

Variable	XI	ETA	ZETA	WGT				
Type	F	F	F	F				

Define LMC property parameters using 8 parameters per card.

Variable	P1	P2	P3	P4	P5	P6	P7	P8
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
ELFORM	…
	GT.100.AND.LT.106: User-defined solid
NIP	Number of integration points for user-defined solid (0 if resultant element)
NXDOF	Number of extra degrees of freedom per node for user-defined solid
IHFG	Flag for using hourglass stabilization (NIP.GT.0)
	EQ.0: Hourglass stabilization is not used
	EQ.1: LS-DYNA hourglass stabilization is used
	EQ.2: User-defined hourglass stabilization is used
ITAJ	Flag for setting up finite element matrices (NIP.GT.0)
	EQ.0: Set up matrices wrt isoparametric domain
	EQ.1: Set up matrices wrt physical domain
LMC	Number of property parameters
NHSV	Number of history variables
XI	First isoparametric coordinate
ETA	Second isoparametric coordinate
ZETA	Third isoparametric coordinate
WGT	Isoparametric weight
PI	Ith property parameter

For more information on the variables the user may consult the previous sections in this appendix.

Sample User Shell Element 101 (Belytschko-Tsay shell)
The geometry of the Belytschko-Tsay element in local coordinates can be written

$$x_i = (x_{iI} + \frac{t}{2}\xi_3\delta_{i3})N_I(\xi_1,\xi_2)$$

$$v_i = (v_{iI} + \frac{t}{2}\xi_3 e_{ij3}\omega_{jI})N_I(\xi_1,\xi_2)$$

where

$x_{iI} = i$: th component of coordinate of node I

$v_{iI} = i$: th component of translational velocity of node I

$\omega_{jI} = j$: th component of rotational velocity of node I

t = thickness of element

e_{ijk} = permutation tensor

N_I = shape function localized at node I

δ_{i3} = Kronecker delta

Taking the derivative of these expressions with respect to the isoparametric coordinate yields

$$\frac{\partial x_i}{\partial \xi_1} = (x_{iI} + \frac{t}{2}\xi_3\delta_{i3})\frac{\partial N_I}{\partial \xi_1}$$
$$\frac{\partial x_i}{\partial \xi_2} = (x_{iI} + \frac{t}{2}\xi_3\delta_{i3})\frac{\partial N_I}{\partial \xi_2}$$
$$\frac{\partial x_i}{\partial \xi_3} = \frac{t}{2}\delta_{i3}$$

and

$$\frac{\partial v_i}{\partial \xi_1} = (v_{iI} + \frac{t}{2}\xi_3 e_{ij3}\omega_{jI})\frac{\partial N_I}{\partial \xi_1}$$
$$\frac{\partial v_i}{\partial \xi_2} = (v_{iI} + \frac{t}{2}\xi_3 e_{ij3}\omega_{jI})\frac{\partial N_I}{\partial \xi_2}$$
$$\frac{\partial v_i}{\partial \xi_3} = \frac{t}{2}e_{ij3}\omega_{jI}N_I$$

respectively. Using these expressions the element is implemented as a user-defined shell as follows.

```
      subroutine ushl_b101(bmtrx,gmtrx,gjac,
     1     xi,eta,zeta,
     2     n1,n2,n3,n4,
     3     dn1dxi,dn2dxi,dn3dxi,dn4dxi,
     4     dn1deta,dn2deta,dn3deta,dn4deta,
     5     x1,x2,x3,x4,y1,y2,y3,y4,z1,z2,z3,z4,
     6     xdof,
     7     thick,thck1,thck2,thck3,thck4,
     8     fx1,fx2,fx3,fx4,
     9     fy1,fy2,fy3,fy4,
     .     fz1,fz2,fz3,fz4,
     .      gl11,gl21,gl31,gl12,gl22,gl32,gl13,gl23,gl33,
     .     lft,llt)
      include 'nlqparm'
c
c     Compute b and g matrix for user-defined shell 101
c
      dimension bmtrx(nlq,3,3,*),gmtrx(nlq,3,3),gjac(nlq)
      REAL n1,n2,n3,n4
      dimension x1(nlq),x2(nlq),x3(nlq),x4(nlq)
      dimension y1(nlq),y2(nlq),y3(nlq),y4(nlq)
      dimension z1(nlq),z2(nlq),z3(nlq),z4(nlq)
      dimension thick(nlq)
      dimension thck1(nlq),thck2(nlq),thck3(nlq),thck4(nlq)
      dimension xdof(nlq,8,3)
      dimension fx1(nlq),fx2(nlq),fx3(nlq),fx4(nlq)
      dimension fy1(nlq),fy2(nlq),fy3(nlq),fy4(nlq)
      dimension fz1(nlq),fz2(nlq),fz3(nlq),fz4(nlq)
       dimension gl11(nlq),gl21(nlq),gl31(nlq),
      .      gl12(nlq),gl22(nlq),gl32(nlq),
      .      gl13(nlq),gl23(nlq),gl33(nlq)
c
      do i=lft,llt
c
```

```
          gmtrx(i,1,1)=
     1         x1(i)*dn1dxi+x2(i)*dn2dxi+
     2         x3(i)*dn3dxi+x4(i)*dn4dxi
          gmtrx(i,2,1)=
     1         y1(i)*dn1dxi+y2(i)*dn2dxi+
     2         y3(i)*dn3dxi+y4(i)*dn4dxi
          gmtrx(i,3,1)=
     1         0.
          gmtrx(i,1,2)=
     1         x1(i)*dn1deta+x2(i)*dn2deta+
     2         x3(i)*dn3deta+x4(i)*dn4deta
          gmtrx(i,2,2)=
     1         y1(i)*dn1deta+y2(i)*dn2deta+
     2         y3(i)*dn3deta+y4(i)*dn4deta
          gmtrx(i,3,2)=
     1         0.
          gmtrx(i,1,3)=
     1         0.
          gmtrx(i,2,3)=
     1         0.
          gmtrx(i,3,3)=
     1         .5*thick(i)
c
          coef=.5*thick(i)*zeta
c
          bmtrx(i,1,1,1) =dn1dxi
          bmtrx(i,1,1,7) =dn2dxi
          bmtrx(i,1,1,13)=dn3dxi
          bmtrx(i,1,1,19)=dn4dxi
c
          bmtrx(i,1,1,5) =coef*dn1dxi
          bmtrx(i,1,1,11)=coef*dn2dxi
          bmtrx(i,1,1,17)=coef*dn3dxi
          bmtrx(i,1,1,23)=coef*dn4dxi
c
          bmtrx(i,1,2,1) =dn1deta
          bmtrx(i,1,2,7) =dn2deta
          bmtrx(i,1,2,13)=dn3deta
          bmtrx(i,1,2,19)=dn4deta
c
          bmtrx(i,1,2,5) =coef*dn1deta
          bmtrx(i,1,2,11)=coef*dn2deta
          bmtrx(i,1,2,17)=coef*dn3deta
          bmtrx(i,1,2,23)=coef*dn4deta
c
          bmtrx(i,2,1,2) =dn1dxi
          bmtrx(i,2,1,8) =dn2dxi
          bmtrx(i,2,1,14)=dn3dxi
          bmtrx(i,2,1,20)=dn4dxi
c
          bmtrx(i,2,1,4) =-coef*dn1dxi
          bmtrx(i,2,1,10)=-coef*dn2dxi
          bmtrx(i,2,1,16)=-coef*dn3dxi
          bmtrx(i,2,1,22)=-coef*dn4dxi
c
          bmtrx(i,1,3,5) =.5*thick(i)*n1
          bmtrx(i,1,3,11)=.5*thick(i)*n2
          bmtrx(i,1,3,17)=.5*thick(i)*n3
          bmtrx(i,1,3,23)=.5*thick(i)*n4
c
          bmtrx(i,3,1,3) =dn1dxi
          bmtrx(i,3,1,9) =dn2dxi
          bmtrx(i,3,1,15)=dn3dxi
```

```
          bmtrx(i,3,1,21)=dn4dxi
c
          bmtrx(i,2,2,2) =dn1deta
          bmtrx(i,2,2,8) =dn2deta
          bmtrx(i,2,2,14)=dn3deta
          bmtrx(i,2,2,20)=dn4deta
c
          bmtrx(i,2,2,4) =-coef*dn1deta
          bmtrx(i,2,2,10)=-coef*dn2deta
          bmtrx(i,2,2,16)=-coef*dn3deta
          bmtrx(i,2,2,22)=-coef*dn4deta
c
          bmtrx(i,2,3,4) =-.5*thick(i)*n1
          bmtrx(i,2,3,10)=-.5*thick(i)*n2
          bmtrx(i,2,3,16)=-.5*thick(i)*n3
          bmtrx(i,2,3,22)=-.5*thick(i)*n4
c
          bmtrx(i,3,2,3) =dn1deta
          bmtrx(i,3,2,9) =dn2deta
          bmtrx(i,3,2,15)=dn3deta
          bmtrx(i,3,2,21)=dn4deta
c
       enddo
c
       return
       end
```

To use the element for a part the section card can be written as

```
*SECTION_SHELL
$    SECID    ELFORM
         1       101
$       T1         T2         T3         T4

$     NIPP     NXDOF      IUNF      IHGF
         1         0         0         1
$       XI       ETA       WGT
        0.        0.        4.
```

Sample User Solid Element 101 (constant stress solid)
The geometry for the constant stress solid is given as

$$x_i = x_{iI} N_I(\xi_1, \xi_2)$$
$$v_i = v_{iI} N_I(\xi_1, \xi_2)$$

where

$x_{iI} = i$: th component of coordinate of node I

$v_{iI} = i$: th component of translational velocity of node I

$N_I =$ shape function localized at node I

The matrices necessary for implementing this element as a user-defined solid are derived from the expressions given by

$$\frac{\partial x_i}{\partial \xi_1} = x_{iI} \frac{\partial N_I}{\partial \xi_1}$$
$$\frac{\partial x_i}{\partial \xi_2} = x_{iI} \frac{\partial N_I}{\partial \xi_2}$$
$$\frac{\partial x_i}{\partial \xi_3} = x_{iI} \frac{\partial N_I}{\partial \xi_3}$$

and

$$\frac{\partial v_i}{\partial \xi_1} = v_{iI} \frac{\partial N_I}{\partial \xi_1}$$
$$\frac{\partial v_i}{\partial \xi_2} = v_{iI} \frac{\partial N_I}{\partial \xi_2}$$
$$\frac{\partial v_i}{\partial \xi_3} = v_{iI} \frac{\partial N_I}{\partial \xi_3}$$

The user element implementation is given by

```
      subroutine usld_b101(bmtrx,gmtrx,gjac,
     1     xi,eta,zeta,
     2     n1,n2,n3,n4,n5,n6,n7,n8,
     3     dn1dxi,dn2dxi,dn3dxi,dn4dxi,
     4     dn5dxi,dn6dxi,dn7dxi,dn8dxi,
     5     dn1deta,dn2deta,dn3deta,dn4deta,
     6     dn5deta,dn6deta,dn7deta,dn8deta,
     7     dn1dzeta,dn2dzeta,dn3dzeta,dn4dzeta,
     8     dn5dzeta,dn6dzeta,dn7dzeta,dn8dzeta,
     9     x1,x2,x3,x4,x5,x6,x7,x8,
     .     y1,y2,y3,y4,y5,y6,y7,y8,
     .     z1,z2,z3,z4,z5,z6,z7,z8,
     .     xdof,
     .     lft,llt)
      include 'nlqparm'
c
c     Compute b and g matrix for user-defined solid 101
c
      dimension bmtrx(nlq,3,3,*),gmtrx(nlq,3,3),gjac(nlq)
      REAL n1,n2,n3,n4,n5,n6,n7,n8
      dimension x1(nlq),x2(nlq),x3(nlq),x4(nlq)
      dimension x5(nlq),x6(nlq),x7(nlq),x8(nlq)
      dimension y1(nlq),y2(nlq),y3(nlq),y4(nlq)
      dimension y5(nlq),y6(nlq),y7(nlq),y8(nlq)
      dimension z1(nlq),z2(nlq),z3(nlq),z4(nlq)
      dimension z5(nlq),z6(nlq),z7(nlq),z8(nlq)
      dimension xdof(nlq,8,3)
c
      do i=lft,llt
c
         gmtrx(i,1,1)=x1(i)*dn1dxi+x2(i)*dn2dxi+
     1        x3(i)*dn3dxi+x4(i)*dn4dxi+
     2        x5(i)*dn5dxi+x6(i)*dn6dxi+
     3        x7(i)*dn7dxi+x8(i)*dn8dxi
```

```
          gmtrx(i,2,1)=y1(i)*dn1dxi+y2(i)*dn2dxi+
     1         y3(i)*dn3dxi+y4(i)*dn4dxi+
     2         y5(i)*dn5dxi+y6(i)*dn6dxi+
     3         y7(i)*dn7dxi+y8(i)*dn8dxi
          gmtrx(i,3,1)=z1(i)*dn1dxi+z2(i)*dn2dxi+
     1         z3(i)*dn3dxi+z4(i)*dn4dxi+
     2         z5(i)*dn5dxi+z6(i)*dn6dxi+
     3         z7(i)*dn7dxi+z8(i)*dn8dxi
          gmtrx(i,1,2)=x1(i)*dn1deta+x2(i)*dn2deta+
     1         x3(i)*dn3deta+x4(i)*dn4deta+
     2         x5(i)*dn5deta+x6(i)*dn6deta+
     3         x7(i)*dn7deta+x8(i)*dn8deta
          gmtrx(i,2,2)=y1(i)*dn1deta+y2(i)*dn2deta+
     1         y3(i)*dn3deta+y4(i)*dn4deta+
     2         y5(i)*dn5deta+y6(i)*dn6deta+
     3         y7(i)*dn7deta+y8(i)*dn8deta
          gmtrx(i,3,2)=z1(i)*dn1deta+z2(i)*dn2deta+
     1         z3(i)*dn3deta+z4(i)*dn4deta+
     2         z5(i)*dn5deta+z6(i)*dn6deta+
     3         z7(i)*dn7deta+z8(i)*dn8deta
          gmtrx(i,1,3)=x1(i)*dn1dzeta+x2(i)*dn2dzeta+
     1         x3(i)*dn3dzeta+x4(i)*dn4dzeta+
     2         x5(i)*dn5dzeta+x6(i)*dn6dzeta+
     3         x7(i)*dn7dzeta+x8(i)*dn8dzeta
          gmtrx(i,2,3)=y1(i)*dn1dzeta+y2(i)*dn2dzeta+
     1         y3(i)*dn3dzeta+y4(i)*dn4dzeta+
     2         y5(i)*dn5dzeta+y6(i)*dn6dzeta+
     3         y7(i)*dn7dzeta+y8(i)*dn8dzeta
          gmtrx(i,3,3)=z1(i)*dn1dzeta+z2(i)*dn2dzeta+
     1         z3(i)*dn3dzeta+z4(i)*dn4dzeta+
     2         z5(i)*dn5dzeta+z6(i)*dn6dzeta+
     3         z7(i)*dn7dzeta+z8(i)*dn8dzeta
c
          bmtrx(i,1,1,1) =dn1dxi
          bmtrx(i,1,1,4) =dn2dxi
          bmtrx(i,1,1,7) =dn3dxi
          bmtrx(i,1,1,10)=dn4dxi
          bmtrx(i,1,1,13)=dn5dxi
          bmtrx(i,1,1,16)=dn6dxi
          bmtrx(i,1,1,19)=dn7dxi
          bmtrx(i,1,1,22)=dn8dxi
c
          bmtrx(i,2,1,2) =dn1dxi
          bmtrx(i,2,1,5) =dn2dxi
          bmtrx(i,2,1,8) =dn3dxi
          bmtrx(i,2,1,11)=dn4dxi
          bmtrx(i,2,1,14)=dn5dxi
          bmtrx(i,2,1,17)=dn6dxi
          bmtrx(i,2,1,20)=dn7dxi
          bmtrx(i,2,1,23)=dn8dxi
c
          bmtrx(i,3,1,3) =dn1dxi
          bmtrx(i,3,1,6) =dn2dxi
          bmtrx(i,3,1,9) =dn3dxi
          bmtrx(i,3,1,12)=dn4dxi
          bmtrx(i,3,1,15)=dn5dxi
          bmtrx(i,3,1,18)=dn6dxi
          bmtrx(i,3,1,21)=dn7dxi
          bmtrx(i,3,1,24)=dn8dxi
c
          bmtrx(i,1,2,1) =dn1deta
          bmtrx(i,1,2,4) =dn2deta
          bmtrx(i,1,2,7) =dn3deta
```

```
           bmtrx(i,1,2,10)=dn4deta
           bmtrx(i,1,2,13)=dn5deta
           bmtrx(i,1,2,16)=dn6deta
           bmtrx(i,1,2,19)=dn7deta
           bmtrx(i,1,2,22)=dn8deta
c
           bmtrx(i,2,2,2) =dn1deta
           bmtrx(i,2,2,5) =dn2deta
           bmtrx(i,2,2,8) =dn3deta
           bmtrx(i,2,2,11)=dn4deta
           bmtrx(i,2,2,14)=dn5deta
           bmtrx(i,2,2,17)=dn6deta
           bmtrx(i,2,2,20)=dn7deta
           bmtrx(i,2,2,23)=dn8deta
c
           bmtrx(i,3,2,3) =dn1deta
           bmtrx(i,3,2,6) =dn2deta
           bmtrx(i,3,2,9) =dn3deta
           bmtrx(i,3,2,12)=dn4deta
           bmtrx(i,3,2,15)=dn5deta
           bmtrx(i,3,2,18)=dn6deta
           bmtrx(i,3,2,21)=dn7deta
           bmtrx(i,3,2,24)=dn8deta
c
           bmtrx(i,1,3,1) =dn1dzeta
           bmtrx(i,1,3,4) =dn2dzeta
           bmtrx(i,1,3,7) =dn3dzeta
           bmtrx(i,1,3,10)=dn4dzeta
           bmtrx(i,1,3,13)=dn5dzeta
           bmtrx(i,1,3,16)=dn6dzeta
           bmtrx(i,1,3,19)=dn7dzeta
           bmtrx(i,1,3,22)=dn8dzeta
c
           bmtrx(i,2,3,2) =dn1dzeta
           bmtrx(i,2,3,5) =dn2dzeta
           bmtrx(i,2,3,8) =dn3dzeta
           bmtrx(i,2,3,11)=dn4dzeta
           bmtrx(i,2,3,14)=dn5dzeta
           bmtrx(i,2,3,17)=dn6dzeta
           bmtrx(i,2,3,20)=dn7dzeta
           bmtrx(i,2,3,23)=dn8dzeta
c
           bmtrx(i,3,3,3) =dn1dzeta
           bmtrx(i,3,3,6) =dn2dzeta
           bmtrx(i,3,3,9) =dn3dzeta
           bmtrx(i,3,3,12)=dn4dzeta
           bmtrx(i,3,3,15)=dn5dzeta
           bmtrx(i,3,3,18)=dn6dzeta
           bmtrx(i,3,3,21)=dn7dzeta
           bmtrx(i,3,3,24)=dn8dzeta
c
      enddo
c
      return
      end
```

To use the element for a part the section card can be written as

```
*SECTION_SOLID
$     SECID     ELFORM
          1        101
$       NIP      NXDOF       IHGF
          1          0          1
$        XI        ETA       ZETA        WGT
         0.         0.         0.        8.0
```

Examples

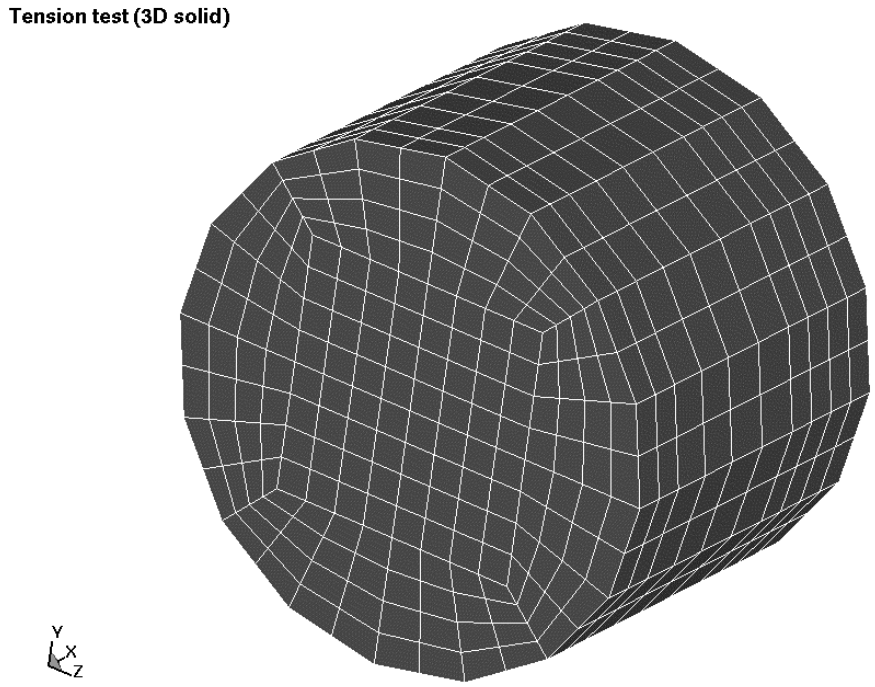


Figure C.1 Solid mesh for user element test.

We present three test examples.

One example was a simple tension-compression test of a solid cylinder. The geometry is shown in Figure C.1. The problem is using the sample implementations of user elements and compared the results and performance with standard LS-DYNA elements. As for the computational efficiency, we note that the performance is worse but this is expected since there is little room for optimization of the code while retaining a user friendly interface. The implicit performance compares well with the other elements in LS-DYNA.

The second example was a combined bending and stretching example with the geometry shown in Figure C.2. Again we ran the problem with the user element implementations and compared the results and performance with standard LS-DYNA elements. We could see the same tendencies as for the solid elements.

Figure C.2 Shell mesh for the user element test.

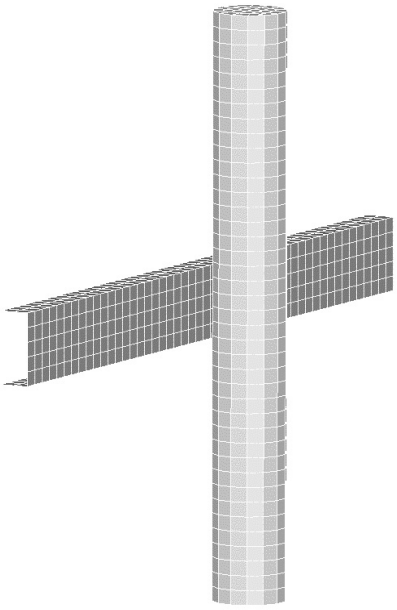


Figure C.3 Impact between a user-defined shell and user-defined solid part.

The third and final example is an impact between a solid bar and shell beam. Both parts are modeled with user-defined elements. The results were very similar to the ones obtained by substituting the sections for standard LS-DYNA sections, but the simulation time was about 3-4 times longer.

APPENDIX D: User Defined Airbag Sensor

The addition of a user sensor subroutine into LS-DYNA is relatively simple. The sensor is mounted on a rigid body which is attached to the structure. The motion of the sensor is provided in the local coordinate system defined for the rigid body in the definition of material model 20–the rigid material. When the user defined criterion is met for the deployment of the airbag, a flag is set and the deployment begins. All load curves relating to the mass flow rate versus time are then shifted by the initiation time. The user subroutine is given below with all the necessary information contained in the comment cards.

```
      SUBROUTINE AIRUSR (RBU,RBV,RBA,TIME,DT1,DT2,PARAM,HIST,ITRNON,
      . RBUG,RBVG,RBAG)
C***********************************************************************
C|    LIVERMORE SOFTWARE TECHNOLOGY CORPORATION (LSTC)                  |
C|    ------------------------------------------------------------     |
C|    COPYRIGHT 1987, 1988, 1989 JOHN O. HALLQUIST, LSTC               |
C|    ALL RIGHTS RESERVED                                              |
C***********************************************************************
C
C     USER SUBROUTINE TO INITIATE THE INFLATION OF THE AIRBAG
C
C     VARIABLES
C
C     DISPLACEMENTS ARE DEFINED AT TIME N+1 IN LOCAL SYSTEM
C     VELOCITIES ARE DEFINED AT TIME N+1/2 IN LOCAL SYSTEM
C     ACCELERATIONS ARE DEFINED AT TIME N IN LOCAL SYSTEM
C
C           RBU(1-3) TOTAL DISPLACEMENTS IN THE LOCAL XYZ DIRECTIONS
C           RBU(3-6) TOTAL ROTATIONS ABOUT THE LOCAL XYZ AXES
C           RBV(1-3) VELOCITIES IN THE LOCAL XYZ DIRECTIONS
C           RBV(3-6) ROTATIONAL VELOCITIES ABOUT THE LOCAL XYZ AXES
C           RBA(1-3) ACCELERATIONS IN THE LOCAL XYZ DIRECTIONS
C           RBA(3-6) ROTATIONAL ACCELERATIONS ABOUT THE LOCAL XYZ AXES
C           TIME IS THE CURRENT TIME
C           DT1 IS TIME STEP SIZE AT N-1/2
C           DT2 IS TIME STEP SIZE AT N+1/2
C           PARAM IS USER DEFINED INPUT PARAMETERS (MAX 25)
C           HIST IS USER DEFINED HISTORY VARIABLES (MAX 25)
C           ITRNON IS FLAG TO TURN ON THE AIRBAG INFLATION
C           RBUG,RBVG,RBAG, ARE SIMILAR TO RBU,RBV,RBA BUT ARE DEFINED
C           GLOBALLY.
C
C     THE USER SUBROUTINE SETS THE VARIABLE ITRNON TO:
C
C              ITRNON=0 BAG IS NOT INFLATED
C              ITRNON=1 BAG INFLATION BEGINS AND THIS SUBROUTINE IN NOT
C                       CALLED AGAIN
C
      DIMENSION RBU(6),RBV(6),PARAM(25),HIST(25),
      . RBUG(6),RBVG(6),RBAG(6)
      RETURN
      END
```

APPENDIX E: User Defined Solution Control

This subroutine may be provided by the user to control the I/O, monitor the energies and other solution norms of interest, and to shut down the problem whenever he pleases. The arguments are defined in the listing provided below. This subroutine is called each time step and does not need any control card to operate.

```
      SUBROUTINE UCTRL1 (NUMNP,NDOF,TIME,DT1,DT2,PRTC,PLTC,FRCI,PRTO,
     . PLTO,FRCO,VT,VR,AT,AR,UT,UR,XMST,XMSR,IRBODY,RBDYN,USRHV,
     . MESSAG,TOTALM,CYCL,IDRINT)
C*****************************************************************
C|    LIVERMORE SOFTWARE TECHNOLOGY CORPORATION (LSTC)             |
C|    ------------------------------------------------------------ |
C|    COPYRIGHT 1987, 1988, 1989 JOHN O. HALLQUIST, LSTC          |
C|    ALL RIGHTS RESERVED                                         |
C*****************************************************************
C
      CHARACTER*(*) MESSAG
      INTEGER CYCLE
C
C
C     USER SUBROUTINE FOR SOLUTION CONTROL
C
C     NOTE:  LS-DYNA USED AN INTERNAL NUMBERING SYSTEM TO
C            ACCOMMODATE ARBITRARY NODE NUMBERING.  TO ACCESS
C            INFORMATION FOR USER NODE N, ADDRESS ARRAY LOCATION M,
C            M=LQF(N,1).  TO OBTAIN USER NODE NUMBER, N,
C            CORRESPONDING TO ARRAY ADDRESS M, SET N=LQFINV(M,1)
C
C     ARGUMENTS:
C        NUMNP=NUMBER OF NODAL POINTS
C        NDOF=NUMBER OF DEGREES IF FREEDOM PER NODE
C        TIME=CURRENT SOLUTION TIME
C        PRTC=OUTPUT INTERVAL FOR LS-DYNA TIME HISTORY DATA
C        PLTC=OUTPUT INTERVAL FOR LS-DYNA STATE DATA
C        FRCI=OUTPUT INTERVAL FOR LS-DYNA INTERFACE FORCE DATA
C        PRTO=OUTPUT TIME FOR TIME HISTORY FILE
C        PLTO=OUTPUT TIME FOR STATE DATA
C        FRCO=OUTPUT TIME FOR FORCE DATA
C        VT(3,NUMNP)  =NODAL TRANSLATIONAL VELOCITY VECTOR
C        VR(3,NUMNP)  =NODAL ROTATIONAL VELOCITY VECTOR.  THIS ARRAY
C                      IS DEFINED IF AND ONLY IF NDOF=6
C        AT(3,NUMNP)  =NODAL TRANSLATIONAL ACCELERATION VECTOR
C        AR(3,NUMNP)  =NODAL ROTATIONAL ACCELERATION VECTOR.  THIS
C                      ARRAY IS DEFINED IF AND ONLY IF NDOF=6
C        UT(3,NUMNP)  =NODAL TRANSLATIONAL DISPLACEMENT VECTOR
C        UR(3,NUMNP)  =NODAL ROTATIONAL DISPLACEMENT VECTOR.  THIS ARRAY
C                      IS DEFINED IF AND ONLY IF NDOF=6
C        XMST(NUMNP)  =RECIPROCAL OF NODAL TRANSLATIONAL MASSES
C        XMSR(NUMNP)  =RECIPROCAL OF NODAL ROTATIONAL MASSES.  THIS
C                      ARRAY IS DEFINED IF AND ONLY IF NDOF=6
C        IRBODY       =FLAG FOR RIGID BODY NODAL POINTS
C                      IF DEFORMABLE NODE THEN SET TO 1.0
C                      IF RIGID BODY NODE THEN SET TO 0.0
C                      DEFINED IF AN ONLY IF RIGID BODY ARE PRESENT
C                      I.E.,IRBODY.NE.0 IF NO RIGID BODY ARE PRESENT
C        USRHV(LENHV)  =USER DEFINED HISTORY VARIABLES THAT ARE STORED
```

```
C                        IN THE RESTART FILE.  LENHV=100+U*NUMMAT WHERE
C                       NUMMAT IS THE # OF MATERIALS IN THE PROBLEM.
C                       ARRAY USRHV IS UPDATED ONLY IN THIS SUBROUTINE.
C        MESSAG        =FLAG FOR DYNA3D WHICH MAY BE SET TO:
C                       'SW1.' LS-DYNA TERMINATES WITH RESTART FILE
C                       'SW3.' LS-DYNA WRITES A RESTART FILE
C                       'SW4.' LS-DYNA WRITES A PLOT STATE
C        TOTALM        =TOTAL MASS IN PROBLEM
C           CYCLE =CYCLE NUMBER
C           IDRINT      =FLAG FOR DYNAMIC RELAXATION PHASE
C                        .NE.0:  DYNAMIC RELAXATION IN PROGRESS
C                        .EQ.0:  SOLUTION PHASE
C
      COMMON/PTIMES/  PRTIMS(32),PRTLST(32),IGMPRT
C
C     PRTIMS(32)=OUTPUT INTERVALS FOR ASCII FILES
C
C     ASCII FILES
C           ( 1)=CROSS SECTION FORCES
C           ( 2)=RIGID WALL FORCES
C           ( 3)=NODAL DATA
C           ( 4)=ELEMENT DATA
C           ( 5)=GLOBAL DATA
C           ( 6)=DISCRETE ELEMENTS
C           ( 7)=MATERIAL ENERGIES
C           ( 8)=NODAL INTERFACE FORCES
C           ( 9)=RESULTANT INTERFACE FORCES
C           (10)=SMUG ANIMATOR
C           (11)=SPC REACTION FORCES
C           (12)=NODAL CONSTRAIN RESULTANT FORCES
C           (13)=AIRBAG STATISTICS
C           (14)=AVS DATABASE
C           (15)=NODAL FORCE GROUPS
C           (16)=OUTPUT INTERVALS FOR NODAL BOUNDARY CONDITIONS
C           (17)-(32)=UNUSED AT THIS TIME
C
C     PRTLST(32)=OUTPUT TIMES FOR ASCII FILES ABOVE.  WHEN SOLUTION TIME
C             EXCEEDS THE OUTPUT TIME A PRINT STATE IS DUMPED.
C
      COMMON/RBKENG/ENRBDY,RBDYX,RBDYY,RBDYZ
C
C     TOTAL RIGID BODY ENERGIES AND MOMENTUMS:
C             ENRBDY=RIGID BODY KINETIC ENERGY
C             RBDYX =RIGID BODY X-MOMENTUM
C             RBDYY =RIGID BODY Y-MOMENTUM
C             RBDYZ =RIGID BODY Z-MOMENTUM
C
      COMMON/RBKENG/ENRBDY,RBDYX,RBDYY,RBDYZ
C
C     TOTAL RIGID BODY ENERGIES AND MOMENTUMS:
C             SWXMOM=STONEWALL X-MOMENTUM
C             SWYMOM=STONEWALL Y-MOMENTUM
C             SWZMOM=STONEWALL Z-MOMENTUM
C             ENRBDY=STONEWALL KINETIC ENERGY
C
```

```
            COMMON/DEENGS/DEENG
C
C     DEENG=TOTAL DISCRETE ELEMENT ENERGY
C
      COMMON/ENERGY/XPE
C
C     XPE   =TOTAL INTERNAL ENERGY IN THE FINITE ELEMENTS
C
      DIMENSION VT(3,*),VR(3,*),AT(3,*),AR(3,*),UT(3,*),UR(3,*)
      XMST(*),XMSR(*),RBDYN(*),USRHV(*)
C
C     SAMPLE MOMENTUM AND KINETIC ENERGY CALCULATIONS
C
C     REMOVE ALL COMMENTS IN COLUMN 1 BELOW TO ACTIVATE
CC
CC
CC    INITIALIZE KINETIC ENERGY, XKE, AND X,Y,Z MOMENTUMS.
CC
C     XKE=2.*SWKENG+2.*ENRBDY
C     XM-SWXMOM+RBDYX
C     YM=SWYMOM+RBDYY
C     ZM=SWZMOM+RBDYZ
CC
C     NUMNP2=NUMNP
C     IF (NDOF.EQ.6) THEN
C     NUMNP2=NUMNP+NUMNP
C     ENDIF
C     PRINT *,NDOF
C     IF(IRBODY.EQ.0) THEN
CC
CC
CC    NO RIGID BODIES PRESENT
CC
CC    NOTE IN BLANK COMMENT VR FOLLOWS VT.  THIS FACT IS USED BELOW.
C     DO 10 N=1,NUMNP2
C     XMSN=1./XMST(N)
C     VN1=VT(1,N)
C     VN2=VT(2,N)
C     VN3=VT(3,N)
C     XM=XM+XMSN*VN1
C     YM=YM+XMSN*VN2
C     ZM=ZM+XMSN*VN3
C     XKE=XKE+XMSN*(VN1*VN1+VN2*VN2+VN3*VN3)
C  10 CONTINUE
CC
C     ELSE
CC
CC    RIGID BODIES PRESENT
CC
C     DO 20 N=1,NUMNP
C     XMSN=1./XMST(N)
C     VN1=RBDYN(N)*VT(1,N)
C     VN2=RBDYN(N)*VT(2,N)
C     VN3=RBDYN(N)*VT(3,N)
C     XM=XM+XMSN*VN1
C     YM=YM+XMSN*VN2
C     ZM=ZM+XMSN*VN3
C     XKE=XKE+XMSN*(VN1*VN1+VN2*VN2+VN3*VN3)
C  20 CONTINUE
C     IF (NDOF.EQ.6) THEN
C     DO 30 N=1,NUMNP
```

```
C     XMSN=1./XMSR(N)
C     VN1=RBDYN(N)*VR(1,N)
C     VN2=RBDYN(N)*VR(2,N)
C     VN3=RBDYN(N)*VR(3,N)
C     XM=XM+XMSN*VN1
C     YM=YM+XMSN*VN2
C     ZM=ZM+XMSN*VN3
C     XKE=XKE+XMSN*(VN1*VN1+VN2*VN2+VN3*VN3)
C  30 CONTINUE
C     ENDIF
CC
C     ENDIF
      RETURN
      END
CC
CC.....TOTAL KINETIC ENERGY
C      XKE=.5*XKE
CC.....TOTAL INTERNAL ENERGY
C      XIE=.XPE+DEENG
CC.....TOTAL ENERGY
C      XTE=XKE+XPE+DEENG
CC.....TOTAL X-RIGID BODY VELOCITY
C      XRBV=XM/TOTALM
CC.....TOTAL Y-RIGID BODY VELOCITY
C      YRBV=YM/TOTALM
CC.....TOTAL Z-RIGID BODY VELOCITY
C      ZRBV=ZM/TOTALM
C
      RETURN
      END
```

APPENDIX F: User Defined Interface Control

This subroutine may be provided by the user to turn the interfaces on and off. This option is activated by the *USER_INTERFACE_CONTROL keyword. The arguments are defined in the listing provided below.

```
      SUBROUTINE UCTRL2 (NSI,NTY,TIME,CYCLE,MSR,NMN,NSV,NSN,
     1     THMR,THSV,VT,XI,UT,ISKIP,IDRINT,NUMNP,DT2,NINPUT,UA)
C*****************************************************************
C|    LIVERMORE SOFTWARE TECHNOLOGY CORPORATION (LSTC)             |
C|    -------------------------------------------------------     |
C|    COPYRIGHT 1987, 1988, 1989 JOHN O. HALLQUIST, LSTC          |
C|    ALL RIGHTS RESERVED                                         |
C*****************************************************************
C
      INTEGER CYCLE
C
C
C     USER SUBROUTINE FOR INTERFACE CONTROL
C
C     NOTE:  LS-DYNA USED AN INTERNAL NUMBERING SYSTEM TO
C            ACCOMMODATE ARBITRARY NODE NUMBERING.  TO ACCESS
C            INFORMATION FOR USER NODE N, ADDRESS ARRAY LOCATION M,
C            M=LQF(N,1).  TO OBTAIN USER NODE NUMBER, N,
C            CORRESPONDING TO ARRAY ADDRESS M, SET N=LQFINV(M,1)
C
C     ARGUMENTS:
C        NSI          =NUMBER OF SLIDING INTERFACE
C        NTY          =INTERFACE TYPE.
C                      .EQ.4:SINGLE SURFACE
C                      .NE.4:SURFACE TO SURFACE
C        TIME         =CURRENT SOLUTION TIME
C        CYCLE        =CYCLE NUMBER
C        MSR(NMN)     =LIST OF MASTER NODES NUMBERS IN INTERNAL
C                       NUMBERING SCHEME
C        NMN          =NUMBER OF MASTER NODES
C        NSV(NSN)     =LIST OF SLAVE NODES NUMBERS IN INTERNAL
C                       NUMBERING SCHEME
C        NSN          =NUMBER OF SLAVE NODES
C        THMR(NMN)    =MASTER NODE THICKNESS
C        THSV(NSN)    =SLAVE NODE THICKNESS
C        VT(3,NUMNP)  =NODAL TRANSLATIONAL VELOCITY VECTOR
C        XI(3,NUMNP)  =INITIAL COORDINATES AT TIME=0
C        UT(3,NUMNP)  =NODAL TRANSLATIONAL DISPLACEMENT VECTOR
C        IDRINT       =FLAG FOR DYNAMIC RELAXATION PHASE
C                      .NE.0:DYNAMIC RELAXATION IN PROGRESS
C                      .EQ.0:SOLUTION PHASE
C        NUMNP        =NUMBER OF NODAL POINTS
C        DT2          =TIME STEP SIZE AT N+1/2
C        NINPUT       =NUMBER OF VARIABLES INPUT INTO UA
C        UA(*)        =USER'S ARRAY, FIRST NINPUT LOCATIONS
C                       DEFINED BY USER.  THE LENGTH OF THIS
C                       ARRAY IS DEFINED ON CONTROL CARD 10.
C                       THIS ARRAY IS UNIQUE TO INTERFACE NSI.
C
C     SET FLAG FOR ACTIVE CONTACT
C       ISKIP=0 ACTIVE
C       ISKIP=1 INACTIVE
C
C*****************************************************************
```

```
      DIMENSION MSR(*),NSV(*),THMR(*),THSV(*),VT(3,*),XI(3,*),
                UT(3,*)UA(*)
C
C     THE FOLLOWING SAMPLE OF CODEING IS PROVIDED TO ILLUSTRATE HOW
C     THIS SUBROUTINE MIGHT BE USED.  HERE WE CHECK TO SEE IF THE
C     SURFACES IN THE SURFACE TO SURFACE CONTACT ARE SEPARATED.  IF
C     SO THE ISKIP=1 AND THE CONTACT TREATMENT IS SKIPPED.
C
      IF (NTY.EQ.4) RETURN
      DT2HLF=DT2/2.
      XMINS= 1.E20
      XMAXS=-XMINS
      YMINS= 1.E20
      YMAXS=-YMINS
      ZMINS= 1.E20
      ZMAXS=-ZMINS
      XMINM= 1.E20
      XMAXM=-XMINM
      YMINM= 1.E20
      YMAXM=-YMINM
      ZMINM= 1.E20
      ZMAXM=-ZMINM
      THKS=0.0
      THKM=0.0
      DO 10 I=1,NSN
      DSP1=UT(1,NSV(I))+DT2HLF*VT(1,NSV(I))
      DSP2=UT(2,NSV(I))+DT2HLF*VT(2,NSV(I))
      DSP3=UT(3,NSV(I))+DT2HLF*VT(3,NSV(I))
      X1=XI(1,NSV(I))+DSP1
      X2=XI(2,NSV(I))+DSP2
      X3=XI(3,NSV(I))+DSP3
      THKS =MAX(THSV(I),THKS)
      XMINS=MIN(XMINS,X1)
      XMAXS=MAX(XMAXS,X1)
      YMINS=MIN(YMINS,X2)
      YMAXS=MAX(YMAXS,X2)
      ZMINS=MIN(ZMINS,X3)
      ZMAXS=MAX(ZMAXS,X3)
   10 CONTINUE
      DO 20 I=1,NMN
      DSP1=UT(1,MSR(I))+DT2HLF*VT(1,MSR(I))
      DSP2=UT(2,MSR(I))+DT2HLF*VT(2,MSR(I))
      DSP3=UT(3,MSR(I))+DT2HLF*VT(3,MSR(I))
      X1=XI(1,MSR(I))+DSP1
      X2=XI(2,MSR(I))+DSP2
      X3=XI(3,MSR(I))+DSP3
      THKM =MAX(THMR(I),THKS)
      XMINS=MIN(XMINM,X1)
      XMAXS=MAX(XMAXM,X1)
      YMINS=MIN(YMINM,X2)
      YMAXS=MAX(YMAXM,X2)
      ZMINS=MIN(ZMINM,X3)
      ZMAXS=MAX(ZMAXM,X3)
   20 CONTINUE
      IF (XMAXS+THKS.LT.XMINM-THKM) GO TO 40
      IF (YMAXS+THKS.LT.YMINM-THKM) GO TO 40
      IF (ZMAXS+THKS.LT.ZMINM-THKM) GO TO 40
      IF (XMAXS+THKM.LT.XMINS-THKS) GO TO 40
      IF (YMAXS+THKM.LT.YMINS-THKS) GO TO 40
      IF (ZMAXS+THKM.LT.ZMINS-THKS) GO TO 40
      ISKIP=0
      RETURN
   40 ISKIP=1
      RETURN
      END
```

APPENDIX G: User Defined Interface Friction and Conductivity

An easy-to-use user contact interface is provided in LS-DYNA where the user has the possibility to define the frictional coefficients (static and dynamic) as well as contact heat transfer conductance as functions of contact pressure, relative sliding velocity, separation and temperature. To be able to use this feature, an object version of the LS-DYNA code is required and the user must write his/her own Fortran (or C) code to define the contact parameters of interest.
In the text file dyn21.f that comes with the object version of LS-DYNA, the subroutines of interest are

```
subroutine usrfrc(fstt,fdyn,…)
```

for defining the frictional coefficients fstt (static) and fdyn (dynamic) and

subroutine usrhcon(h,…)

for defining the heat transfer contact conductance h.
We emphasize at this point that the user friction interface differs between LS-DYNA (SMP) and MPP-DYNA (MPP), for reasons that have to do with how the contacts are implemented in general. In LS-DYNA (SMP) the user is required not only to define the frictional coefficients but also to assemble and store contact forces and history, whereas in MPP-DYNA (MPP) only the frictional coefficients have to be defined.
For the friction interface (SMP and MPP) the user may associate history variables with each contact node. Unfortunately the user friction interface is currently not supported by all available contacts in LS-DYNA and MPP-DYNA, but should cover the most interesting ones. Upon request by customers new contact types can be supported.
One of the arguments to the user contact routines is the curve array `crv`, also available in the user material interface. Note that when using this array, the curve identity must be converted to an internal number or the subroutine `crvval` may be utilized. For more information, see the appendix A on user materials.
For definition of user contact parameters the user must define the keywords

***USER_INTERFACE_FRICTION**

or

***USER_INTERFACE_CONDUCTIVITY**

The card format for these two keywords are identical and can be found in other sections in this manual.
There is an alternate route to defining the conductivity parameters for a user defined thermal contact. On the *CONTACT_..._THERMAL_FRICTION optional card the parameter FORMULA may be set to a negative number. This will automatically create a user defined conductivity interface and invoke reading of –FORMULA contact parameters immediately following the card including the FORMULA parameter. Note that FORMULA is related to NOC and NOCI in the *USER_INTERFACE_CONDUCTIVITY keyword as

–FORMULA=NOC=NOCI.

Note that the pressure is automatically computed for each user conductivity interface, i.e., the keyword *LOAD_SURFACE_STRESS is not necessary.

A sample friction subroutine is provided below for SMP.

```
      subroutine usrfrc(nosl,time,ncycle,dt2,insv,areas,xs,ys,zs,
     . lsv,ix1,ix2,ix3,ix4,aream,xx1,xx2,xx3,stfn,stf,fni,
     . dx,dy,dz,fdt2,ninput,ua,side,iisv5,niisv5,n1,n2,n3,fric1,
     . fric2,fric3,fric4,bignum,fdat,iseg,fxis,fyis,fzis,ss,tt,
     . ilbsv,stfk,frc,numnp,npc,pld,lcfst,lcfdt,temp,temp_bot,
     . temp_top,isurface)
c
c*****************************************************************
c|  LIVERMORE SOFTWARE TECHNOLOGY CORPORATION  (LSTC)             |
c|  ------------------------------------------------------------  |
c|  COPYRIGHT © 1987-2007 JOHN O. HALLQUIST, LSTC                 |
c|  ALL RIGHTS RESERVED                                           |
c*****************************************************************
c
c     user subroutine for interface friction control
c
c     note:  LS-DYNA uses an internal numbering system to
c            accomodate arbitrary node numbering.   to access
c            information for user node n, address array location m,
c            m=lqf(n,1). to obtain user node number, n,
c            corresponding to array address m, set n=lqfinv(m,1)
c
c     arguments:
c
c          nosl       =number of sliding interface
c          time       =current solution time
c          ncycle     =ncycle number
c          dt2        =time step size at n+1/2
c          insv       =slave node array where the nodes are stored
c                      in ls-dyna3d internal numbering.  User numbers
c                      are given by function: lqfinv(insv(ii),1)
c                      for slave node ii.
c          areas(ii)  =slave node area (interface types 5&10 only) for
c                      slave node ii
c          xs(ii)     =x-coordinate slave node ii (projected)
c          ys(ii)     =y-coordinate slave node ii (projected)
c          zs(ii)     =z-coordinate slave node ii (projected)
c          lsv(ii)    =master segment number for slave node ii
c          ix1(ii), ix2(ii), ix3(ii), ix4(ii)
c                     =master segment nodes in ls-dyna3d internal
c                      numbering for slave node ii
c          aream(ii)  =master segment area for slave node ii.
c          xx1(ii,4)  =x-coordinates master surface (projected) for
c                      slave node ii
c          xx2(ii,4)  =y-coordinates master surface (projected) for
c                      slave node ii
c          xx3(ii,4)  =z-coordinates master surface (projected) for
c                      slave node ii
c          stfn       =slave node penalty stiffness
c          stf        =master segment penalty stiffness
c          fni        =normal force
c          dx,dy,dz   =relative x,y,z-displacement between slave node and
c                      master surface.  Multipling by fdt2 defines the
c                      relative velocity.
c          n1,n2,n3   =x,y, and z components of master segments normal
c                      vector
c
c*****************************************************************
```

```
c       frictional coefficients defined for the contact interface
c
c          fric1      =static friction coefficient
c          fric2      =dynamic friction coefficient
c          fric3      =decay constant
c          fric4      =viscous friction coefficient (setting fric4=0
c                      turns this option off)
c
c***********************************************************************
c
c          bignum     =0.0 for one way surface to surface and
c                      for surface to surface, and 1.e+10 for nodes
c                      to surface contact
c          ninput     =number of variables input into ua
c          ua(*)      =users' array, first ninput locations
c                      defined by user.  the length of this
c                      array is defined on control card 10.
c                      this array is unique to interface nosl.
c
c          side       ='master' for first pass.  the master
c                        surface is the surface designated in the
c                        input
c                     ='slave' for second pass after slave and
c                      master surfaces have be switched for
c                      the type 3 symmetric interface treatment.
c
c          iisv5      =an array giving the pointers to the active nodes
c                      in the arrays.
c
c          niisv5     =number of active nodes
c
c          fdat       =contact history data array
c          iseg       =contact master segment from previous step.
c          fxis       =slave node force component in global x dir.
c                      to be updated to include friction
c          fyis       =slave node force component in global y dir.
c                      to be updated to include friction
c          fzis       =slave node force component in global z dir.
c                      to be updated to include friction
c          ss(ii)     =s contact point (-1 to 1) in parametric coordinates
c                      for slave node ii.
c          tt(ii)     =t contact point (-1 to 1) in parametric coordinates
c                      for slave node ii.
c          ilbsv(ii)  =pointer for node ii into global arrays.
c          stfk(ii)   =penalty stiffness for slave node ii which was used
c                      to compute normal interface force.
c          frc(1,lsv(ii))
c                     =Coulomb friction scale factor for segment lsv(ii)
c          frc(2,lsv(ii))
c                     =viscous friction scale factor for segment lsv(ii)
c
c***********************************************************************
c       parameters for a coupled thermal-mechanical contact
c
c          numnp      = number of nodal points in the model
c          npc        = load curve pointer
c          pld        = load curve (x,y) data
c          lcfst(nosl)= load curve number for static coefficient of
c                       friction versus temperture for contact
c                       surface nosl
c          lcfdt(nosl)= load curve number for dynamic coefficient of
c                       friction versus temperture for contact
c                       surface nosl
c          temp(j)    = temperature for node point j
c          temp_bot(j)= temparature for thick thermal shell bottom
c                       surface
```

```
c          temp_top(j)= temparature for thick thermal shell top
c                       surface
c          numsh12    = number of thick thermal shells
c          itopaz(1)  = 999 ==> thermal-mechanical analysis
c          isurface   = thick thermal shell surface pointer
c
c=======================================================================
```

APPENDIX H: User Defined Thermal Material Model

The addition of a thermal user material routine into LS-DYNA is fairly straightforward. The thermal user material is controlled using the keyword *MAT_THERMAL_USER_ DEFINED, which is described at the appropriate place in the manual.

The thermal user material can be used alone or in conjunction with any given mechanical material model in a coupled thermal-mechanical solution. A heat-source can be included and the specific heat updated so that it possible to model e.g. phase transformations including melt energy.

If for the same part (shell or solid elements) both a thermal and mechanical user material model is defined then the two user material models have (optionally) read access to each other's history variables. If the integration points of the thermal and mechanical elements not are coincident then interpolation or extrapolation is used when reading history variables. Linear interpolation or extrapolation using history data from the two closest integration points is used in all cases except when reading history variables from the thick thermal shell (TSHELL=1 on *CONTROL_SHELL). For the latter thermal shell, the shape functions of the element are used for the interpolation or extrapolation.

The thermal user materials are thermal material types 11-15. These thermal user material subroutines are defined in file dyn21.f as subroutines thumat11, … , thumat15. The latter subroutines are called from the subroutine thusrmat. The source code of subroutine thusrmat is also in file dyn21.f. Additional useful information is available in the comments of subroutines thusrmat, thumat12, and umat46 that all reside in the source file dyn21.f

Thermal history variables

Thermal history variables can be used by setting NVH greater that 0. Thermal history variables are output to the tprint file, see *DATABASE_TPRINT.

Interchange of history variables with mechanical user material

In a coupled thermo-mechanical solution there is for each mechanical shell, thick shell, or solid element a corresponding thermal element. A pair consisting of a mechanical and a corresponding thermal element both have integration points and possibly history variables. The mechanical and thermal elements do not necessarily have the same number of integration points.

By setting IHVE to 1, a thermal user material model can read, but not write, the history variables from a mechanical user material model and vice versa.

If the locations of the points where the history variables are located differ between the mechanical and thermal element differ interpolation or extrapolation is used to calculate the history value. More information is available in the comments to the subroutines thusrmat and thumat11.

Limitations

Currently there are a few limitations of the thermal user material implementation. LS-DYNA will in most cases give an appropriate warning or error message when such a limit is violated. The limitations include:

1. Option IHVE.EQ.1 is only supported for a limited range of mechanical elements:
   - Solid elements: ELFORM=1, 2, 10, 13.

- Shell elements: ELFORM=2, 3, 4, 16. Note that user-defined integration rules are not supported.

2. Thermal history variables limitations:
   - Thermal history variables are not output to d3plot.

3. The thermal solver includes not only the plastically dissipated energy as a heat source but also wrongly the elastic energy. The latter however is in most cases not of practical importance.

Example source code

Example source code for thermal user material models is available in thumat11 and thumat12 as well as in umat46. Note that there is space for up to 64 material parameters in r_matp (material parameter array) but only 32 can be read in from the *MAT_THERMAL_USER_DEFINED card. The material parameters in r_matp(i), i=41-64, which are initially set to 0.0, may be used by the user to store additional data.

Subroutine crvval evaluates load curves. Note that when using crvval the load curves are first re-interpolated to 100 equidistant points. See Appendix A for more information on subroutine crvval.

Following is a short thermal user material model. The card format is in this case, if enabling orthotropic conduction, and with sample input in SI-units:

***MAT_THERMAL_USER_DEFINED**

Card 1

Variable	MID	RO	MT	LMC	NVH	AOPT	IORTHO	IHVE
Type	21	7800.0	12.0	6.0	3.0	0.0	1.0	0.0

Card 2

Variable	XP	YP	ZP	A1	A2	A3		
Type	0.0	0.0	0.0	0.0	0.0	0.0		

Card 3

Variable	D1	D2	D3					
Type	0.0	0.0	0.0					

Card 4

Variable	C1	C2	C3	HC	HSRC	HCFAC		
Type	25.0	25.0	20.0	470.0	11.0	12.0		

VARIABLE	DESCRIPTION
C1-C3	Heat conduction in 11, 22, and 33 direction of material coordinate system.
HC	Heat capacity
HSRC	Load curve ID of load curve giving a heat source output (W/m3) as a function of time.
HCFAC	Load curve ID of load curve giving a scaling of the heat capacity as function of time.

The source code is:

```
      subroutine thumat12(c1,c2,c3,cvl,dcvdtl,hsrcl,dhsrcdtl,
     1     hsv,hsvm,nmecon,r_matp,crv,
     2     nel,nep,iep,eltype,dt,atime,ihsrcl)
      character*(*) eltype
      dimension hsv(*),hsvm(*),r_matp(*),crv(101,2,*)
      include 'iounits.inc'
c
c     Thermal user-material number 12.
c
c     See comments at the beginning of subroutine thusrmat
c     for instructions.
c
c     Example: isotropic/orthotropic material with k1=P1 and
c     cvl=P2 for solid and shell elements including optional
c     change of heat capacity and a heat source, both functions
c     of time input as load curves.
c
c     Print out some info on start-up, use material parameter 64
c     as a flag.
      if(nint(r_matp(64)).eq.0) then
         r_matp(64)=1.
         write( *,1200)  (r_matp(8+i),i=1,6)
         write(iohsp,1200) (r_matp(8+i),i=1,6)
         write(59,1200) (r_matp(8+i),i=1,6)
      endif
c
c     Calculate response
      c1=r_matp(8+1)
```

```
      c2=r_matp(8+2)
      c3=r_matp(8+3)
      cvl=r_matp(8+4)
      dcvdtl=0.0
      eid=nint(r_matp(8+6))
      if(nint(eid).gt.0) then
         call crvval(crv,eid,atime,cvlfac,tmp1)
         cvl=cvl*cvlfac
         dcvdtl=0.0
      endif
c
c     If flux or time step calculation then we are done.
      if(eltype.eq.'soliddt'.or.eltype.eq.'flux'.or.
     .     eltype.eq.'shelldt') return
      eid=nint(r_matp(8+5))
      if(nint(eid).gt.0) then
         ihsrcl=1
         call crvval(crv,eid,atime,hsrcl,tmp1)
         dhsrcdtl=0.0
      endif
c
c     Update history variables
      hsv(1)=cvl
      hsv(2)=atime
      hsv(3)=hsv(3)+1.0
c
c     Done
      return
 1200 format(/'This is thermal user defined material #12. '/
     1        /' Material parameter c1-c3       : ',3E10.3
     2        /' Material parameter hc          : ',E10.3
     3        /' Heat source load curve         : ',F10.0
     4        /' hc scale factor load curve     : ',F10.0
     5        /' Thermal History variable 1     : cv'
     6        /' Thermal History variable 2-3   : Dummy'/)
      return
      end
```

APPENDIX I: Occupant Simulation Including the Coupling to Programs CAL3D and MADYMO

INTRODUCTION

LS-DYNA is coupled to occupant simulation codes to generate solutions in automotive crashworthiness that include occupants interacting with the automotive structure. In such applications LS-DYNA provides the simulation of the structural and deformable aspects of the model and the OSP (Occupant Simulation Program) simulates the motion of the occupant. There is some overlap between the two programs which provides flexibility in the modeling approach. For example, both the OSP and LS-DYNA have the capability of modeling seat belts and other deformable restraints. The advantage of using the OSP is related to the considerable databases and expertise that have been developed in the past for simulating dummy behavior using these programs.

The development of the interface provided LSTC a number of possible approaches. The approach selected is consistent with the LSTC philosophy of providing the most flexible and useful interface possible. This is important because the field of non-linear mechanics is evolving rapidly and techniques which are used today are frequently rendered obsolete by improved methodologies and lower cost computing which allows more rigorous techniques to be used. This does make the learning somewhat more difficult as there is not any single procedure for performing a coupling.

One characteristic of LS-DYNA is the large number of capabilities, particularly those associated with rigid bodies. This creates both an opportunity and a difficulty: LSDYNA3D has many ways approximating different aspects of problems, but they are frequently not obvious to users without considerable experience. Therefore, in this Appendix we emphasize modeling methods rather than simply listing capabilities.

THE LS-DYNA/OCCUPANT SIMULATION PROGRAM LINK

Coupling between the OSP and LS-DYNA is performed by combining the programs into a single executable. In the case of CAL3D, LS-DYNA calls CAL3D as a subroutine, but in the case of MADYMO, LS-DYNA is called as a subroutine. The two programs are then integrated in parallel with the results being passed between the two until a user defined termination time is reached.

The OSP and LS-DYNA have different approaches to the time integration schemes. The OSP time integrators are based on accurate implicit integrators which are valid for large time steps which are on the order of a millisecond for the particular applications of interest here. An iterative solution is used to insure that the problem remains in equilibrium. The implicit integrators are extremely good for smoothly varying loads, however, sharp nonlinear pulses can introduce considerable error. An automatic time step size control which decreases the time step size quickly restores the accuracy for such events. The LS-DYNA time integrator is based on an explicit central difference scheme. Stability requires that the time step size be less than the highest frequency in the system. For a coarse airbag mesh, this number is on the order of 100

microseconds while an actual car crash simulation is on the order of 1 microsecond. The smallest LS-DYNA models have at least 1,000 elements. Experience indicates that the cost of a single LS-DYNA time step for a small model is at least as great as the cost of a time step in the OSP. Therefore, in the coupling, the LS-DYNA time step is used to control the entire simulation including the OSP part. This approach has negligible cost penalties and avoids questions of stability and accuracy that would result by using a subcycling scheme between the two programs. Optionally, a subcycling scheme can be used, however, the results of the analysis have to be checked with care.

LS-DYNA has a highly developed rigid body capability which is used in different parts of automobile crash simulation. In particular, components such as the engine are routinely modeled with rigid bodies. These rigid bodies have been modified so that they form the basis of the coupling procedure in LS-DYNA to the OSP.

In LS-DYNA, the geometry of a model is broken down into nodal points which identify positions in space. These nodes are then connected by elements so that the volume of a structure is identified. Each element has a "material" associated with it. If the element is deformable, then the material will specify its characteristics such as density and Young's Modulus. A crash model can consist of 100 or more separate materials which are each assigned a "material number," and each material number has an associated "material type" which determines if it is elastic, plastic, viscoelastic, orthotropic, etc.

The material type may also specify that it is a rigid body. In this case, all elements of the same material number are treated as a single rigid body. These elements are integrated to determine the mass, centroid and moments of inertia for the group. This group is then treated as a rigid body with six degrees-of-freedom including three translations and three rotations. The positions of the rigid bodies are updated in LS-DYNA by a time integrator which works together with the central difference time integration.

There is an additional flag which specifies that the LS-DYNA rigid body is coupled to an OSP rigid body. This flag can be found in the description of the rigid body material *MAT_RIGID (formerly material type 20). In coupled updates, the OSP rigid body time integrator takes over control of the LS-DYNA rigid body and the normal LS-DYNA updates are bypassed. The time integration procedure is then as follows:

1. At the beginning of a step, LS-DYNA determines the locations and updates the positions of all of the rigid bodies which are coupled to the OSP. This information is obtained from common block information in the OSP.

2. Using the information on rigid body locations, LS-DYNA proceeds to update the stresses and history variables of all of the deformable structures and computes the resultant forces acting on all rigid bodies.

3. The resultant forces are stored into an OSP common block along with the current time step. Control is then returned to the OSP so that the step can be completed by the OSP determining the new positions of the rigid bodies based on the applied forces.

At the end of the calculation LS-DYNA terminates normally, closing its files, and then control is returned to OSP which will also terminate normally. The termination time for the coupled run is

taken as the minimum of the termination time provided to LS-DYNA and the termination time provided to the OSP.

The executable for the coupling with MADYMO currently needs to be specially created at each site. TNO provides all of the appropriate load modules with their libraries, and the appropriate load modules for LS-DYNA may be obtained by the corporate contact point at the LS-DYNA distributor. A complete executable must then be made by linking the two libraries. A revised password file must be obtained from TNO prior to running the coupled code. Coupling with CAL3D requires special on-site modification of the client's CAL3D version to eliminate conflicting I/O unit numbers and to ensure that the common block lengths between the codes are consistent. LSTC does not distribute or support CAL3D.

To make the coupled program run, an input deck must be provided to both the OSP and LS-DYNA. The two input decks must be provided in the same set of consistent units. This can potentially require a major conversion to either the OSP input or the LS-DYNA input. With two legitimate and consistent input decks, the coupled program should run to completion with no problems. Additional inputs are required to make the models interact between the OSP and LS-DYNA portions of the run.

The simplest form of a coupled simulation is simply to include a single body in an OSP run. No special modifications are needed to the OSP input deck for use in the coupled simulation. Ellipsoids and planes in the OSP are usually attached to "segments" which correspond to LS-DYNA "rigid bodies." Because the coupling procedure works on the basis of shared information on LS-DYNA rigid bodies with the OSP segments, the ellipsoids/planes listed in the OSP section must correspond to the segments which are to be coupled. These ellipsoids and planes may be actual geometry which is used for contact, or they may be simply artificial shapes to permit the data transfer between the OSP and LS-DYNA.

DUMMY MODELING

The dummy is typically modeled entirely within the OSP. The coupling of the dummy into LS-DYNA requires the creation of a separate LS-DYNA rigid body material for each segment of the OSP. The easiest way to create a mesh for the model is to set the LS-DYNA rigid body coupling option to 2.0. This causes LS-DYNA to search all of the ellipsoids connected to the appropriate segment and generate meshes which are then slaved to the OSP dummy. Thus, with minimal input, a complete dummy may be generated and the kinematics may be traced in LS-DYNA and displayed in the LS-DYNA post-processor, LS-PREPOST.

Once the basic dummy coupling has been accomplished, the deformable finite element structure can be added. Assuming that an ellipsoid is available for the steering wheel, a flat airbag can be added in the proper location. One or more nodes must be attached to the steering wheel. This is done by identifying the attached nodes as "Extra Nodes for Rigid Body" which is input in LS-DYNA by *CONSTRAINED_EXTRA_NODES_Option. The nodes are slaved to the LS-DYNA material which has been coupled to the MADYMO steering wheel model. Contact must now be identified between the airbag and the steering wheel, the windshield, and the various body parts which may be affected. This requires the use of one geometric contact entity (see *CONTACT_ENTITY) for each plane or ellipsoid which may interact with the airbag. A control volume specifying inflation properties for the airbag must be specified (see *AIRBAG_*OPTION*) to complete the model.

AIRBAG MODELING

Modeling of airbags is accomplished by use of shell or membrane elements in conjunction with a control volume (see *AIRBAG_*OPTION*) and possibly a single surface contact algorithm to eliminate interpenetrations during the inflation phase (see *CONTACT_*OPTION*). The contact types showing an "a" in front are most suited for airbag analysis. Current recommended material types for the airbags are:

*MAT_ELASTIC = Type 1. Elastic

*MAT_COMPOSITE_DAMAGE = Type 22. Layered orthotropic elastic for composites

*MAT_FABRIC = Type 34. Fabric model for folded airbags

Model 34 is a "fabric" model which can be used for flat bags. As a user option this model may or may not support compression.

The elements which can be used are as follows:

Belytschko-Tsay quadrilateral with 1 point quadrature. This element behaves rather well for folded and unfolded cases with only a small tendency to hourglass. The element tends to be a little stiff. Stiffness form hourglass control is recommended.

Belytschko-Tsay membrane. This model is softer than the normal Belytschko-Tsay element and can hourglass quite badly. Stiffness form hourglass is recommended. As a better option, the fully integrated Belytschko-Tsay membrane element can be chosen.

C0 Triangular element. The C0 triangle is very good for flat bag inflation and has no tendency to hourglass.

The best choice is a specially developed airbag membrane element with quadrilateral shape. This is an automatic choice when the fabric material is used.

As an airbag inflates, a considerable amount of energy is transferred to the surrounding air. This energy transfer decreases the kinetic energy of the bag as it inflates. In the control volume logic, this is simulated either by using either a mass weighted damping option or a back pressure on the bag based on a stagnation pressure. In both cases, the energy that is absorbed is a function of the fabric velocity relative to a rigid body velocity for the bag. For the mass weighted case, the damping force on a node is proportional to the mass times the damping factor times the velocity vector. This is quite effective in maintaining a stable system, but has little physical justification. The latter approach using the stagnation pressure method estimates the pressure needed to accelerate the surrounding air to the speed of the fabric. The formula for this is:

$$P = Area \times \alpha \times \left(\left(\vec{V}_i - \vec{V}_{cg} \right) \cdot \hat{n} \right)^2$$

This formula accomplishes a similar function and has a physical justification. Values of the damping factor, α, are limited to the range of 0 to 1, but a value of 0.1 or less is more likely to be a good value.

KNEE BOLSTER

The knee-to-knee bolster interactions are characterized by the stiffness of the knee being comparable to that of the knee bolster. Therefore, modeling the knee as a rigid body may produce large errors in the interaction forces. Calibrated force-deflection curves could be determined, but they would have no predictive value for slight changes to knee bolster designs. For this reason, a more accurate modeling of the compliance of the knee bolster and the knee is required.

The knee can be modeled as a combined rigid/deformable body. The rigid body is coupled to the OSP. Overlaying the rigid body are brick elements which model the "skin" that exists over the knees of the dummy. These brick elements use material type 6 (*MAT_VISCOELASTIC) which is a viscoelastic model that does a reasonable job of approximating the hysteretic behavior of rubbers. The inner layer of the brick elements is attached to the rigid body through the *CONSTRAINED_EXTRA_NODES Option. Between the knee bolster is a SURFACE-TO-SURFACE contact definition.

COMMON ERRORS

1. Improper airbag inflation or no inflation.

The most common problem is inconsistency in the units used for the input constants. An inflation load curve must also be specified. The normals for the airbag segments must all be consistent and facing outwards. If a negative volume results, this can sometimes be quickly cured by using the "flip" flag on the control volume definition to force inward facing normals to face outwards.

2. Excessive airbag distortions.

Check the material constants. Triangular elements should have less distortion problems than quadrilaterals. Overlapped elements at time zero can cause locking to occur in the contact leading to excessive distortions. The considerable energy input to the bag will create numerical noise and some damping is recommended to avoid problems.

3. The dummy passes through the airbag.

A most likely problem is that the contacts are improperly defined. Another possibility is that the models were developed in an incompatible unit system. The extra check for penetration flag if set to 1 on the contact control cards variable PENCHK in the *CONTACT_... definitions may sometimes cause nodes to be prematurely released due to the softness of the penalties. In this case the flag should be turned off.

4. The OSP fails to converge.

This may occur when excessively large forces are passed to the OSP. First, check that unit systems are consistent and then look for improperly defined contacts in the LS-DYNA input.

5. Time step approaches zero.

This is almost always in the airbag. If elastic or orthotropic (*MAT_ELASTIC or *MAT_COMPOSITE material 1 or 22) is being used, then switch to fabric material *MAT_FABRIC which is less time step size sensitive and use the fully integrated membrane element. Increasing the damping in the control volume usually helps considerably. Also, check for “cuts” in the airbag where nodes are not merged. These can allow elements to deform freely and cut the time step to zero.

APPENDIX J: Interactive Graphics Commands

Only the first four or less characters of command are significant. These commands are available in the interactive phase of LS-DYNA. The interactive graphics are available by using the "SW5." command after invoking the Ctrl-C interrupt. The MENU command brings up a push button menu. Only available in Unix and Linux.

ANIMATE	Animate saved sequence, stop with switch 1.
BACK	Return to previous display size after zoom, then list display attributes.
BGC	Change display background color RGB proportions BGC <red> <green> <blue>.
BIP	Select beam integration point for contour; BIP <#>.
CENTER	Center model, center on node, or center with mouse, i.e., center cent <value> or cent gin.
CL	Classification labels on display; class commercial_in_confidence.
CMA	Color materials on limited color displays.
COLOR	Set or unset shaded coloring of materials.
CONTOUR	View with colored contour lines; contour <component #> <list mat #>; see TAURUS manual.
COOR	Get node information with mouse.
COP	Hardcopy of display on the PC copy <laserj paintj tekcol coljet or epson>.
CR	Restores cutting plane to default position.
CUT	Cut away model outside of zoom window; use mouse to set zoom window size.
CX	Rotate slice plane at zmin about x axis.
CY	Rotate slice plane at zmin about y axis.
CZ	Rotate slice plane at zmin about z axis.
DIF	Change diffused light level for material; DIF <mat #, -1 for all> <value>.

DISTANCE	Set distance of model from viewer; DIST <value in normalized model dimensions>.
DMATERIALS	Delete display of material in subsequent views; DMAT <ALL or list of numbers>.
DRAW	Display outside edges of model.
DSCALE	Scale current displacement from initial shape.
DYN	After using TAURUS command will reset display to read current DYNA3D state data.
ELPLT	Set or unset element numbering in subsequent views.
END	Delete display and return to execution.
ESCAPE	Escapes from menu pad mode.
EXECUTE	Return to execution and keep display active.
FCL	Fix or unfix current contour levels.
FOV	Set display field of view angle; FOV <value in degrees>.
FRINGE	View with colored contour fringes; fringe <component #> <list mat #>; see TAURUS manual.
GETFRAME	Display a saved frame; GETF <frame #>.
HARDWARE	Hardware mode; workstation hardware calls are used to draw, move and color model; repeat command to reset to normal mode.
HELP	
HZB	Switch on or off hardware zbuffer for a subsequent view, draw or contour command; rotations and translations will be in hardware.
LIMIT	Set range of node numbers subsequent views; limit <first node #> <last node #>.
MAT	Re-enable display of deleted materials mat <all or list of numbers>.
MENU	Button menu pad mode.

MOTION	Motion of model through mouse movement or use of a dial box. The left button down enables translation in the plane, middle button rotation about axes in the plane; and with right button down in the out of plane axis; left and middle button down quit this mode.
MOV	Drag picked part to new position set with mouse.
NDPLT	Set or unset node numbering in subsequent views.
NOFRAME	Set and unset drawing of a frame around the picture.
PAUSE	Animation display pause in seconds
PHS2 or THISTORY	Time history plotting phase. Similar to LS-TAURUS.
PICK	Get element information with mouse.
POST	Enable or disable postscript mode on the PC and eps file is written as picture is drawn; remove eofs and initgraphics for eps use.
QUIT	Same as execute.
RANGE	Set fix range for contour levels; range <minvalue> <maxvalue>.
RAX	Reflect model about xy plane; restore command will switch-off reflections.
RAY	Reflect model about yz plane; restore command will switch-off reflections.
RAZ	Reflect model about zx plane, restore command will switch-off reflections.
RESTORE	Restores model to original position, also switches off element and node numbers, slice capper, reflections and cut model.
RETURN	Exit.
RGB	Change color red green blue element <mat #> <red> <green> <blue>.
RX	Rotate model about x axis.
RY	Rotate model about y axis.
RZ	Rotate model about z axis.

SAVE	Set or unset saving of display for animation.
SEQUENCE	Periodic plot during execution; SEQ <# of cycles> <commands> EXE.
SHR	Shrink element facets towards centroids in subsequent views, shrink <value>.
SIP	Select shell integration point for contour; SIP <#>.
SLICE	Slice model a z-minimum plane; slice <value in normalized model dimension> this feature is removed after using restore. Slice enables internal details for brick elements to be used to generate new polygons on the slice plane.
SNORMAL	Set or unset display of shell direction normals to indicate topology order.
SPOT	Draw node numbers on model spot <first #> <last # for range>.
TAURUS	LS-DYNA database, TAU <state #>, or state <state #>, reads LS-TAURUS file to extract previous state data.
TRIAD	Set or unset display of axis triad.
TSHELL	Set or unset shell element thickness simulation in subsequent views.
TV	Change display type.
TX	Translates model along x axis.
TY	Translates model along y axis.
TZ	Translates model along z axis.
V	Display model using painters algorithm.
VECTOR v or d	View with vector arrows of velocity or displacement; <v> or <d>.
ZB	Switch on or off zbuffer algorithm for subsequent view; or draw commands.
ZIN	Zoom in using mouse to set display size and position.
ZMA	Set position of zmax plane; ZMAX <value in normalized model dimensions>.
ZMI	Set position of zmin plane; ZMIN <value in normalized model dimensions>.
ZOUT	Zoom out using mouse to set displays size expansion and position.

APPENDIX K: Interactive Material Model Driver

INTRODUCTION

The interactive material model driver in LS-DYNA allows calculation of the material constitutive response to a specified strain path. Since the constitutive model subroutines in LS-DYNA are directly called by this driver, the behavior of the constitutive model is precisely that which can be expected in actual applications. In the current implementation the constitutive subroutines for both shell elements and solid elements can be examined.

INPUT DEFINITION

The material model driver is invoked by setting the total number of beam, shell, and solid elements to zero in a standard LS-DYNA input file. The number of material model definitions should be set to one, the number of load curves should be nine, and the termination time to the desired length of the driver run. The complete state dump interval is interpreted as the time step to be used in the material model driver run. Plotting information is saved for every step of a driver run and sufficient memory is allocated to save this information in core for the interactive plotting phase.

The input deck consists only of the TITLE card, the CONTROL cards, one MATERIAL DEFINITION, and NINE LOAD CURVES describing the strain path should be defined. These nine curves define the time history of the displacement gradient components shown in Table K.1.

The velocity gradient matrix, L_{ij}, is approximated by taking the time derivative of the components in Table K.1. If these components are considered to form a tensor S_{ij} , then

$$L_{ij}(t) = \frac{S_{ij}(t) - S_{ij}(t_{k-1})}{(t - t_k)}$$

and the strain rate tensor is defined as

$$d_{ij} = \frac{L_{ij} + L_{ij}^t}{2}$$

and the spin tensor as

$$\omega_{ij} = \frac{L_{ij} - L_{ij}^t}{2}$$

Table K.1 Load Curve Definitions versus Time

Load Curve Number	Component Definition
1	$\frac{\partial u}{\partial x}$
2	$\frac{\partial v}{\partial y}$
3	$\frac{\partial w}{\partial z}$
4	$\frac{\partial u}{\partial y}$
5	$\frac{\partial v}{\partial x}$
6	$\frac{\partial u}{\partial z}$
7	$\frac{\partial w}{\partial x}$
8	$\frac{\partial v}{\partial z}$
9	$\frac{\partial w}{\partial y}$

INTERACTIVE DRIVER COMMANDS

After reading the input file and completing the calculations, LS-DYNA gives a command prompt to the terminal. A summary of the available interactive commands is given below. An on-line help package is available by typing HELP. Only available in Unix and Linux.

ACCL	Scale all abscissa data by f. Default is f=1.
ASET amin omax	Set min and max values on abscissa to amin and amax, respectively. If amin=amax=0, scaling is automatic.
CHGL n	Change label for component n. LS-DYNA prompts for new label.
CONTINUE	Re-analyze material model.
CROSS c_1 c_2	Plot component c_1 versus c_2.
ECOMP	Display component numbers on the graphics display: 1 x-stress, 2 y-stress, 3 z-stress, 4 xy-stress, 5 yz-stress, 6 zx-stress, 7 effective plastic strain, 8 pressure, 9 von Mises (effective) stress, 10 1st principal deviatoric stress, 11 2nd principal deviatoric stress, 12 3rd principal deviatoric stress, 13 maximum shear stress, 14 1st principal stress, 15 2nd principal stress, 16 3rd principal stress, 17 ln (v/v0), 18 relative volume, 19 v0/v - 1.0, 20 1st history variable, 21 2nd history variable. Adding 100 or 400 to component numbers 1-16 yields strains and strain rates, respectively.
FILE name	Change pampers filename to name for printing.
GRID	Graphics displays will be overlaid by a grid of orthogonal lines.

NOGRID	Graphics displays will not be overlaid by a grid of orthogonal lines.
OSCL	Scale all ordinate data by f. Default is f=1.
OSET omin omax	Set min and max values on ordinate to omin and omax, respectively. If omin=omax=0, scaling is automatic.
PRINT	Print plotted time history data into file "pampers." Only data plotted after this command is printed. File name can be changed with the "file" command.
QUIT, END, T	Exit the material model driver program.
RDLC m n r_1 z_1 ... r_n z_n	Redefine load curve m using n coordinate pairs (r_1,z_1) (r_2,z_2),...(r_n,z_n).
TIME c	Plot component c versus time.
TV n	Use terminal output device type n. LS-DYNA provides a list of available devices.

Presently, the material model drive is implemented for solid and shell element material models. The driver does not yet support material models for beam elements.

APPENDIX L: VDA Database

VDA surfaces describe the surface of geometric entities and are useful for the simulation of sheet forming problems. The German automobile and automotive supplier industry (VDA) has defined the VDA guidelines [VDA 1987] for a proper surface definition used for the exchange of surface data information. In LS-DYNA, this format can be read and used directly. Some files have to be provided for proper linkage to the motion of the correlation parts/materials in LS-DYNA.

Linking is performed via names. To these names surfaces are attached, which in turn can be linked together from many files externally to LS-DYNA. Thus, arbitrary surfaces can be provided by a preprocessor and then can be written to various files. The so-called VDA file given on the LS-DYNA execution line via V=**vda** contains references to all other files. It also contains several other parameters affecting the treatment in the contact subroutines; see below.

The procedure is as follows. If VDA surfaces are to be used, the file specified by **vda** must have the following form. The file is free formatted with blanks as delimiters. Note that the characters "}" and "{" must be separated from the other input by spaces or new lines. The **vda** file may contain any number of input file specifications of the form:

file **afile bfile** {

alias definitions

}

alias definitions

followed by optional runtime parameters and a final end statement.

The file, **afile,** is optional, and if given must be the name of an ASCII input file formatted in accordance with the VDA Surface Interface Definitions as defined by the German automobile and automotive supply industry. **bfile** is required, and is the name of a binary VDA file. In a first run **afile** is given and **bfile** is created. In any further run, if the definitions have not changed, **afile** can be dropped and only **bfile** is needed. The purpose of **bfile** is that it allows for much faster initialization if the same VDA surfaces are to be used in a future LS-DYNA run.

If **afile** is given, **bfile** will always be created or overwritten. The alias definitions are used for linking to LS-DYNA and between the various surface definitions in the files defined by **afile** and **bfile.**

The alias definitions are of the form

alias **name** { el1 el2 ... eln }

where **name** is any string of up to 12 characters, and el1,...,eln are the names of VDA elements as specified in **afile**. The list of elements can be empty, in which case all the SURF and FACE VDA elements in **afile** will be used. Care should be taken to ensure that the alias **name** is unique, not only among the other aliases, but among the VDA element names in **afile**. This

collection of VDA elements can later be indicated by the alias **name**. In particular, **name** may appear in later alias definitions.

Often it is required that a punch or die be created by a simple offset. This can be achieved in the **vda** files in two ways, either on VDA elements directly, or on parts defined by aliases. This feature offers great capability in generating and using surface data information.

Offset version 1:

As an option, the keyword **offset** may appear in the alias list which allows a new surface to be created as a normal offset (plus translation) of a VDA element in the file. The keyword **offset** my be applied to VDA elements only, not aliases. The usage of **offset** follows the form

offset elem **normal x y z**

where **normal** is the amount to offset the surface along the normal direction, and **x**,**y**,**z** are the translations to be applied. The default normal direction is given by the cross product of the local u and v directions on the VDA surface, taken in that order. **normal** can be negative.

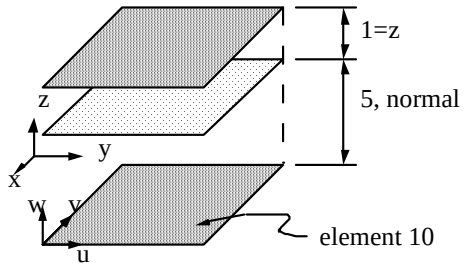
Offset version 2:

Frequently, it is convenient to create a new alias **name** by offsetting and translating an existing **name**. The keyword **goffset** provides this function:

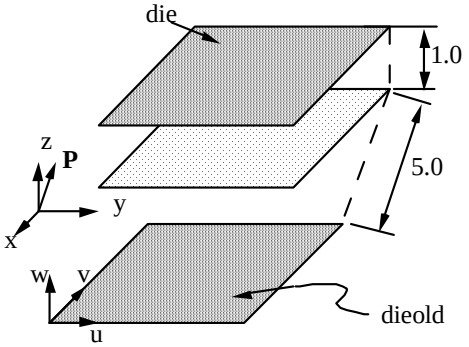
goffset **alias_name x_c y_c z_c normal x y z** { previous **alias_name** }

where **normal**, **x**, **y**, and **z** are defined as in the offset keyword. A reference point $\mathbf{x_c}$, $\mathbf{y_c}$, and $\mathbf{z_c}$ defines a point in space which determines the normal direction to the VDA surface, which is a vector from the origin to $P(x_c,y_c,z_c)$. See example below.

offset 10 5.0 0 0 1.0



offset alias die 1.0 2.0 1.0 5.0 0.0 1.0
{ previous alias dieold }



Finally, several parameters affecting the VDA surface iteration routines can be reset in the file **vda**. These parameters, and their default values in square brackets [ ], are:

gap [5.0] The maximum allowable surface gap to be filled in during the iterations. Points following the surface will effectively extend the edges of surfaces if necessary to keep them from falling through cracks in the surface smaller than this. This number should be set as small as possible while still allowing correct results. In particular, if your VDA surfaces are well formed (having no gaps), this parameter can be set to 0.0. The default value is 5.0.

t**rack** [2.0] A point must be within this distance of contact to be continually tracked. When a point not being tracked comes close to a surface, a global search is performed to find the near surface point. While a point is being tracked, iterations are performed every cycle. These iterations are much faster, but if the point is far away it is faster to occasionally do the global search. The default value is 2.0.

t**rack2** [5.0] Every VDA surface is surrounded by a bounding box. When a global search needs to be performed but the distance from a point to this box is > t**rack2**, the actual global search is not performed. This will require another global search to be performed sooner than if the actual distance to the surface were known, but also allows many global searches to be skipped. The default value is 5.0.

ntrack [4] The number of VDA surfaces for which each point maintains actual distance information. A global lower bound on distance is maintained for all remaining surfaces. Whenever the point moves far enough to violate this global lower bound, all VDA surfaces must have the global search performed for them. Hence, this parameter should be set to the maximum number of surfaces that any point can be expected to be near at one time (the largest number of surfaces that come together at one point). Setting **ntrack** higher will require more memory but result in faster execution. If **ntrack** is too low, performance may be unacceptably slow. The default value is 4.0.

toroid [.01] Any surface with opposing edges which are within distance [t] of each other is assumed to be cylindrical. Contacts occurring on one edge can pass to the adjacent edge. The default value is 0.01.

converge [.01] When surface iterations are performed to locate the near point, iteration is continued until convergence is detected to within this distance (all VDA coordinates are in mm). The default value is 0.01.

iterate [8] Maximum number of surface iterations allowed. Since points being tracked are checked every cycle, if convergence fails it will be tried again next cycle, so setting this parameter high does not necessarily help much. On the other hand, a point converging to a crease in the VDA surface (a crease between patches with discontinuous derivative, for example) may bounce back and forth between patches up to this many times, without actually moving. Hence, this value should not be too large. The default value is 8.

el_size [t mx mn]

Controls the generation of elements where:

t =surface tolerance for mesh generation,

mx=maximum element size to generate,

mn=minimum element size to generate.

The default values are [0.25 100. 1.0]

aspect [s1 s2] Controls the generation of elements where:

s1=maximum difference in aspect ratio between elements generated in neighboring VDA patches,

s2=maximum aspect ratio for any generated element.

The default values are [1.5 4.0]

cp_space [10] Determines the spacing around the boundaries of parts at which the size of elements is controlled. In the interior of the part, the element size is a weighted function of these control points as well as additional control points in the interior of the region. If there are too few control points around the boundary, elements generated along or near straight boundaries, but between control points, may be too small. The default value is 10.

meshonly The existence of this keyword causes LS-DYNA to generate a file containing the mesh for the VDA surfaces and then terminate.

onepatch The existence of this keyword causes LS-DYNA to generate a single element on each VDA patch.

somepatch [n] Like onepatch, but generates an element for 1 out of every [n] patches.

Example for file V=**vda**. It contains the following data:

```
file vda1 vda1.bin {
        alias die {
                sur0001
                sur0003
                offset fce0006 1.5 0 0 120
        }
        alias holder1 { sur008 }
}
file vda2 vda2.bin {
        alias holder2 { sur003 }
}
alias holder { holder1 holder2 }
ntrack 6
gap 0.5

end
```

Explanation:

vda1	This file contains the surfaces/face elements sur0001,sur0003, fce0006, and sur0008.
alias die face	Combines the surface/face elements sur0001, sur0003, and the offsetted fce0006 to a global surface.
alias holder1	Defines the surface/face element sur0008 as holder1.
vda2	This file contains the surface/face element sur0003.
alias holder2	Defines the surface/face element sur0003 as holder2.
alias holder	Combines the surfaces holder1 and holder2 into a combined surface holder.
ntrack 6	For each point the actual distances to 6 VDA surfaces are maintained.
gap 0.5	Surface gaps of 0.5mm or less are filled.
end	Closes reading of this file.

APPENDIX M: Commands for Two-Dimensional Rezoning

The rezoner in LS-DYNA contains many commands that can be broken down into the following categories:

- general,
- termination of interactive rezoning,
- redefinition of output intervals for data,
- graphics window controls,
- graphics window controls for x versus y plots,
- mesh display options,
- mesh modifications,
- boundary modifications,
- MAZE line definitions,
- calculation graphics display control parameters,
- calculation graphics display,
- cursor commands.

The use of the rezoner is quite simple. Commands for rezoning material number n can be invoked after the material is specified by the "M n" command. To view material n, the command "V" is available. The interior mesh can be smoothed with the "S" command and the boundary nodes can be adjusted after the "B" command is used to display the part side and boundary node numbers. Commands that are available for adjusting boundary nodes following the "B" command include:

ER, EZ, ES, VS, BD, ERS, EZS, ESS, VSS, BDS, SLN, SLNS

Rezoning is performed material by material. An example is shown.

Do not include the graphics display type number (see the "TV" command below) when setting up a command file for periodic noninteractive rezoning. No plotting is done when the rezoner is used in this mode.

REZONING COMMANDS BY FUNCTION

Interactive Real Time Graphics

SEQ n commands EXE	Every n time steps execute the graphics commands which follow. For example the line seq 100 g exe would cause the grid to be updated on the graphics display device every 100 cycles. The real time graphics can be terminated by using ctrl-c and typing "sw7."

General

C	Comment - proceed to next line.
FRAME	Frame plots with a reference grid (default).
HELP	Enter HELP package and display all available commands. Description of each command is available in the HELP package.
HELP/commandname	Do not enter HELP package but print out the description on the terminal of the command following the slash.
LOGO	Put LLNL logo on all plots (default). Retyping this command removes the logo.
NOFRAME	Do not plot a reference grid.
PHP ans	Print help package - If answer equals 'y' the package is printed in the high speed printer file.
RESO n_x n_y	Set the x and y resolutions of plots to n_x and n_y, respectively. We default both n_x and n_y to 1024.
TV n	Use graphics output device type n. The types are installation dependent and a list will be provided after this command is invoked.
TR t	At time t, LS-DYNA will stop and enter interactive rezoning phase.

Termination of Interactive Rezoning

F	Terminate interactive phase, remap, continue in execution phase.

FR	Terminate interactive phase, remap, write restart dump, and call exit.
T or END	Terminate.

Redefinition of Output Intervals for Data

PLTI Δt	Reset the node and element data dump interval Δt.
PRTI Δt	Reset the node and element printout interval Δt.
TERM t	Reset the termination to t.

Graphics Window Controls

ESET n	Center picture at element n with a Δr by Δz window. This window is set until it is released by the unfix command or reset with another window.
FF	Encircle picture with reference grid with tickmarks. Default grid is plotted along bottom and left side of picture.
FIX	Set the display to its current window. This window is set until it is reset by the "GSET, "FSET", or "SETF" commands or released by the "UNFIX" command.
FSET n Δr Δz	Center display at node n with a rectangular $\Delta r \infty \Delta z$ window. This window is set until it is reset with or the "UNFIX" command is typed.
GSET r z Δl	Center display picture at point (r,z) with square window of width Δl. This window is set until it is reset or the "UNFIX" command is typed.
GRID	Overlay graphics displays with a grid of orthogonal lines.
NOGRID	Do not overlay graphics displays with a grid of orthogonal lines (default).
SETF r z Δr Δz	Center display at point (r,z) with a rectangular $\Delta r \times \Delta z$ window. This window is set until it is reset or the "UNFIX" command is typed.
UNFIX	Release current display window set by the "FIX", "GSET", "FSET" or "SETF" commands.

UZ a b Δl	Zoom in at point (a,b) with window Δl where a, b, and Δl are numbers between 0 and 1. The picture is assumed to lie in a unit square.
UZG	Cover currently displayed picture with a 10 by 10 square grid to aid in zooming with the unity zoom, "UZ", command.
UZOU a b Δl	Zoom out at point (a,b) with window Δl where a, b, and Δl are numbers between 0 and 1. The current window is scaled by the factor 1/Δl.The picture is assumed to lie in a unit square.
Z r z Δl	Zoom in at point (r,z) with window Δl.
ZOUT r z Δl	Zoom out at point (r,z) with window Δl. The window is enlarged by the ratio of the current window and Δl. The cursor may be used to zoom out via the cursor command DZOU and entering two points with the cursor to define the window. The ratio of the current window with the specified window determines the picture size reduction. An alternative cursor command, DZZO, may be used and only needs one point to be entered at the location where the reduction (2×) is expected.

Graphics Window Controls for x versus y plots

The following commands apply to line plots, interface plots, etc.

ASCL f_a	Scale all abscissa data by f_a. The default is $f_a = 1$.
ASET amin amax	Set minimum and maximum values on abscissa to amin and amax, respectively. If amin=amax=0.0 (default) LS-DYNA determines the minimum and maximum values.
OSCL f_o	Scale all ordinate data by f_o. The default is $f_o = 1$.
OSET omin omax	Set minimum and maximum values on ordinate to omin and omax, respectively. If omin=omax=0.0 (default) LS-DYNA determines the minimum and maximum values.
SMOOTH n	Smooth a data curve by replacing each data point by the average of the 2n adjacent points. The default is n=0.

Mesh Display Options

ELPLT	Plot element numbers on mesh of material n.
FSOFF	Turn off the "FSON" command.

FSON	Plot only free surfaces and slideline interfaces with "O" command. (Must be used before "O" command.)
G	View mesh.
GO	View mesh right of centerline and outline left of centerline.
GS	View mesh and solid fill elements to identify materials by color.
M n	Material n is to be rezoned.
MNOFF	Do not plot material numbers with the "O", "G", and "GO" commands (default).
MNON	Plot material numbers with "O", "G", and "GO" commands.
NDPLT	Plot node numbers on mesh of material n.
O	Plot outlines of all material.
RPHA	Reflect mesh, contour, fringe, etc., plots about horizontal axis. Retyping "RPHA" turns this option off.
RPVA	Reflect mesh, contour, fringe, etc., plots about vertical axis. Retyping "RPVA" turns this option off.
TN r z Δl	Type node numbers and coordinates of all nodes within window (r $\pm$ $\Delta l/2$, z $\pm$ $\Delta l/2$).
UG	Display undeformed mesh.
V	Display material n on graphics display. See command M.
VSF	Display material n on graphics display and solid fill elements.

Mesh Modifications

BACKUP	Restore mesh to its previous state. This command undoes the result of the last command.
BLEN s	Smooth option where s=0 and s=1 correspond to equipotential and isoparametric smoothing, respectively. By letting $0 \le s \le 1$ a combined blending is obtained.
CN m r z	Node m has new coordinate (r,z).

DEB n f_1 l_1 ... f_n l_n	Delete n element blocks consisting of element numbers f_1 to l_1, f_2 to l_2 ... , and f_o l_n inclusive. These elements will be inactive when the calculation resume.
DE e_1 e_2	Delete elements e_1 to e_2.
DMB n m_1 m_2 ... m_n	Delete n material blocks consisting of all elements with material numbers m_1, m_2,..., and m_n. These materials will be inactive when the calculations resume.
DM n m_1 m_2 ... m_n	Delete n materials including m_1, m_2,..., and m_n.
DZER k d incr nrow	Delete element row where k is the kept element, d is the deleted element, incr is the increment, and nrow is the number of elements in the row.
DZLN number n_1 n_2 n_3...n_{last}	Delete nodal row where number is the number of nodes in the row and n_1, n_2, ... n_{last} are the ordered list of deleted nodes.
DZNR l j incr	Delete nodal row where l is the first node in the row, j is the last node in the row, and incr is the increment.
R	Restore original mesh.
S	Smooth mesh of material n. To smooth a subset of elements, a window can be set via the "GSET", "FSET", OR "SETF" commands. Only the elements lying within the window are smoothed.

Boundary Modifications

A	Display all slidelines. Slave sides are plotted as dashed lines.
B	Determine boundary nodes and sides of material n and display boundary with nodes and side numbers.
BD m n	Dekink boundary from boundary node m to boundary node n (counterclockwise).
BDS s	Dekink side s.
DSL n l_1 l_2...l_n	Delete n slidelines including slideline numbers l_1 l_2..., and l_n.
ER m n	Equal space in r-direction boundary nodes m to n (counterclockwise).
ERS s	Equal space in the r-direction boundary nodes on side s.

ES m n	Equal space along boundary, boundary nodes m to n (counterclockwise).
ESS s	Equal space along boundary, boundary nodes on side s.
EZ m n	Equal space in z-direction boundary nodes m to n (counterclockwise).
EZS s	Equal space in the z-direction boundary nodes on side s.
MC n	Check master nodes of slideline n and put any nodes that have penetrated through the slave surface back on the slave surface.
MD n	Dekink master side of slideline n. After using this command, the SC or MC command is sometimes advisable.
MN n	Display slideline n with master node numbers.
SC n	Check slave nodes of slideline n and put any nodes that have penetrated through the master surface back on the master surface.
SD n	Dekink slave side of slideline n; after using this command, the SC or MC command is sometimes advisable.
SLN m n	Equal space boundary nodes between nodes m to n on a straight line connecting node m to n.
SLNS n	Equal space boundary nodes along side n on a straight line connecting the corner nodes.
SN n	Display slideline n with slave node numbers.
VS m n r	Vary the spacing of boundary nodes m to n such that r is the ratio of the first segment length to the last segment length.
VSS s r	Vary the spacing of boundary nodes on side s such that r is the ratio of the first segment length to the last segment length.

MAZE Line Definitions

B	Determine boundary nodes and sides of material n and display boundary with nodes and side numbers. See command “M”.
LD n k l	Line definition n for MAZE includes boundary nodes k to l
LDS n l	Line definition n for MAZE consists of side number l.
M n	Material n is active for the boundary command B.

Calculation Graphics Display Control Parameters

MOLP	Overlay the mesh on the contour, fringe, principal stress, and principal strain plots. Retyping "MOLP" turns this option off.
NLOC	Do not plot letters on contour lines.
NUMCON n	Plot n contour levels. The default is 9.
PLOC	Plot letters on contour lines to identify their levels (default).
RANGE r_1 r_2	Set the range of levels to be between r_1 and r_2 instead of in the range chosen automatically by LS-DYNA. To deactivate this command, type RANGE 0 0.

Calculation Graphics Display

CONTOUR c n m_1 m_2...m_n	Contour component number c on n materials including materials m_1, m_2, ..., m_n. If n is zero, only the outline of material m_1 with contours is plotted. Component numbers are given in Table M.1.
FRINGE c n m_1 m_2...m_n	Fringe component number c on n materials including m_1, m_2,...,m_n. If n is zero, only the outline of material m_1 with contours is plotted. Component numbers are given in Table M.1.
IFD n	Begin definition of interface n. If interface n has been previously defined, this command has the effect of destroying the old definition.
IFN l m	Include boundary nodes l to m (counterclockwise) in the interface definition. This command must follow the "B" command.
IFP c m	Plot component c of interface m. Component numbers are given in Table M.2.
IFS m	Include side m in the interface definition. Side m is defined for material n by the "B" command.
IFVA r_c z_c	Plot the angular location of the interface based on the center point (r_c,z_c) along the abscissa. Positive angles are measured counterclockwise from the y-axis.
IFVS	Plot the distance along the interface from the first interface node along the abscissa (default).

LINE c n m_1 m_2...m_n	Plot variation of component c along line defined with the "NLDF", "PLDF", "NSDF", or the "NSSDF" commands given below. In determining variation, consider n materials including material number m_1, m_2,...m_n.
NCOL n	Number of colors in fringe plots is n. The default value for n is 6 which includes colors magenta, blue, cyan, green, yellow, and red. An alternative value for n is 5 which eliminates the minimum value magenta.
NLDF n n_1 n_2...n_3	Define line for "LINE" command using n nodes including node numbers n_1, n_2,...n_n. This line moves with the nodes.
NSDF m	Define line for "LINE" command as side m. Side m is defined for material n by the "B" command.
NSSDF l m	Define line for "LINE" command and that includes boundary nodes l to m (counterclockwise) in the interface definitions. This command must follow the "B" command.
PLDF n r_1 z_1...r_n z_n	Define line for "LINE" command using n coordinate pairs (r_1,z_1), (r_2,z_2), ...(r_n,z_n). This line is fixed in space.
PRIN c n m_1 m_2...m_n	Plot lines of principal stress and strain in the yz plane on n materials including materials m_1, m_2,...,m_n. If n is zero, only the outline of material m_1 is plotted. The lines are plotted in the principal stress and strain directions. Permissible component numbers in Table M.1 include 0, 5, 6, 100, 105, 106,...,etc. Orthogonal lines of both maximum and minimum stress are plotted if components 0, 100, 200, etc. are specified.
PROFILE c n m_1 m_2...m_n	Plot component c versus element number for n materials including materials m_1, m_2,...,m_n. If n is ∅ then component c is plotted for all elements. Component numbers are given in Table M.1.
VECTOR c n m_1 m_2...m_n	Make a vector plot of component c on n materials including materials m_1, m_2,...,m_n. If n is zero, only the outline of material m_1 with vectors is plotted. Component c may be set to "D" and "V" for vector plots of displacement and velocity, respectively.

No.	Component	No.	Component
1	y	21*	ln (V/Vo) (volumetric strain)
2	z	22*	y-displacement
3	hoop	23*	z-displacement
4	yz	24*	maximum displacement
5	maximum principal	25*	y-velocity, y-heat flux
6	minimum principal	26*	z-velocity, y-heat flux
7	von Mises (Appendix A)	27*	maximum velocity, maximum heat flux
8	pressure or average strain	28	ij normal
9	maximum principal-minimum principal	29	jk normal
10	y minus hoop	30	kl normal
11	maximum shear	31	li normal
12	ij and kl normal (Appendix D)	32	ij shear
13	jk and li normal	33	jk shear
14	ij and kl shear	34	kl shear
15	jk and li shear	35	li shear
16	y-deviatoric	36*	relative volume V/Vo
17	z-deviatoric	37*	Vo/V-1
18	hoop-deviatoric	38*	bulk viscosity, Q
19*	effective plastic strain	39*	P + Q
20*	temperature/internal energy density	40*	density
41*-70*	element history variables		
71*	r-peak acceleration	76*	peak value of min in plane prin. stress
72*	z-peak acceleration	77*	peak value of maximum hoop stress
73*	r-peak velocity	78*	peak value of minimum hoopstress
74*	z-peak velocity	79*	peak value of pressure
75*	peak value of max. in plane prin. stress		

Table M.1. Component numbers for element variables. By adding 100, 200 300, 400, 500 and 600 to the component numbers not followed by an asterisk, component numbers for infinitesimal strains, lagrange strains, almansi strains, strain rates, extensions, and residual strain are obtained. Maximum and minimum principal stresses and strains are in the rz plane. The corresponding hoop quantities must be examined to determine the overall extremum. ij, jk, etc. normal components are normal to the ij, jk, etc side. The peak value database must be flagged on Control Card 4 in columns 6-10 or components 71-79 will not be available for plotting.

No.	Component
1	pressure
2	shear stress
3	normal force
4	tangential force
5	y-force
6	z-force

Table M.2. Component numbers for interface variables. In axisymmetric geometries the force is per radian.

Cursor Commands

DBD a b	Use cursor to define points a and b on boundary. Dekink boundary starting at a, moving counterclockwise, and ending at b.
DCN a b	Use cursor to define points a and b. The node closest to point a will be moved to point b.
DCSN n a	Move nodal point n to point a defined by the cursor.
DCNM a b	Use cursor to define points a and b. The node at point a is given the coordinate at point b.
DER a b	Use cursor to define points a and b on boundary. Equal space nodes in r-direction along boundary starting at a, moving counterclockwise, and ending at b.
DES a b	Use cursor to define points a and b on boundary. Equal space nodes along boundary starting at a, moving counterclockwise, and ending at b.
DEZ a b	Use cursor to define points a and b on boundary. Equal space nodes in z-direction along boundary starting at a, moving counterclockwise, and ending at b.
DTE a b	Use cursor to define points a and b on the diagonal of a window. The element numbers and coordinates of elements lying within the window are typed on the terminal.
DTN a b	Use cursor to define points a and b on the diagonal of a window. The node numbers and coordinates of nodal points lying within the window are typed on the terminal.

DTNC a	Use cursor to define point a. The nodal point number and nodal coordinates of the node lying closest to point a will be printed.
DVS a b r	Use cursor to define points a and b on boundary. Variable space nodes along boundary starting at a, moving counterwise, and ending at b. The ratio of the first segment length to the last segment length is give by r (via terminal).
DZ a b	Use cursor to define points a and b on the diagonal of a window for zooming.
DZOUT a b	Enter two points with the cursor to define the window. The ratio of the current window with the specified window determines the picture size reduction.
DZZ a	Use cursor to define point a and zoom in at this point. The new window is .15 as large as the previous window. The zoom factor can be reset by the crzf command for the .15 default.
DZZO a	Zoom out at point a by enlarging the picture two times.

APPENDIX N: Rigid Body Dummies

The two varieties of rigid body dummies available in LS-DYNA are described in this appendix. These are generated internally by including the appropriate *COMPONENT keyword. A description of the GEBOD dummies begins on this page and the HYBRID III family on page N.7.

GEBOD Dummies

Rigid body dummies can be generated and simulated within LS-DYNA using the keyword *COMPONENT_GEBOD. Physical properties of these dummies draw upon the GEBOD database [Cheng et al. 1994] which represents an extensive measurement program conducted by Wright-Patterson AFB and other agencies. The differential equations governing motion of the dummy are integrated within LS-DYNA separate from the finite element model. Interaction between the dummy and finite element structure is achieved using contact interfaces (see *CONTACT_GEBOD).

The dynamical system representing a dummy is comprised of fifteen rigid bodies (segments) and include: lower torso, middle torso, upper torso, neck, head, upper arms, forearms/hands, upper legs, lower legs, and feet. Ellipsoids are used for visualization and contact purposes. Shown in Figure N.1 is a 50th percentile male dummy generated using the keyword command *COMPONENT_GEBOD_MALE. Note that the ellipsoids representing the shoulders are considered to be part of the upper torso segment and the hands are rigidly attached to the forearms.

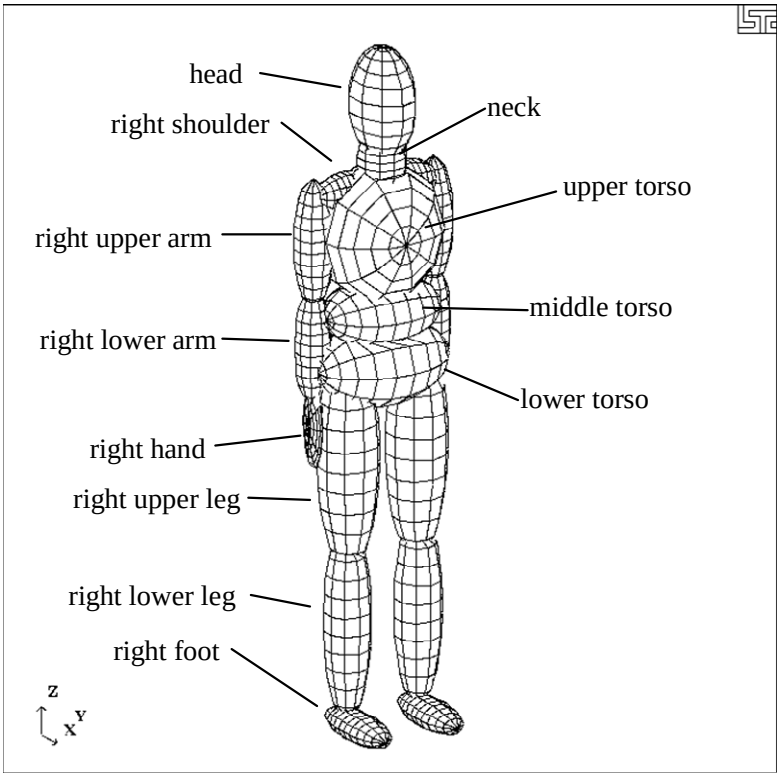


Figure N.1 50th percentile male dummy in the nominal position.

Each of the rigid segments which make up the dummy is connected to its neighbor with a joint which permits various relative motions of the segments. Listed in the Table N.1 are the joints and their applicable degrees of freedom.

Table N.1 Joints and associated degrees of freedom. Local axes are in parentheses.

Joint Name	**Degree(s) of Freedom**		
	1st	2nd	3rd
pelvis	lateral flexion (x)	forward flexion (y)	torsion (z)
waist	lateral flexion (x)	forward flexion (y)	torsion (z)
lower neck	lateral flexion (x)	forward flexion (y)	torsion (z)
upper neck	lateral flexion (x)	forward flexion (y)	torsion (z)
shoulders	abduction-adduction (x)	internal-external rotation (z)	flexion-extension (y)
elbows	flexion-extension (y)	n/a	n/a
hips	abduction-adduction (x)	medial-lateral rotation (z)	flexion-extension (y)
knees	flexion-extension (y)	n/a	n/a
ankles	inversion-eversion (x)	dorsi-plantar flexion (y)	medial-lateral rotation (z)

Orientation of a segment is effected by performing successive right-handed rotations of that segment relative to its parent segment - each rotation corresponds to a joint degree of freedom. These rotations are performed about the local segment axes and the sequence is given in Table N.1. For example, the left upper leg is connected to the lower torso by the left hip joint; the limb is first abducted relative to lower torso, it then undergoes lateral rotation, followed by extension. The remainder of the lower extremity (lower leg and foot) moves with the upper leg during this orientation process.

By default all joints are assigned stiffnesses, viscous characteristics, and stop angles which should give reasonable results in a crash simulation. One or all default values of a joint may be altered by applying the *COMPONENT_GEBOD_JOINT_*OPTION* command to the joint of interest. The default shape of the resistive torque load curve used by all joints is shown in Figure N.2. A scale factor is applied to this curve to obtain the proper stiffness relationship. Listed in Table N.2 are the default values of joint characteristics for dummies of all types and sizes. These values are given in the English system of units; the appropriate units are used if a different system is specified in card 1 of *COMPONENT_GEBOD_*OPTION*.

Table N.2 Default joint characteristics for all dummies.

joint degrees of freedom	load curve scale factor (in-lbf)	damping coef. (in-lbf-s/rad)	low stop angle (degrees)	high stop angle (degrees)	neutral angle (degrees)
pelvis - 1	65000	5.77	-20	20	0
pelvis - 2	65000	5.77	-20	20	0
pelvis - 3	65000	5.77	-5	5	0
waist - 1	65000	5.77	-20	20	0
waist - 2	65000	5.77	-20	20	0
waist - 3	65000	5.77	-35	35	0
lower neck - 1	10000	5.77	-25	25	0
lower neck - 2	10000	5.77	-25	25	0
lower neck - 3	10000	5.77	-35	35	0
upper neck - 1	10000	5.77	-25	25	0
upper neck - 2	10000	5.77	-25	25	0
upper neck - 3	10000	5.77	-35	35	0
l. shoulder - 1	100	5.77	-30	175	0
r. shoulder - 1	100	5.77	-175	30	0
shoulder - 2	100	5.77	-65	65	0
shoulder - 3	100	5.77	-175	60	0
elbow - 1	100	5.77	1	-140	0
l. hip - 1	10000	5.77	-25	70	0
r. hip - 1	10000	5.77	-70	25	0
hip - 2	10000	5.77	-70	70	0
hip - 3	10000	5.77	-140	40	0
knee - 1	100	5.77	-1	120	0
l. ankle - 1	100	5.77	-30	20	0
l. ankle - 1	100	5.77	-20	30	0
ankle - 2	100	5.77	-20	45	0
ankle - 3	100	5.77	-30	30	0

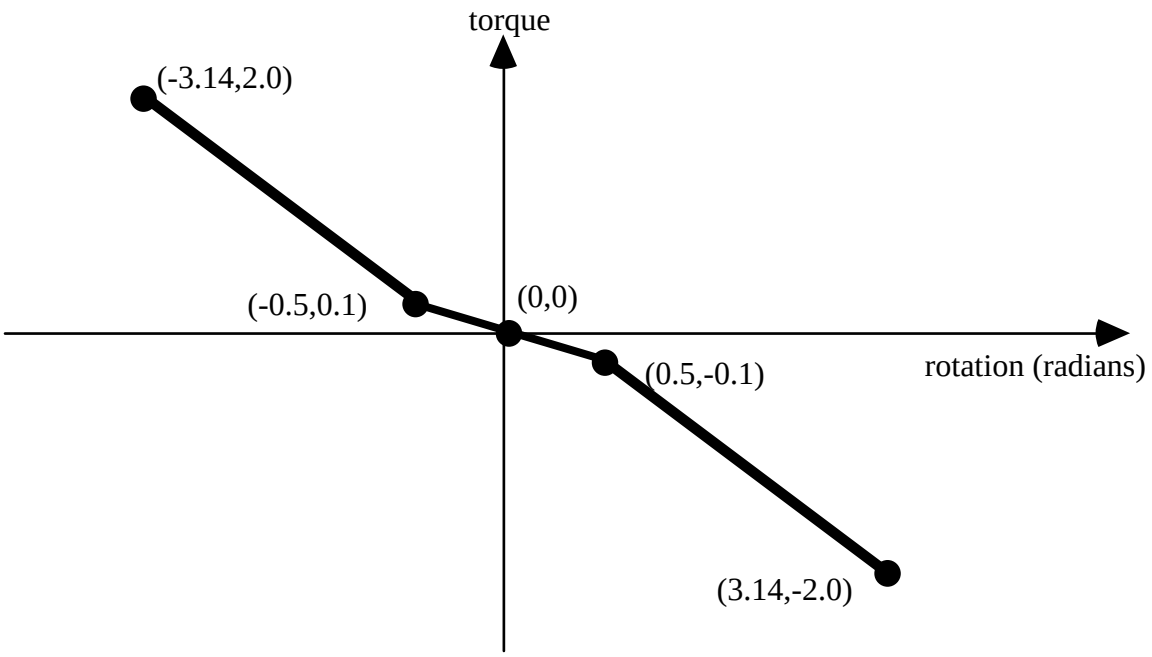


Figure N.2 Characteristic torque curve shape used by all joints.

The dummy depicted in Figure N.1 appears in what is referred to as its "nominal" position. In this position the dummy is standing upright facing in the positive x direction and the toe-to-head direction points in positive z. Additionally, the dummy's hands are at the sides with palms facing inward and the centroid of the lower torso is positioned at the origin of the global coordinate system. Each of the dummy's segments has a local coordinate system attached to it and in the nominal position all of the local axes are aligned with the global axes.

When performing a simulation involving a *COMPONENT_GEBOD dummy, a positioning file named "gebod.*did*" must reside in the directory with the LS-DYNA input file; here the extension *did* is the dummy ID number, see card 1 of *COMPONENT_GEBOD_*OPTION*. The contents of a typical positioning file is shown in Table N.3; it consists of 40 lines formatted as (59a1,e30.0). All of the angular measures are input as degrees, while the lower torso global positions depend on the choice of units in card 1 of *COMPONENT_GEBOD_*OPTION*. Setting all of the values in this file to zero yields the so-called "nominal" position.

Table N.3 Typical contents of a dummy positioning file.

lower torso	centroid global x position		0.0
lower torso	centroid global y position		0.0
lower torso	centroid global z position		0.0
total body	global x rotation		0.0
total body	global y rotation		-20.0
total body	global z rotation		180.0
pelvis	lateral flexion	+ = tilt right	0.0
pelvis	forward flexion	+ = lean fwd	0.0
pelvis	torsion	+ = twist left	0.0
waist	lateral flexion	+ = tilt right	0.0
waist	forward flexion	+ = lean fwd	0.0
waist	torsion	+ = twist left	0.0
lower neck	lateral flexion	+ = tilt right	0.0
lower neck	forward flexion	+ = nod fwd	0.0
lower neck	torsion	+ = twist left	0.0
upper neck	lateral lexion	+ = tilt right	0.0
upper neck	forward flexion	+ = nod fwd	0.0
upper neck	torsion	+ = twist left	0.0
left shoulder	abduction-adduction	+ = abduction	30.0
left shoulder	internal-external rotation	+ = external	-10.0
left shoulder	flexion-extension	- = fwd raise	-40.0
right shoulder	abduction-adduction	- = abduction	-30.0
right shoulder	internal-external rotation	- = external	10.0
right shoulder	flexion-extension	- = fwd raise	-40.0
left elbow	flexion-extension	+ = extension	-60.0
right elbow	flexion-extension	+ = extension	-60.0
left hip	abduction-adduction	+ = abduction	0.0
left hip	medial-lateral rotation	+ = lateral	0.0
left hip	flexion-extension	+ = extension	-80.0
right hip	abduction-adduction	- = abduction	0.0
right hip	medial-lateral rotation	- = lateral	0.0
right hip	flexion-extension	+ = extension	-80.0
left knee	flexion-extension	+ = flexion	50.0
right knee	flexion-extension	+ = flexion	50.0
left ankle	inversion-eversion	+ = eversion	0.0
left ankle	dorsi-plantar flexion	+ = plantar	0.0
left ankle	medial-lateral rotation	+ = lateral	0.0
right ankle	inversion-eversion	- = eversion	0.0
right ankle	dorsi-plantar flexion	+ = plantar	0.0
right ankle	medial-lateral rotation	- = lateral	0.0

In Figure N.3 the 50th percentile male dummy is shown in a seated position and some of its joints are labeled. The file listed in Table N.3 was used to put the dummy into the position shown. Note that the dummy was first brought into general orientation by setting nonzero values for two of the lower torso local rotations. This is accomplished by performing right-handed rotations successively about local axes fixed in the lower torso, the sequence of which follows: the first about local x, next about local y, and the last about local z. The dummy in Figure N.3 was made to pitch backward by setting "total body global y rotation" equal to -20. Setting the "total body global z rotation" equal to 180 caused the dummy to rotate about the global z-axis and face in the -x direction.

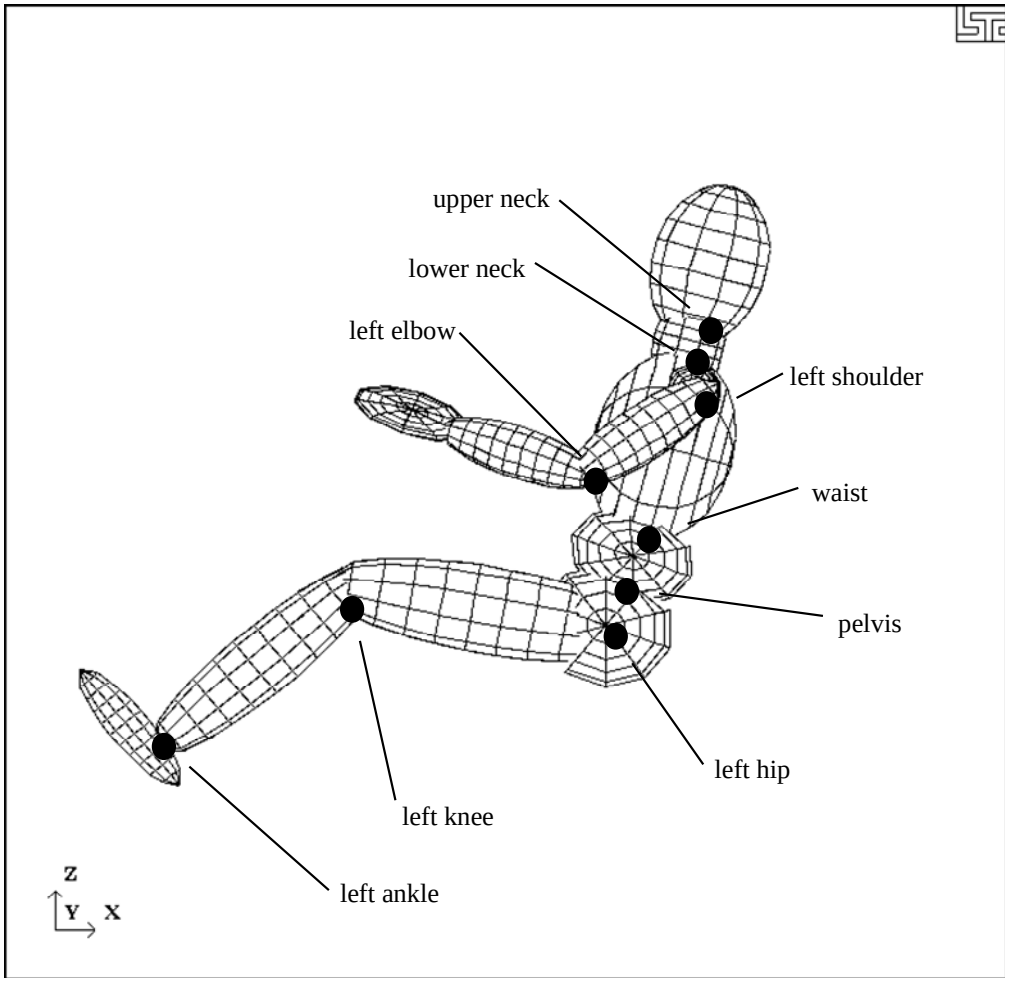


Figure N.3 Dummy seated using the file listed in Table N.3.

HYBRID III Dummies

A listing of applicable joint degrees of freedom of the Hybrid III dummy is given below.

Table N.4 Joints and associated degrees of freedom. Local axes are in parentheses.

Joint Name	Degree(s) of Freedom		
	1st	2nd	3rd
lumbar	flexion (y)	torsion (z)	
lower neck	flexion (y)	torsion (z)	
upper neck	flexion (y)	torsion (z)	
shoulders	flexion-extension (y)	abduction-adduction (x)	n/a
elbows	flexion-extension (y)	n/a	n/a
wrists	flexion-extension (x)	n/a	n/a
hips	abduction-adduction (x)	medial-lateral rotation (z)	flexion-extension (y)
knees	flexion-extension (y)	n/a	n/a
ankles	inversion-eversion (x)	medial-lateral rotation (z)	dorsi-plantar flexion (y)
sternum	translation (x)	rotation (y)	rotation (z)
knee sliders	translation (z)		

Joint springs of the *COMPONENT_HYBRIDIII dummies are formulated in the following manner.

$$T = a_{lo}(q - q_{lo}) + b_{lo}(q - q_{lo})^3 \qquad q \le q_{lo}$$

$$T = a_{hi}(q - q_{hi}) + b_{hi}(q - q_{hi})^3 \qquad q \ge q_{hi}$$

$$T = 0 \qquad q_{lo} < q < q_{hi}$$

where

T is the joint torque

q is the joint generalized coordinate

a_{lo} and b_{lo} are the linear and cubic coefficients, respectively, for the low regime

a_{hi} and b_{hi} are the linear and cubic coefficients, respectively, for the high regime

q_{lo} and q_{hi} are the activation values for the low and high regimes, respectively

APPENDIX O: LS-DYNA MPP User Guide

This is a short user's guide for the MPP version of LS-DYNA. For a general guide to the capabilities of LS-DYNA and a detailed description of the input, consult the LS-DYNA User's Manual. If you have questions about this guide, find errors or omissions in it, please email manual@lstc.com.

Supported Features
The only input formats currently supported are 920 and later, including keyword. Models in any of the older formats will need to be converted to one of these input formats before they can be run with the current version of LS-DYNA for massively parallel processors, mpp.

The large majority of LS-DYNA options are available on MPP computers. Those that are not supported are being systematically added. Unless otherwise noted here, all the options of LS-DYNA version 93x are supported by MPP/LS-DYNA.

Here is the list of **unsupported** features:

- *BOUNDARY_THERMAL_WELD
- *BOUNDARY_USA_SURFACE
- *CONTACT_1D
- *DATABASE_AVS
- *DATABASE_MOVIE
- *DATABASE_MPGS
- *DATABASE_TRACER
- *DATABASE_BINARY_XTFILE
- *INTERFACE_JOY
- *LOAD_SUPERELASTIC_*OPTION*
- *USER
- *TERMINATION_NODE

Contact Interfaces
MPP/LS-DYNA uses a completely redesigned, highly parallel contact algorithm. The contact options currently **unsupported** include:

- *CONTACT_TIED with birth time
- *CONTACT_FORCE_TRANSDUCER_CONSTRAINT

Because these options are all supported via the new, parallel contact algorithms, slight differences in results may be observed as compared to the serial and SMP versions of LS-DYNA. Work has been done to minimize these differences, but they may still be evident in some models.

For each of the supported CONTACT_ control cards, there is an optional string _MPP which can be appended to the end. Adding these characters triggers the reading of a new control card immediately following (but after the TITLE card, if any). This card contains 5 integer parameters in I10 format. The parameters are:

trackpen
If 1, any initial penetrations for slave nodes are compensated for in the contact algorithm. No nodes are moved to eliminate penetrations, and no initial penetration checking is performed. The algorithm detects these penetrations and allows for them in computing forces, so excessively large forces are avoided. As the slave node moves in such a way as to reduce or eliminate the penetration, the full contact distance/material thickness is imposed. Use of this option is encouraged as it can greatly help stability, particularly in models with many initial penetrations. By default this option is disabled.

bucket
Bucket sorting frequency for this contact interface

lcbucket
Load curve giving bucket sort frequency as a function of simulation time. Currently this option is not supported by any of the contact algorithms.

nseg2track
Number of contact segments to track for each slave node

inititer
Number of iterations for initial penetration checking

The defaults for each are taken from the corresponding options in the pfile (described below). For example, if you had the control card:

```
*CONTACT_SINGLE_SURFACE_TITLE
This is my title card
```

you could change this to

```
*CONTACT_SINGLE_SURFACE_TITLE_MPP
This is my title card
     1
```

to turn on the initial penetration tracking option. The serial/SMP code will ignore these options.

Output Files and Post-Processing

For performance reasons, many of the ASCII output files normally created by LS-DYNA have been combined into a new binary format used by MPP/LS-DYNA. There is a post-processing program l2a, which reads this binary database of files and produces as output the corresponding ASCII files. The new binary files will be created in the directory specified as the global directory in the pfile (See section pfile). The files (up to one per processor) are named binout.*nnnn*, where *nnnn* is replaced by the four-digit processor number. To convert these files to ASCII simply feed them to the l2a program like this:

l2a binout*

LS-PREPOST is able to read the binout files directly, so conversion is not required, it is provided for backward compatibility.

The **supported** ASCII files are:

- *DATABASE_SECFORC
- *DATABASE_RWFORC
- *DATABASE_NODOUT
- *DATABASE_NODOUTHF
- *DATABASE_ELOUT
- *DATABASE_GLSTAT
- *DATABASE_DEFORC
- *DATABASE_MATSUM
- *DATABASE_NCFORC
- *DATABASE_RCFORC
- *DATABASE_SPCFORC
- *DATABASE_SWFORC
- *DATABASE_DEFGEO
- *DATABASE_ABSTAT
- *DATABASE_NODOFR
- *DATABASE_BNDOUT
- *DATABASE_GCEOUT
- *DATABASE_RBDOUT
- *DATABASE_SLEOUT
- *DATAGASE_JNTFORC
- *DATABASE_SBTOUT
- *DATABASE_SPHOUT
- *DATABASE_TPRINT

Some of the normal LS-DYNA files will have corresponding collections of files produced by MPP/LS-DYNA, with one per processor. These include the d3dump files (new names = d3dump.*nnnn*), the messag files (now mes*nnnn*) and others. Most of these will be found in the local directory specified in the pfile.

The format of the d3plot file has not been changed. It will be created in the global directory, and can be directly handled with your current graphics post-processor.

Parallel Specific Options
There are a few new command line options that are specific to the MPP version of LS-DYNA.

In the serial and SMP versions of LS-DYNA, the amount of memory required to run the problem can be specified on the command line using the keyword *memory=XXX*, where *XXX* is the number of words of memory to be allocated. For the MPP code, this will result in each processor allocating *XXX* words of memory. If pre-decomposition has not been performed, one processor must perform the decomposition of the problem. This can require substantially more memory than will be required once execution has started. For this reason, there is a second memory command line option, *memory2=YYY*. If used together with the memory keyword, the decomposing processor will allocate *XXX* words of memory, and all other processors will allocate *YYY* words of memory.

For example, in order to run a 250,000 element crash problem on 4 processors, you might need *memory=80m* and *memory2=20m*. To run the same problem on 16 processors, you still need *memory=80m*, but can set *memory2=6m*. The value for *memory2* drops nearly linearly with the number of processors used to run the program, which works well for shared-memory systems.

The full deck restart capability is supported by the MPP version of LS-DYNA, but in a manner slightly different than the SMP code. Each time a restart dump file is written, a separate restart file is also written with the base name D3FULL. For example, when the third restart file d3dump03 is written (one for each processor, d3dump03.0000, d3dump03.0001, etc), there is also a single file written named d3full03. This file is required in order to do a full deck restart and the d3dump files are not used in this case by the MPP code. In order to perform a full deck restart with the MPP code, you first must prepare a full deck restart input file as for the serial/SMP version. Then, instead of giving the command line option *r=d3dump03* you would use the special option *n=d3full03*. The presence of this command line option tells the MPP code that this is a restart, not a new problem, and that the file d3full03 contains the geometry and stress data carried over from the previous run.

PFILE
There is a new command line option: p=pfile. pfile contains MPP specific parameters that affect the execution of the program. The file is split into sections, with several options in each section. Currently, these sections: **directory**, **decomposition**, **contact**, and **general** are available. First, here is a sample pfile:

```
directory {
 global rundir
 local /tmp/rundir
 }
 contact {
 inititer 3
 }
```

The file is case insensitive and free format input. The sections and options currently supported are:

- **directory**. Holds directory specific options

transfer_files
If this keyword is given, then processor 0 will write all output and restart files to the **global** directory (see "global" below), and scratch files to the **local** directory. All other processors will write all data to their **local** directory. At normal termination, all restart and output files will be copied from the processor specific **local** directories to the **global** directory. Also, if this is a restart from a dump file, the dump files will be distributed to the processors from the **global** directory. With this option enabled, there is no need for the processors to have shared access to a single disk for output – all files will be transferred as needed to and from the **global** directory.
Default = disabled.

global *path*
Path to a directory where program output should be written. If **transfer_files** is not given, this directory nees to be accessible to all processors – otherwise it is only accessed by processor 0. This directory will be created if necessary.
Default = current working directory

local *path*
Path to a processor specific local directory for scratch files. This directory will be created if necessary. This should be a local disk on each processor, for performance reasons.
Default = global path

rmlocal
If this keyword is given and **transfer_files** is active, the program attempts to clean up the **local** directories on each processor. In particular, it deletes files that are successfully transferred back to the **global** directory, and removes the **local** directory if it was created. It will not delete any files if there is a failure during file copying, nor will it delete directories it did not create.
Default = disabled

repository *path*
Path to a safe directory accessible from processor 0. This directory will be created if necessary. This is intended to be used as a safekeeping/backup of files during execution

and should only be used if **transfer_files** is also given. If this directory is specified then the following actions occur:

- At program start up, any required files (d3dump, binout, etc) that cannot be located in the **global** directory are looked for in the **repository** for copying to the **local** processor directories.
- Important output files (d3dump, runrsf, d3plot, binout and others) are synchronized to the repository regularly. That is, every time one of these files is updated on the node local or the global directories, a synchronized copy is updated in the repository.

The intension is that the repository be on a redundant disk, such as NAS, to allow restarting the problem if a hardware failure should occur on the machine running the problem. It must be noted that some performance penalty must be paid for the extra communication and I/O. Effort has been made to minimize this overhead, but this option is not recommended for general use.

Default = unspecified

- **decomposition** Holds decomposition specific options

file *filename*

The name of the file that holds the decomposition information. This file will be created in the current working directory if it does not exist. If the filename does not end with the extension *.pre* then this extension is added. If this option is not specified, MPP/LS-DYNA will perform the decomposition.

Default = None

numproc *n*

The problem will be decomposed for n processors. If $n > 1$ and you are running on 1 processor, or if the number of processors you are running on does not evenly divide *n*, then execution terminates immediately after decomposition. Otherwise, the decomposition file is written and execution continues. For a decomposition only run, both numproc and file should be specified.

Default = the number of processors you are running on.

method *name*

Currently, there are two decomposition methods supported, namely *rcb* and *greedy*. Method *rcb* is Recursive Coordinate Bisection. Method *greedy* is a simple neighborhood expansion algorithm. The impact on overall runtime is problem dependent, but rcb generally gives the best performance.

Default = rcb

region rx ry rz sx sy sz c2r s2r 3vec mat

See the section below on Special Decompositions for details about these decomposition options.

show

If this keyword appears in the decomposition section, the d3plot file is doctored so that the decomposition can be viewed with the post processor. Displaying material 1 will show that portion of the problem assigned to processor 0, and so on. The problem will

not actually be run, but the code will terminate once the initial d3plot state has been written.

rcblog *filename*
This option is ignored unless the decomposition method is RCB. A record is written to the indicated file recording the steps taken during decomposition. This is an ascii file giving each decomposition **region** (see the section on Special Decompositions) and the location of each subdivision for that **region**. This information can be placed in the **decomposition** section of the pfile for a subsequent problem, which will result in a decomposition as similar as possible between the two runs. For example, suppose a simulation is run twice, but the second time with a slightly different mesh. Because of the different meshes the problems will be distributed differently between the processors, resulting in slightly different answers due to roundoff errors. If an rcblog is used, then the resulting decompositions would be as similar as possible.

vspeed
If this option is specified a brief measurement is taken of the performance of each processor by timing a short floating point calculation. The resulting information is used during the decomposition to distribute the problem according to the relative speed of the processors. This might be of some use if the cluster has machines of significantly different speed.

automatic
If this option is given, an attempt is made to automatically determine a reasonable decomposition, primarily based on the initial velocity of nodes in the model. Use of the **show** option is recommended to verify a reasonable decomposition.

aledist
Distribute ALE elements to all processors.

dcmem *n*
It may be in some cases that the memory requirements during the first phase of decomposition are too high. If that is found to be the case (if you get out of memory errors during decomposition phase 1), then this may provide a work around. Specifying a value ***n*** here will cause some routines to process the model in blocks of ***n*** items, when normal processing would read the whole set (of nodes, elements, whatever) all at once. This will reduce memory requirements at the cost of greater communication overhead. Most users will not need this option. Values in the range of 10,000 to 50,000 would be reasonable.

- **contact**

This section has been largely replaced by the _MPP option on the normal contact card. The only remaining useful option here is:

alebkt n
Sets the bucket sort frequency for FSI (fluid structure interaction) to once every **n** cycles. **default = 50**

- **general** Holds general options

nodump
If this keyword appears, all restart dump file writing will be suppressed: d3dump, runrsf, and d3full files will not be written.

nofull
If this keyword appears, writing of d3full (full deck restart) files will be suppressed.

nod3dump
If this keyword appears, writing of d3dump and runrsf files will be suppressed.

runrsfonly
If this keyword appears, writing of d3dump files will not occur – runrsf files will be witten instead. Any time a d3dump OR runrsf file would normally be written, a runrsf file will be written.

nofail
If this keyword appears, the check for failed elements in the contact routines will be skipped. This can improve efficiency if you do not have element failure in the model.

swapbytes
If this keyword appears, the d3plot and interface component analysis files are written in swapped byte ordering.

nobeamout
Generally, whenever a beam, shell, or solid element fails, and element failure report is written to the d3hsp and message files. This can generate a lot of output. If this keyword appears, the element failure report is suppressed.

Special Decompositions
These options appear in the "decomposition" section of the pfile and are only valid if the decomposition method is **rcb**. The rcb decomposition method works by recursively dividing the model in half, each time slicing the current piece of the model perpendicularly to one of the three coordinate axes. It chooses the axis along which the current piece of the model is longest. The result is that it tends to generate cube shaped domains aligned along the coordinate axes. This is inherent in the algorithm, but is often not the behavior desired.

This situation is addressed by providing a set of coordinate transformation functions which are applied to the model before it is decomposed. The resulting deformed geometry is then passed to the decomposition algorithm, and the resulting domains are mapped back to the undeformed model. As a simple example, suppose you wanted rectangular domains aligned along a line in the xy plane, 30 degrees from the x axis, and twice as long along this line as in the other two dimensions. If you applied these transformations:

sx 0.5
rz -30

then you would achieve the desired effect.

Furthermore, it may be desireable for different portions of the model to be decomposed differently. It is now possible to specify different regions of the model to be decomposed with different transformations. The general form for a special decomposition would look like this:

```
decomposition {
 region { <region specifiers> <transformation> <grouping> }
 region { <region specifiers> <transformation> <grouping> }
 <transformation>
}
```

Where the region specifiers are logical combinations of **box, sphere, clinder, parts,** and **silist**. The transformation is a series of **sx, sy, sz, rx, ry, rz, c2r, s2r, 3vec,** and **mat**. The grouping is either **lumped** or empty. The portion of the model falling in the first region will be decomposed according to the given transformation. Any remaining part of the model in the second region will then be treated, and finally anything left over will be decomposed according to the final transformation. Any number of regions may be given, including 0. Any number of transformations may be specified. They are applied to the region in the order given.

The region specifiers are:

box xmin xmax ymin ymax zmin zmax
A box with the given extents.

sphere xc yc zc r
The sphere centered at (**xc,yc,zc**) and having radius **r**. If **r** is negative it is treated as infinite.

cylinder xc yc zc ax ay az r d
A cylinder with center at (**xc,yc,zc**) and radius **r**, extending out in the direction of (**ax,ay,az**) for a distance of **d**. If **d** is 0, the cylinder is infinte in both directions.

parts n1 n2 n3 n4….
All parts whose user id matches one of the given values are included in the region. Any number of values may be given.

silist n1 n2 n3 n4….
All elements involved in a contact interface whose user id matches one of the given values are included in the region.

The transformations available are:

sx t
scale the current *x* coordinates by *t*.

sy t
scale the current *y* coordinates by *t*.

sz t

scale the current *z* coordinates by *t*.

rx t
rotate around the current *x* axis by *t* degrees.

ry t
rotate around the current *y* axis by *t* degrees.

rz t
rotate around the current *z* axis by *t* degrees.

mat m11 m12 m13 m21 m22 m23 m31 m32 m33

transform the coordinates by matrix multiplication:

transformed			original
x		m11 m12 m13	x
y	=	m21 m22 m23	y
z		m31 m32 m33	z

3vec v11 v12 v13 v21 v22 v23 v31 v32 v33

Transform the coordinates by the inverse of the transpose matrix:

original			transformed
x		v11 v21 v31	x
y	=	v12 v22 v32	y
z		v13 v23 v33	z

This appears complicated, but in practice is very intuitive: instead of decomposing into cubes aligned along the coordinate axes, rcb will decompose into parallelipipeds whose edges are aligned with the three vectors (v11, v12, v13), (v21, v22, v23), and (v31, v32, v33). Furthermore, the relative lengths of the edges of the decomposition domains will correspond to the relative lengths of these vectors.

C2R x0 y0 z0 vx1 vy1 vz1 vx2 vy2 vz2

The part is converted into a cylindrical coordinate system with origin at (x0, y0, z0), cylinder axis (vx1, vy1, vz1) and theta=0 along the vector (vx2, vy2, vz2). You can think of this as tearing the model along the (vx2, vy2, vz2) vector and unwrapping it around the (vx1, vy1, vz1) axis. The effect is to create decomposition domains that are "cubes" in cylindrical coordinates: they are portions of cylindrical shells. The actual transformation is:

new (x,y,z) = cylindrical coordinates (r,theta,z)

Knowing the order of the coordinates is important if combining transformations, as in the example below.

S2R x0 y0 z0 vx1 vy1 vz1 vx2 vy2 vz2
Just like the above, but for spherical coordinates. The (vx1,vy1,vz1) vector is the phi=0 axis.

New (x,y,z) = spherical coordinates (rho, theta, phi)

The grouping qualifier is:

lumped
Group all elements in the region on a single processor. If this qualifier is not given, the elements in the region are distributed across all processors.

Examples:

rz 45
will generate domains rotated -45 degrees around the z axis.

C2R 0 0 0 0 0 1 1 0 0
will generate cylindrical shells of domains. They will have their axis along the vector (0,0,1), and will start at the vector (1,0,0) Note that the part will be cut at (1,0,0), so no domains will cross this boundary. If there is a natural boundary or opening in your part, the "theta=0" vector should point through this opening. Note also that if the part is, say, a cylinder 100 units tall and 50 units in radius, after the C2R transformation the part will fit inside the box x=[0,50], y=[0, 2PI), z=[0,100]. In particular, the new y coordinates (theta) will be very small compared to the other coordinate directions. It is therefore likely that every decomposition domain will extend through the complete transformed y direction. This means that each domain will be a shell completely around the original cylinder. If you want to split the domains along radial lines, try this pair of transformations:

C2R 0 0 0 0 0 1 1 0 0
SY 5000
This will do the above C2R, but then scale y by 5000. This will result in the part appearing to be about 30,000 long in the y direction -- long enough that every decomposition domain will divide the part in this (transformed) y direction. The result will be decomposition domains that are radial "wedges" in the original part.

General combinations of transformations can be specified, and they are applied in order:

SX 5 SY .2 RZ 30
will scale x, then y, then rotate.

A more general decomposition might look like:

```
decomposition { rx 45 sz 10
region { parts 1 2 3 4 5 and sphere 0 0 0 200 lumped }
region { box 0 100 –1.e+8 1.e+8 0 500 or sphere 100 0 200 200 rx 20 }
}
```

This would take elements that have user ID 1, 2, 3, 4, or 5 for their part, AND that lie in the sphere of radius 200 centered at (0,0,0), and place them all on one processor.

Then, any remaining elements that lie in the given box OR the sphere of radius 200 centered at (100,0,200) would be rotated 20 degress in x then decomposed across all processors. Finally, anything remaining would be rotated 45 degrees in x, scaled 10 in z, and distributed to all processors. In general, region qualifiers can be combined using the logical operations **and**, **or**, and **not**. Grouping using parentheses is also supported.

Execution of MPP/LS-DYNA

MPP/LS-DYNA runs under a parallel environment which provided by the hardware vendor. The execution of the program therefore varies from machine to machine. On some platforms, command line parameters can be passed directly on the command line. For others, the use of the names file is required. The names file is supported on all systems.

The serial/SMP code supports the use of the SIGINT signal (usually Ctrl-C) to interrupt the execution and prompt for user input, generally referred to as "sense switches." The MPP code also supports this capability. However, on many systems a shell script or front end program (generally "mpirun") is required to start MPI applications. Pressing Ctrl-C on some systems will kill this process, and thus kill the running MPP-DYNA executable. As a workaround, when the MPP code begins execution it creates a file "bg_switch" in the current working directory. This file contains the following single line:

rsh <machine name> kill -INT <PID>

where <machine name> is the hostname of the machine on which the root MPP-DYNA process is running, and <PID> is its process id. (on HP systems, "rsh" is replaced by "remsh"). Thus, simply executing this file will send the appropriate signal.

Here is a simple table to show how to run the program on various platforms. Of course, scripts are often written to mask these differences.

Platform	Execution Command
DEC Alpha	dmpirun –np *n mpp-dyna*
Fujitsu	jobexec –vp *n* –mem *m* *mpp-dyna*
Hitachi	mpirun –np *n mpp-dyna*
HP	*mpp-dyna* –np *n*
IBM	#!/bin./ksh export MP_PROC=*n* export MP_LABELIO=no export MP_EUILIB=us export MPI_EUIDEVICE=css0 poe *mpp-dyna*
NEC	mpirun –np *n mpp-dyna*
SGI	mpirun –np *n mpp-dyna*
Sun	tmrun –np *n mpp-dyna*

Where ***n*** is the number of processors, ***mpp-dyna*** is the name of the MPP/LS-DYNA executable, and ***m*** is the MB of real memory.

APPENDIX P: Implicit Solver

INTRODUCTION

The terms implicit and explicit refer to time integration algorithms. In the explicit approach, internal and external forces are summed at each node point, and a nodal acceleration is computed by dividing by nodal mass. The solution is advanced by integrating this acceleration in time. The maximum time step size is limited by the Courant condition, producing an algorithm which typically requires many relatively inexpensive time steps.

While explicit analysis is well suited to dynamic simulations such as impact and crash, it can become prohibitively expensive to conduct long duration or static analyses. Static problems such as sheet metal springback after forming are one application area for implicit methods.

In the implicit method, a global stiffness matrix is computed, inverted, and applied to the nodal out-of-balance force to obtain a displacement increment. The advantage of this approach is that time step size may be selected by the user. The disadvantage is the large numerical effort required to form, store, and factorize the stiffness matrix. Implicit simulations therefore typically involve a relatively small number of expensive time steps.

The implicit analysis capability was first released in Version 950. Initially targeted at metal forming springback simulation, this new capability allowed static stress analysis. Version 970 adds many additional implicit features, including new element formulations for linear and modal analysis.

For best implicit performance, it is important to provide enough memory to allow the stiffness matrix factorization to run in-core. In most cases, the default memory size must be increased. See the Linear Equation Solver section below.

SETTING UP AN IMPLICIT SIMULATION

The keyword *CONTROL_IMPLICIT_GENERAL is used to activate the implicit method. LS-DYNA can conduct either a linear or a nonlinear implicit analysis. The keyword *CONTROL_IMPLICIT_SOLUTION is used to select between these implicit analysis types. In addition, an implicit eigenvalue analysis can be performed to extract frequencies and mode shapes.

To perform a linear implicit analysis, use the *CONTROL_IMPLICIT_GENERAL keyword to activate the implicit method and to specify the time step size. Enter the termination time using the *CONTROL_TERMINATION keyword. For a single step analysis, select the step size to be equal to the termination time. Use the *CONTROL_IMPLICIT_SOLUTION keyword to request a linear analysis. Select linear element formulations using the *SECTION_SOLID and/or *SECTION_SHELL keywords. For best accuracy, a double precision version of LS-DYNA should be used for linear analysis.

To perform an eigenvalue analysis, use the *CONTROL_IMPLICIT_GENERAL keyword to activate the implicit method and to specify a time step size. Enter the termination time using the *CONTROL_TERMINATION keyword (the time step size and termination time must be nonzero, but will otherwise be ignored as LS-DYNA will presently just compute the eigenvalues and stop.) Use the *CONTROL_IMPLICIT_EIGENVALUE keyword to indicate the desired number of eigenvalues and frequency ranges of interest. For best accuracy, a double precision version of LS-DYNA should be used for eigenvalue analysis.

A nonlinear implicit simulation is typically divided into several steps. In a dynamic simulation, these are *time steps*. In a static simulation, these are *load steps*. Multiple steps may be used to divide the nonlinear behavior into manageable pieces, to obtain results at intermediate stages during the simulation, or perhaps to resolve a particular frequency of motion in dynamic simulations. In each step, an equilibrium geometry is sought which balances internal and external forces in the model. The *nonlinear equation solver* performs an iterative search using one of several Newton based methods. *Convergence* of this iterative process is obtained when norms of displacement and/or energy fall below user-prescribed tolerances.

Control parameters for the nonlinear equation solver are input using the keyword *CONTROL_IMPLICIT_SOLUTION. By default, the progress of the equilibrium search is not shown to the screen. This output can be activated either using the NLPRINT input parameter, or interactively toggled on and off by entering "<ctrl-c> nlprint". The box below shows a typical iteration sequence, where the norms of displacement (du/u) and energy (Ei/E0) are displayed. When these norms are reduced below user prescribed tolerances (default 1.0e-3 and 1.0e-2, respectively), equilibrium is reached within sufficient accuracy, the iteration process is said to have *converged*, and the solution proceeds to the next time step.

```
BEGIN time step     3

=========================================================

               time =  1.50000E-01

  current step size =  5.00000E-02

Iteration:   1     *|du|/|u| =  3.4483847E-01     *Ei/E0 =  1.0000000E+00

Iteration:   2     *|du|/|u| =  1.7706435E-01     *Ei/E0 =  2.9395439E-01

Iteration:   3     *|du|/|u| =  1.6631174E-03     *Ei/E0 =  3.7030904E-02

Iteration:   4     *|du|/|u| =  9.7088516E-05     *Ei/E0 =  9.6749731E-08
```

A typical print-out showing the progress of the nonlinear equation solver. By default, the progress of the equilibrium search is not shown to the screen. This output can be activated either using the NLPRINT input parameter, or interactively toggled on and off by entering: "<ctrl-c> nlprint".

LINEAR EQUATION SOLVER

Within each equilibrium iteration, a linear system of equations of the form $\mathbf{K}\Delta\mathbf{u} = \mathbf{R}$ must be solved. To do this, the stiffness matrix $\mathbf{K}$ is inverted and applied to the out-of-balance load or residual $\mathbf{R}$, yielding a displacement increment $\Delta\mathbf{u}$. Storing and solving this linear system represents a large portion of the memory and CPU costs of an implicit analysis.

Control parameters for solving the linear system $\mathbf{K}\Delta\mathbf{u} = \mathbf{R}$ are input using the keyword *CONTROL_IMPLICIT_SOLVER. Several different linear equation solvers are available, including direct (Gaussian elimination) and iterative (conjugate gradient, Lanczos) methods. A sparse storage scheme is used to minimize memory requirements, which are still often substantial. Two options are available for matrix reordering, allowing nodes and elements to be numbered arbitrarily by the user.

It is very important to allow enough memory for the stiffness matrix factorization to run incore. Although the direct solvers can run out-of-core, using disk files for scratch space, this can slow performance by 100x or more. To view memory requirements for a particular model, select LPRINT=1 on *CONTROL_IMPLICIT_SOLVER, or interactively type "<ctrl-c> lprint". Summary information will be printed to the screen and messag files. Use the command line option "memory=..." to increase memory until this summary reports that the TOTAL AVAILABLE memory is large enough that the solver runs "INCORE". The memory size may also be specified on *KEYWORD.

NONLINEAR EQUATION SOLVER

Several different nonlinear equation solvers are available for finding equilibrium within each step. All are iterative in nature. In the *full Newton method*, a new stiffness matrix is formed and inverted each equilibrium iteration. This is the most costly method, but can require fewer iterations to reach equilibrium. In the *modified Newton method*, several iterations are performed using the same stiffness matrix. After each iteration, the geometry is updated using $\Delta\mathbf{u}$ and a new $\mathbf{R}$ is computed. This approach reduces cost by avoiding some forming and factoring of the stiffness matrix $\mathbf{K}$, but usually requires more iterations to reach equilibrium.

The default nonlinear equation solver is the BFGS solver, which uses a *quasi-Newton method*. In this method, the inverted stiffness matrix $\mathbf{K}$ is used for several iterations, but is improved after each iteration using an inexpensive rank two update. If convergence is not reached after 10 iterations, or if *divergence* (increasing $\mathbf{R}$) is detected, then a new stiffness matrix is automatically formed and inverted. This hybrid method combines the efficiency of the modified Newton method with the reliability of the full Newton method. The number of iterations between stiffness matrix reformations is a user input, defaulting to 10. If a value of one is chosen, then the full Newton method is recovered.

```
   BEGIN time step     1
============================================================
               time =  1.00000E+00
  current step size =  1.00000E+00

Iteration:   1     *|du|/|u| =  2.5517753E+00     *Ei/E0 =  1.0000000E+00

DIVERGENCE (increasing residual norm) detected:
    |{Fe}-{Fi}| ( 7.5426269E+03) exceeds |{Fe}| ( 5.0000000E+00)
automatically REFORMING stiffness matrix...

Iteration:   2     *|du|/|u| =  6.0812935E-01     *Ei/E0 =  4.0526413E-01
Iteration:   4     *|du|/|u| =  1.0974191E-02     *Ei/E0 =  2.3907781E-04
Iteration:   5     *|du|/|u| =  1.0978787E-02     *Ei/E0 =  1.7910795E-04
Iteration:   6     *|du|/|u| =  4.2201181E-03     *Ei/E0 =  4.2557768E-05
Iteration:   7     *|du|/|u| =  4.1142219E-03     *Ei/E0 =  3.0658711E-05
Iteration:   8     *|du|/|u| =  1.9794757E-03     *Ei/E0 =  9.1215551E-06
Iteration:   9     *|du|/|u| =  1.7957653E-03     *Ei/E0 =  6.1669480E-06
Iteration:  10     *|du|/|u| =  1.2022830E-03     *Ei/E0 =  2.9031284E-06

ITERATION LIMIT reached, automatically REFORMING stiffness matrix...

Iteration:  11     *|du|/|u| =  5.4011414E-04     *Ei/E0 =  1.0553019E-06
```

The print-out above shows typical behavior of the default BFGS nonlinear equation solver. Two automatic stiffness reformations are performed, initially due to divergence, and later when the default limit of 10 iterations is exceeded. By default, the progress of the equilibrium search is not shown to the screen. This output can be activated either using the NLPRINT input parameter, or interactively toggled on and off by entering "<ctrl-c> nlprint".

$$\mathbf{K}_{n+1}^{-1} = \left(\mathbf{I} + \mathbf{w}\mathbf{v}^T\right)\mathbf{K}_n^{-1}\left(\mathbf{I} + \mathbf{v}\mathbf{w}^T\right)$$

The BFGS update: A new stiffness matrix inverse is approximated by the old stiffness matrix inverse, and the outer product of two carefully chosen vectors.

ELEMENT FORMULATIONS FOR IMPLICIT ANALYSIS

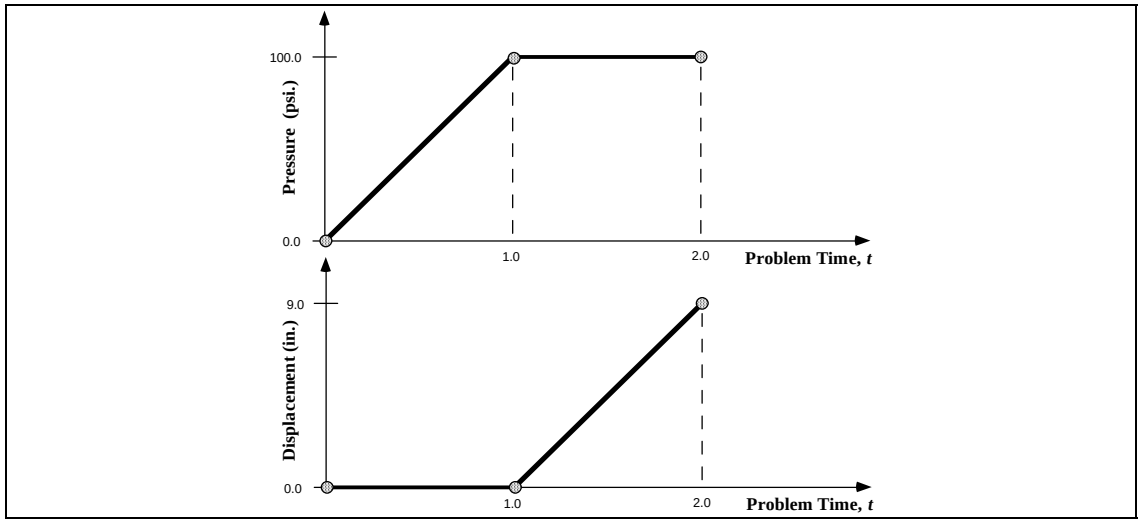
The default element formulations in LS-DYNA are highly efficient, using single point integration. For implicit analysis it is generally more effective to use more expensive element formulations which are less susceptible to hourglass instability. The Hughes-Liu brick element #2 and shell element #6, and the fast shell #16 are good choices for implicit analysis. Stiffness forms of hourglass control are recommended, with hourglass type #6 required for use with implicit solid elements.

APPLYING LOADS DURING IMPLICIT ANALYSIS

Loading is applied using the same keywords as in explicit analysis. Load curves are used to control the magnitude of each load as the simulation proceeds. Typically, the magnitude of each load begins at zero, and is increased to its full value at the end of the last step in the simulation. In this case, the load curve may be defined using only two points.

For example, consider a static analysis where a pressure of 100 psi. is to be applied in 4 steps. Since the analysis is static, the step size can be chosen arbitrarily. For convenience, choose a step size of 0.25, giving a termination time of 1.0. For this problem, the load curve has only two points: (0.0, 0.0) and (1.0, 100.0). LS-DYNA will automatically use linear interpolation to determine the load magnitude at each of the intermediate steps.

In a more complex example, consider a static problem with two types of loading. First, a static pressure of 100 psi. is to be applied, followed by a prescribed displacement of 9 inches. Two load curves are used for this problem, one to control the pressure, and one for the displacement, as shown below. Notice that the displacement is prescribed to be zero while the pressure is applied, then the pressure is held constant while the displacement is applied.



Load curves for a static simulation with two loading events. For convenience in this static simulation, the pressure loading is specified to begin at time 0.0 and end at time 1.0, and the displacement begins at time 1.0 and ends at time 2.0.

AUTOMATIC TIME STEP SIZE CONTROL

In the most simple multi-step nonlinear implicit analysis, the user specifies the *termination time* using the *CONTROL_TERMINATION keyword, and the *time step size* using the *CONTROL_IMPLICIT_GENERAL keyword, and each step is the same size. But for many simulations, the degree of nonlinearity varies through the course of the analysis. In this case the step size should ideally be varied such that solving for equilibrium in each step is equally difficult. This is accomplished by invoking automatic time step control, using the *CONTROL_IMPLICIT_AUTO keyword.

There are two advantages to using automatic time step control. First, the time step size is automatically increased and/or decreased in response to the nonlinearity of the analysis. Nonlinearity is measured simply by the number of iterations required to reach equilibrium. An additional advantage is that if the equilibrium search fails during a time step, LS-DYNA does not terminate. Instead, the step is automatically repeated using a different step size. This process of backing up and retrying difficult steps lends much persistence to the analysis, and is often the only procedure for solving highly nonlinear problems short of adjusting the step size manually.

The input parameters for automatic time step control allow specification of the *optimum number of equilibrium iterations per step*. This indicates how hard LS-DYNA should work in each time step. If equilibrium is reached in fewer than optimum iterations, the size of the next step is increased, and likewise if the equilibrium search requires more than the optimum number of iterations, then the next step size is decreased. Minimum and maximum limits for step size are also input.

IMPLICIT STRESS INITIALIZATION

A common application of the implicit method is to perform static stress initialization for an explicit dynamic calculation. This can be done using two individual calculations, or by switching methods during a calculation. In the first approach, the keyword *INTERFACE_SPRINGBACK_LSDYNA is used to generate a "dynain" output file at the end of the simulation. This file is written in keyword format at the end of the simulation, and contains *NODE, *ELEMENT, and *INITIAL_STRESS data The dynain file can be included into a second input deck to initialize the explicit dynamic analysis.

LS-DYNA can switch "on-the-fly" between the implicit and explicit methods. To use this feature, define a curve which indicates which formulation to use as a function of simulation time. Formulation switching incurs no overhead, and may be performed several times during a simulation. See the IMFLAG parameter on the *CONTROL_IMPLICIT_GENERAL keyword for more information.

TROUBLESHOOTING CONVERGENCE PROBLEMS

Convergence of the nonlinear equilibrium iteration process presents one of the greatest challenges to using the implicit mode of LS-DYNA. Below are some useful troubleshooting approaches:

Eigenvalue Analysis

Many convergence problems in static implicit analysis are caused by unconstrained rigid body modes. These are created when an insufficient number of constraints are applied to the model, or when individual model parts are left disconnected. Eigenvalue analysis is an excellent diagnostic tool to check for these problems.

To perform an eigenvalue analysis, simply add the *CONTROL_IMPLICIT_EIGENVALUE keyword to an implicit input deck. Use the first parameter NEIGV=20 to compute the lowest 20 modes. Then view the frequencies in the output text file “eigout” and animate the mode shapes

in the binary output file d3eigv using LS-PREPOST. Look for frequencies which are nearly zero. Add constraints as necessary to eliminate unconstrained motion.

D3ITER Plot Database

To diagnose convergence trouble which develops in the middle of a simulation, get a picture of the deformed mesh. Adjust the d3plot output interval to produce an output state after every step leading up to the problematic time.

An additional binary plot database named “d3iter” is available which shows the deformed mesh during each equilibrium iteration. This output is activated and de-activated interactively by entering "<ctrl-c> iteration". View this database using LS-PREPOST. Note that stress data is not included. Frequently the problem will become obvious, especially as deformation is magnified.

Prescribed Motion with Death Time

A common static analysis problem occurs when small contact gaps exist between parts at time=0. An example is a load-driven punch which deforms a panel, with a small initial contact gap. This creates instantaneous unconstrained rigid body modes until contact between parts is established. (These modes will be obvious in an eigenvalue analysis, as described above.) To overcome this problem, apply a prescribed motion boundary condition to move the parts into contact. Once contact is established, use the optional death time to “kill” the prescribed motion, and allow the applied force or pressure to provide further loading. Monitor reaction forces from the prescribed motion, and adjust the applied loads to match reasonably well at the death time.

APPENDIX Q: User Defined Weld Failure

The addition of a user weld failure subroutine into LS-DYNA is relatively simple. The UWELDFAIL subroutine is called every time step when OPT=2 is specified in MAT_SPOTWELD**.** As data, the identification number for the spotweld material, six constants specified in the input by the locations NRR through MTT, the radius of the cross section of the spotwelds, the current time, and the current values of the resultants for the spotwelds, which are stored in array STRR, are passed to the subroutine. The subroutine loops over the welds from LFT through LLT, and sets the values of the failure flag array FLAG.

```
      SUBROUTINE UWELDFAIL(IDWELD,STRR,FAIL,FIBL,CM,TT,LFT,LLT)
C***********************************************************************
C|  LIVERMORE SOFTWARE TECHNOLOGY CORPORATION  (LSTC)                   |
C|  ------------------------------------------------------------------  |
C|  COPYRIGHT 2002 JOHN O. HALLQUIST, LSTC                              |
C|  ALL RIGHTS RESERVED                                                 |
C***********************************************************************
C
C***  SPOTWELD FAILURE ROUTINE
C
C***  LOCAL COORDINATES: X IS TANGENT TO BEAM, Y & Z ARE NORMAL
C
C***  VARIABLES
C          IDWELD ---- WELD ID NUMBER
C          STRR ------ STRESS RESULTANTS
C                      (1) AXIAL (X DIRECTION) FORCE
C                      (2) Y SHEAR FORCE
C                      (3) Z SHEAR FORCE
C                      (4) MOMENT ABOUT Z
C                      (5) MOMENT ABOUT Y
C                      (6) TORSIONAL RESULTANT
C          FAIL ------ FAILURE FLAG
C                      =0 NOT FAILED
C                      =1 FAIL ELEMENT
C          FIBL ------ LOCATION (1,*) GIVES THE SPOTWELD DIAMETER
C          CM -------- 6 CONSTANTS SUPPLIED BY USER
C          TT -------- CURRENT SIMULATION TIME
C          LFT,LLT --- DO-LOOP RANGE FOR STRR
C
      DIMENSION IDWELD(*),STRR(6,*),FAIL(*),CM(*),FIBL(5,*)
C
C
      RETURN
      END
```

APPENDIX R: User Defined Cohesive Model

The addition of a user cohesive material subroutine into LS-DYNA is relatively simple. The UMATiC subroutine is called every time step where i ranges from 41 to 50**.** Input for the material model follows the *MAT_USER_DEFINED_MATERIAL definition. The user has the option of providing either a scalar or vectorized subroutine. As discussed in the Remarks for the user-defined material, the first two material parameters are reserved to specify how the density is treated and the number of integration points required for the failure of the element.

The cohesive model calculates the tractions on the mid-surface of the element as a function of the differences of the displacements and velocities of the upper (defined by nodes 5-6-7-8) and lower surfaces (defined by nodes 1-2-3-4). The displacements, velocities, and the calculated tractions are in the local coordinate system of the element, where the first two components of the vectors are in the plane of the mid-surface and the third component is normal to the mid-surface.

A stiffness must also be calculated by the user for the explicit time step calculation in LS-DYNA. This stiffness must provide an upper bound on the stiffness in all three directions.

The material fails at an integration point when `ifail=.true`. For an element to be deleted from the calculation, the number of integration points specified by the second material parameter must fail. If the second parameter is zero, elements cannot fail regardless of the specification of IFAIL in the user-defined material input.

The following example is a vectorized model with two elastic constants and failure:

```
      subroutine umat41c(idpart,cm,lft,llt,fc,dx,dxdt,aux,ek,
     &                   ifail,dt1siz,crv)
      include 'nlqparm'
c
c***  vector cohesive material user model example
c
c***  variables
c          idpart ---- Part ID
c          cm -------- material constants
c          lft,llt --- start and end of block
c          fc -------- components of the cohesive force
c          dx -------- components of the displacement
c          dxdt ------ components of the velocity
c          aux ------- history storage
c          ek -------- max. stiffness/area for time step calculation
c          ifail ----- =.false. not failed
c                      =.true. failed
c          dt1siz ---- time step size
c          crv ------- curve array
c
c***  dx, dxdt, and fc are in the local coordinate system:
c     components 1 and 2 are in the plane of the cohesive surface
c     component 3 is normal to the plane
c
c***  cm storage convention
c     (1) =0 density is per area
c         =1 density is per volume
c     (2) number of integration points for element deletion
c         =0 no deletion
c     (3:48) material model constants
c
      logical ifail
      dimension cm(*),fc(nlq,*),dx(nlq,*),dxdt(nlq,*),
     &          aux(nlq,*),ek(*),ifail(*),dt1siz(*),crv(101,2,*)
```

```
c
      et=cm(3)
      en=cm(4)
      eki=max(et,en)
      fcfail=cm(5)
c
      do i=lft,llt
        fc(i,1)=et*dx(i,1)
        fc(i,2)=et*dx(i,2)
        fc(i,3)=en*dx(i,3)
        ek(i)=eki
        ifail(i)=fc(i,3).gt.fcfail
      enddo
c
      return
      end
```

The second example implements the Tveergard-Hutchinson cohesive model with failure in both the vectorized (UMAT42C) and scalar (UMAT43C) forms. Note the LFT and LLT are passed to the scalar version, however their value is zero.

```
      subroutine umat42c(idpart,params,lft,llt,fTraction,jump_u,dxdt,
     &                   aux,ek,ifail,dt1siz,crv)
      include 'nlqparm'
c
c***  vector cohesive material user model example
c
c***  variables
c          idpart ---- part ID
c          params ---- material constants
c          lft,llt --- start and end of block
c          fTraction - components of the cohesive force
c          jump_u ---- components of the displacement
c          dxdt ------ components of the velocity
c          aux ------- history storage
c          ek -------- max. stiffness/area for time step calculation
c          ifail ----- =.false. not failed
c                      =.true. failed
c          dt1siz ---- time step size
c          crv ------- curve array
c
c***  jump_u, dxdt, and fTraction are in the local coordinate system:
c     components 1 and 2 are in the plane of the cohesive surface
c     component 3 is normal to the plane
c
c***  cm storage convention
c     (1) =0 density is per area
c         =1 density is per volume
c     (2) number of integration points for element deletion
c         =0 no deletion
c     (3:48) material model constants
c
c     Tveergard-Hutchinson model based on:
c     tahoe/src/elements/cohesive_surface/cohesive_models/TvergHutch3DT.cpp

c     the declaration below is processed by the C preprocessor and
c     is real*4 or real*8 depending on whether LS-DYNA is single or double
c     precision
      REAL L,jump_u

      logical ifail
```

```
      dimension params(*),fTraction(nlq,*),jump_u(nlq,*),
     &          dxdt(nlq,*),aux(nlq,*),ek(*),ifail(*),dt1siz(*),
     &          crv(101,2,*)

      fsigma_max=params(3)
      fd_c_n=params(4)
      fd_c_t=params(5)
      fL_1=params(6)
      fL_2=params(7)
      fpenalty=params(8)

      fK=fpenalty*fsigma_max/(fL_1*fd_c_n)

      fac=min(fd_c_n/fd_c_t**2,1./fd_c_n)

      do i=lft,llt
      u_t1 = jump_u(i,1)
      u_t2 = jump_u(i,2)
      u_n = jump_u(i,3)

      r_t1 = u_t1/fd_c_t
      r_t2 = u_t2/fd_c_t
      r_n = u_n/fd_c_n
      L = sqrt(r_t1*r_t1 + r_t2*r_t2 + r_n*r_n)

      if (L .lt. fL_1) then
         sigbyL=fsigma_max/fL_1
      else if (L .lt. fL_2) then
         sigbyL = fsigma_max/L
      else if (L .lt. 1.) then
         sigbyL = fsigma_max*(1. - L)/(1. - fL_2)/L
      else
         sigbyL = 0.0
         ifail(i)=.true.
      endif

      fTraction(i,1) = sigbyL*r_t1*(fd_c_n/fd_c_t)
      fTraction(i,2) = sigbyL*r_t2*(fd_c_n/fd_c_t)
      fTraction(i,3) = sigbyL*r_n

c     penetration
      if (u_n .lt. 0) fTraction(i,3)=fTraction(i,3)+fK*u_n

c     approximate stiffness for time step
      if (u_n .lt. 0) then
        ek(i)=fac*sigbyL+fK
      else
        ek(i)=fac*sigbyL
      endif

      enddo

      return
      end
      subroutine umat43c(idpart,params,lft,llt,fTraction,jump_u,dxdt,
     &                   aux,ek,ifail,dt1siz,crv)
c
c***  scalar cohesive material user model example
c
c***  variables
c          idpart ---- part ID
c          params ---- material constants
```

```
c          lft,llt --- start and end of block
c          fTraction - components of the cohesive force
c          jump_u ---- components of the displacement
c          dxdt ------ components of the velocity
c          aux ------- history storage
c          ek -------- max. stiffness/area for time step calculation
c          ifail ----- =.false. not failed
c                      =.true. failed
c          dt1siz ---- time step size
c          crv ------- curve array
c
c***  jump_u, dxdt, and fTraction are in the local coordinate system:
c     components 1 and 2 are in the plane of the cohesive surface
c     component 3 is normal to the plane
c
c***  cm storage convention
c     (1) =0 density is per area
c         =1 density is per volume
c     (2) number of integration points for element deletion
c         =0 no deletion
c     (3:48) material model constants
c
c     Tveergard-Hutchinson model based on:
c     tahoe/src/elements/cohesive_surface/cohesive_models/TvergHutch3DT.cpp

c     the declaration below is processed by the C preprocessor and
c     is real*4 or real*8 depending on whether LS-DYNA is single or double
c     precision
      REAL L,jump_u

      logical ifail
      dimension params(*),fTraction(*),jump_u(*),
     &          dxdt(*),aux(*),crv(101,2,*)

      fsigma_max=params(3)
      fd_c_n=params(4)
      fd_c_t=params(5)
      fL_1=params(6)
      fL_2=params(7)
      fpenalty=params(8)

      fK=fpenalty*fsigma_max/(fL_1*fd_c_n)

      fac=min(fd_c_n/fd_c_t**2,1./fd_c_n)

      u_t1 = jump_u(1)
      u_t2 = jump_u(2)
      u_n = jump_u(3)

      r_t1 = u_t1/fd_c_t
      r_t2 = u_t2/fd_c_t
      r_n = u_n/fd_c_n
      L = sqrt(r_t1*r_t1 + r_t2*r_t2 + r_n*r_n)

      if (L .lt. fL_1) then
         sigbyL=fsigma_max/fL_1
      else if (L .lt. fL_2) then
         sigbyL = fsigma_max/L
      else if (L .lt. 1.) then
         sigbyL = fsigma_max*(1. - L)/(1. - fL_2)/L
      else
         sigbyL = 0.0
```

```
      ifail=.true.
      endif

      fTraction(1) = sigbyL*r_t1*(fd_c_n/fd_c_t)
      fTraction(2) = sigbyL*r_t2*(fd_c_n/fd_c_t)
      fTraction(3) = sigbyL*r_n

c     penetration
      if (u_n .lt. 0) fTraction(3)=fTraction(3)+fK*u_n

c     approximate stiffness for time step
      if (u_n .lt. 0) then
        ek=fac*sigbyL+fK
      else
        ek=fac*sigbyL
      endif

      return
      end
```

APPENDIX S: User Defined Boundary Flux

A user defined boundary flux interface is provided in LS-DYNA where it is possible to define the thermal heat flux (power per surface area) in or out of a surface segment as an arbitrary function of temperature and history. The user may associate history variables with each individual flux interface and also use load curves.

The user flux interface is invoked using the keyword *BOUNDARY_FLUX_*OPTION*. This is accomplished with the parameter NHISV. When it is defined with a value greater than 0, the user subroutine

```
subroutine usrflux(fl,flp,…)
```

is called to compute the flux (`fl`) defined as heat (energy) per time and per surface area.

Other parameters that are passed to the user flux subroutine include the segment nodal temperatures at the previous (T_0) and current time (T_1), the segment nodal coordinates and the time integration parameter α. Also, the current thermal simulation time t, the time step Δt and average segment temperature (T_α) at time $t+\alpha\Delta t$ is provided together with the curve array for accessing defined load curves in the keyword input file. For computing load curve values, note that load curve IDs need to be transformed to internal numbers or the subroutine crvval should be used, see the appendix on user defined materials for details.

The segment coordinates available in the subroutine are such that the outward normal vector follows the well-known right-hand rule, thus segments corresponding to the lower surface of thick thermal shells are reversed before passed to the subroutine. For shells in general, the segment connectivity should follow the connectivity of the actual shell element to avoid problems.

Optionally, the user may define the derivative of the flux `fl` with respect to the average segment temperature (T_α) at time $t+\alpha\Delta t$, `flp`. This value is used in the nonlinear thermal solver for assembling the correct stiffness matrix and must be set by the user. If possible, it is recommended to use a value that reflects the nonlinearity of the flux model, otherwise the value 0 should be used.

An array of history variables, identical with the input parameters defined in the keyword input file, are passed to the subroutine that can be updated with time or kept constant throughout the simulation. An example of usage would be to integrate the flux with time to keep track of the dissipated energy per surface area in order to simulate the effects of spray cooling in hot-stamping.

```
      subroutine usrflux(fl,flp,x,tnpl,tnl,nodes,
     .     alpha,atime,atemp,dt,time,fhsv,nfhsv,crv)
C*************************************************************
C|  LIVERMORE SOFTWARE TECHNOLOGY CORPORATION  (LSTC)         |
C|  ---------------------------------------------------------|
C|  COPYRIGHT © 2007 JOHN O. HALLQUIST, LSTC                  |
C|  ALL RIGHTS RESERVED                                       |
C*************************************************************
c
c     User subroutine for boundary thermal flux
```

```
c
c     Purpose: To define thermal flux parameter (heat per surface area and
c     time)
c
c     Variables:
c
c     fl              = flux intensity (output)
c     flp             = flux intensity derivative wrt atemp (output)
c     x(3,nodes)      = global segment coordinates (input)
c     tnpl(nodes)     = temperatures at time time (input)
c     tnl(nodes)      = temperatures at time time-dt (input)
c     nodes           = number of nodes in segment (3,4 or 6) (input)
c     alpha           = time integration parameter (input)
c     atime           = time+(alpha-1)*dt
c     atemp           = average segment temperature at time atime
c     dt              = time step size (input)
c     time            = time at which the new temperature is sought (input)
c     fhsv(nfhsv)     = flux history variables (input/output)
c     nfhsv           = number of flux history variables for this segment
c                       (input)
c     crv             = curve array (input)
c
      include 'nlqparm'
      dimension x(3,*),tnpl(*),tnl(*)
      dimension fhsv(*),crv(lq1,2,*)
c
c     Define flux by linear convection
c     that optionally decays (in an ad-hoc way) as power
c     dissipates from surface
c
c     fhsv(1) = convection coefficient
c     fhsv(2) = ambient temperature
c     fhsv(3) = total amount of energy per surface area available
c     fhsv(4) = dissipated energy per surface area at current
c
      hcon=fhsv(1)
      tinf=fhsv(2)
      flin=hcon*(tinf-atemp)
      if (nfhsv.gt.2) then
         q=(1.-fhsv(4)/fhsv(3))/
     .         (1.+.5*dt*flin/fhsv(3))
         flp=-q*hcon
         if (q.gt.1.) then
            q=1.
            flp=-hcon
         elseif (q.lt.0.) then
            q=0.
            flp=0.
         endif
         fl=q*flin
         fhsv(4)=fhsv(4)+dt*.5*fl
         fhsv(4)=min(fhsv(3),fhsv(4))
      else
         fl=flin
         flp=-hcon
      endif
c
      return
      end
```

LS-DYNA®
KEYWORD USER'S MANUAL

VOLUME II
Material Models

May 2010
Version 971 / Rev 5
(Beta)

LIVERMORE SOFTWARE TECHNOLOGY CORPORATION (LSTC)

Corporate Address
Livermore Software Technology Corporation
P. O. Box 712
Livermore, California 94551-0712

Support Addresses
Livermore Software Technology Corporation
7374 Las Positas Road
Livermore, California 94551
Tel: 925-449-2500 ♦ Fax: 925-449-2507
Email: sales@lstc.com
Website: www.lstc.com

Livermore Software Technology Corporation
1740 West Big Beaver Road
Suite 100
Troy, Michigan 48084
Tel: 248-649-4728 ♦ Fax: 248-649-6328

Disclaimer

ISBN 0-9778540-3-5

TABLE OF CONTENTS

TABLE OF CONTENTS

TABLE OF CONTENTS

TABLE OF CONTENTS

*MAT

LS-DYNA has historically referenced each material model by a number. As shown below, a 3-digit numerical designation can still be used, e.g., *MAT_001, and is equivalent to a corresponding descriptive designation, e.g., *MAT_ELASTIC. The two equivalent commands for each material model, one numerical and the other descriptive, are listed below. The numbers in square brackets (see key below) identify the element formulations for which the material model is implemented. The number in the curly brackets, **{*n*}**, indicates the default number of history variables per element integration point that are stored in addition to the 7 history variables which are stored by default. For the type 16 fully integrated shell elements with 2 integration points through the thickness, the total number of history variables is 8 x (*n*+7). For the Belytschko-Tsay type 2 element the number is 2 x (n+7).

An additional option **_TITLE** may be appended to a ***MAT** keyword in which case an additional line is read in 80a format which can be used to describe the material. At present, LS-DYNA does not make use of the title. Inclusion of titles simply gives greater clarity to input decks.

Key to numbers in square brackets

0 - Solids
1H - Hughes-Liu beam
1B - Belytschko resultant beam
1I - Belytschko integrated solid and tubular beams
1T - Truss
1D - Discrete beam
1SW - Spotweld beam
2 - Shells
3 - Thick shells
4 - Special airbag element
5 - SPH element
6 - Acoustic solid
7 - Cohesive solid
8A - Multi-material ALE solid (validated)
8B - Multi-material ALE solid (implemented but not validated[1])

***MAT_ADD_EROSION [2]**

***MAT_ADD_PERMEABILTY**

***MAT_ADD_THERMAL_EXPANSION [2]**

***MAT_NONLOCAL [2]**

***MAT_001: *MAT_ELASTIC [0,1H,1B,1I,1T,2,3,5,8A] {0}**

[1] Error associated with advection inherently leads to state variables that may be inconsistent with nonlinear constitutive routines and thus may lead to nonphysical results, nonconservation of energy, and even numerical instability in some cases. Caution is advised, particularly when using the 2nd tier of material models implemented for ALE multi-material solids (designated by [8B]) which are largely untested as ALE materials.
[2] These three commands do not, by themselves, define a material model but rather can be used in certain cases to supplement material models.

***MAT_001_FLUID:**	***MAT_ELASTIC_FLUID [0,8A] {0}**
***MAT_002:**	***MAT_*OPTION*TROPIC_ELASTIC [0,2,3] {15}**
***MAT_003:**	***MAT_PLASTIC_KINEMATIC [0,1H,1I,1T,2,3,5,8A] {5}**
***MAT_004:**	***MAT_ELASTIC_PLASTIC_THERMAL [0,1H,1T,2,3,8B] {3}**
***MAT_005:**	***MAT_SOIL_AND_FOAM [0,5,8A] {0}**
***MAT_006:**	***MAT_VISCOELASTIC [0,1H,2,5,8B] {19}**
***MAT_007:**	***MAT_BLATZ-KO_RUBBER [0,2,8B] {9}**
***MAT_008:**	***MAT_HIGH_EXPLOSIVE_BURN [0,5,8A] {4}**
***MAT_009:**	***MAT_NULL [0,1,2,5,8A] {3}**
***MAT_010:**	***MAT_ELASTIC_PLASTIC_HYDRO_{*OPTION*} [0,5,8B] {4}**
***MAT_011:**	***MAT_STEINBERG [0,5,8B] {5}**
***MAT_011_LUND:**	***MAT_STEINBERG_LUND [0,5,8B] {5}**
***MAT_012:**	***MAT_ISOTROPIC_ELASTIC_PLASTIC [0,2,3,5,8B] {0}**
***MAT_013:**	***MAT_ISOTROPIC_ELASTIC_FAILURE [0,5,8B] {1}**
***MAT_014:**	***MAT_SOIL_AND_FOAM_FAILURE [0,5,8B] {1}**
***MAT_015:**	***MAT_JOHNSON_COOK [0,2,3,5,8A] {6}**
***MAT_016:**	***MAT_PSEUDO_TENSOR [0,5,8B] {6}**
***MAT_017:**	***MAT_ORIENTED_CRACK [0] {10}**
***MAT_018:**	***MAT_POWER_LAW_PLASTICITY [0,1H,2,3,5,8B] {0}**
***MAT_019:**	***MAT_STRAIN_RATE_DEPENDENT_PLASTICITY [0,2,3,5,8B] {6}**
***MAT_020:**	***MAT_RIGID [0,1H,1B,1T,2,3] {0}**
***MAT_021:**	***MAT_ORTHOTROPIC_THERMAL [0,2,3] {29}**
***MAT_022:**	***MAT_COMPOSITE_DAMAGE [0,2,3] {12}**
***MAT_023:**	***MAT_TEMPERATURE_DEPENDENT_ORTHOTROPIC [0,2,3] {19}**
***MAT_024:**	***MAT_PIECEWISE_LINEAR_PLASTICITY [0,1H,2,3,5,8A] {5}**
***MAT_025:**	***MAT_GEOLOGIC_CAP_MODEL [0,5] {12}**
***MAT_026:**	***MAT_HONEYCOMB [0] {20}**
***MAT_027:**	***MAT_MOONEY-RIVLIN_RUBBER [0,1T,2,8B] {9}**
***MAT_028:**	***MAT_RESULTANT_PLASTICITY [1B,2] {5}**
***MAT_029:**	***MAT_FORCE_LIMITED [1B] {30}**
***MAT_030:**	***MAT_SHAPE_MEMORY [0,2,5] {23}**
***MAT_031:**	***MAT_FRAZER_NASH_RUBBER_MODEL [0,8B] {9}**
***MAT_032:**	***MAT_LAMINATED_GLASS [2,3] {0}**
***MAT_033:**	***MAT_BARLAT_ANISOTROPIC_PLASTICITY [0,2,3] {9}**
***MAT_033_96:**	***MAT_BARLAT_YLD96 [2,3] {9}**
***MAT_034:**	***MAT_FABRIC [4] {17}**
***MAT_035:**	***MAT_PLASTIC_GREEN-NAGHDI_RATE [0,5,8B] {22}**

***MAT_036:**	***MAT_3-PARAMETER_BARLAT [2,3] {7}**
***MAT_037:**	***MAT_TRANSVERSELY_ANISOTROPIC_ELASTIC_PLASTIC [2,3] {9}**
***MAT_038:**	***MAT_BLATZ-KO_FOAM [0,2,8B] {9}**
***MAT_039:**	***MAT_FLD_TRANSVERSELY_ANISOTROPIC [2,3] {6}**
***MAT_040:**	***MAT_NONLINEAR_ORTHOTROPIC [0,2] {17}**
***MAT_041-050:**	***MAT_USER_DEFINED_MATERIAL_MODELS [0,1H,1T,1D,2,3,5,8B] {0}**
***MAT_051:**	***MAT_BAMMAN [0,2,3,5,8B] {8}**
***MAT_052:**	***MAT_BAMMAN_DAMAGE [0,2,3,5,8B] {10}**
***MAT_053:**	***MAT_CLOSED_CELL_FOAM [0,8B] {0}**
***MAT_054-055:**	***MAT_ENHANCED_COMPOSITE_DAMAGE [2] {20}**
***MAT_057:**	***MAT_LOW_DENSITY_FOAM [0,5,8B] {26}**
***MAT_058:**	***MAT_LAMINATED_COMPOSITE_FABRIC [2,3] {15}**
***MAT_059:**	***MAT_COMPOSITE_FAILURE_*OPTION*_MODEL [0,2,5] {22}**
***MAT_060:**	***MAT_ELASTIC_WITH_VISCOSITY [0,2,5,8B] {8}**
***MAT_060C:**	***MAT_ELASTIC_WITH_VISCOSITY_CURVE [0,2,5,8B] {8}**
***MAT_061:**	***MAT_KELVIN-MAXWELL_VISCOELASTIC [0,5,8B] {14}**
***MAT_062:**	***MAT_VISCOUS_FOAM [0,8B] {7}**
***MAT_063:**	***MAT_CRUSHABLE_FOAM [0,5,8B] {8}**
***MAT_064:**	***MAT_RATE_SENSITIVE_POWERLAW_PLASTICITY [0,2,3,5,8B] {30}**
***MAT_065:**	***MAT_MODIFIED_ZERILLI_ARMSTRONG [0,2,3,5,8B] {6}**
***MAT_066:**	***MAT_LINEAR_ELASITC_DISCRETE_BEAM [1D] {8}**
***MAT_067:**	***MAT_NONLINEAR_ELASITC_DISCRETE_BEAM [1D] {14}**
***MAT_068:**	***MAT_NONLINEAR_PLASITC_DISCRETE_BEAM [1D] {25}**
***MAT_069:**	***MAT_SID_DAMPER_DISCRETE_BEAM [1D] {13}**
***MAT_070:**	***MAT_HYDRAULIC_GAS_DAMPER_DISCRETE_BEAM [1D] {8}**
***MAT_071:**	***MAT_CABLE_DISCRETE_BEAM [1D] {8}**
***MAT_072:**	***MAT_CONCRETE_DAMAGE [0,5,8B] {6}**
***MAT_072R3:**	***MAT_CONCRETE_DAMAGE_REL3 [0,5] {6}**
***MAT_073:**	***MAT_LOW_DENSITY_VISCOUS_FOAM [0,8B] {56}**
***MAT_074:**	***MAT_ELASTIC_SPRING_DISCRETE_BEAM [1D] {8}**
***MAT_075:**	***MAT_BILKHU/DUBOIS_FOAM [0,5,8B] {8}**
***MAT_076:**	***MAT_GENERAL_VISCOELASTIC [0,2,5,8B] {53}**
***MAT_077_H:**	***MAT_HYPERELASTIC_RUBBER [0,2,5,8B] {54}**
***MAT_077_O:**	***MAT_OGDEN_RUBBER [0,2,8B] {54}**
***MAT_078:**	***MAT_SOIL_CONCRETE [0,8B] {3}**
***MAT_079:**	***MAT_HYSTERETIC_SOIL [0,5,8B] {77}**
***MAT_080:**	***MAT_RAMBERG-OSGOOD [0,8B] {18}**

***MAT_081:**	***MAT_PLASTICITY_WITH_DAMAGE [0,2,3] {5}**
***MAT_082(_RCDC):**	***MAT_PLASTICITY_WITH_DAMAGE_ORTHO(_RCDC) [0,2,3] {22}**
***MAT_083:**	***MAT_FU_CHANG_FOAM [0,5,8B] {54}**
***MAT_084-085:**	***MAT_WINFRITH_CONCRETE [0,8B] {54}**
***MAT_086:**	***MAT_ORTHOTROPIC_VISCOELASTIC [2,3] {17}**
***MAT_087:**	***MAT_CELLULAR_RUBBER [0,5,8B] {19}**
***MAT_088:**	***MAT_MTS [0,2,3,5,8B] {5}**
***MAT_089:**	***MAT_PLASTICITY_POLYMER [2] {45}**
***MAT_090:**	***MAT_ACOUSTIC [6] {25}**
***MAT_091:**	***MAT_SOFT_TISSUE [0,2] {16}**
***MAT_092:**	***MAT_SOFT_TISSUE_VISCO [0,2] {58}**
***MAT_093:**	***MAT_ELASTIC_6DOF_SPRING_DISCRETE_BEAM [1D] {25}**
***MAT_094:**	***MAT_INELASTIC_SPRING_DISCRETE_BEAM [1D] {9}**
***MAT_095:**	***MAT_INELASTC_6DOF_SPRING_DISCRETE_BEAM [1D] {25}**
***MAT_096:**	***MAT_BRITTLE_DAMAGE [0,8B] {51}**
***MAT_097:**	***MAT_GENERAL_JOINT_DISCRETE_BEAM [1D] {23}**
***MAT_098:**	***MAT_SIMPLIFIED_JOHNSON_COOK [0,1H,1B,1T,2,3] {6}**
***MAT_099:**	***MAT_SIMPLIFIED_JOHNSON_COOK_ORTHOTROPIC_DAMAGE [0,2,3] {22}**
***MAT_100:**	***MAT_SPOTWELD_{*OPTION*} [0,1SW] {6}**
***MAT_100_DA:**	***MAT_SPOTWELD_DAIMLERCHRYSLER [0] {6}**
***MAT_101:**	***MAT_GEPLASTIC_SRATE_2000A [2] {15}**
***MAT_102:**	***MAT_INV_HYPERBOLIC_SIN [0,8B] {15}**
***MAT_103:**	***MAT_ANISOTROPIC_VISCOPLASTIC [0,2,3,5] {20}**
***MAT_103_P:**	***MAT_ANISOTROPIC_PLASTIC [2,3] {20}**
***MAT_104:**	***MAT_DAMAGE_1 [0,2,3] {11}**
***MAT_105:**	***MAT_DAMAGE_2 [0,2,3] {7}**
***MAT_106:**	***MAT_ELASTIC_VISCOPLASTIC_THERMAL [0,2] {20}**
***MAT_107:**	***MAT_MODIFIED_JOHNSON_COOK [0,2,5,8B] {15}**
***MAT_108:**	***MAT_ORTHO_ELASTIC_PLASTIC [2] {15}**
***MAT_110:**	***MAT_JOHNSON_HOLMQUIST_CERAMICS [0,5] {15}**
***MAT_111:**	***MAT_JOHNSON_HOLMQUIST_CONCRETE [0,5] {25}**
***MAT_112:**	***MAT_FINITE_ELASTIC_STRAIN_PLASTICITY [0,5] {22}**
***MAT_113:**	***MAT_TRIP [2] {5}**
***MAT_114:**	***MAT_LAYERED_LINEAR_PLASTICITY [2] {13}**
***MAT_115:**	***MAT_UNIFIED_CREEP [0,2,5] {1}**
***MAT_116:**	***MAT_COMPOSITE_LAYUP [2] {30}**
***MAT_117:**	***MAT_COMPOSITE_MATRIX [2] {30}**

***MAT_118:**	***MAT_COMPOSITE_DIRECT [2] {10}**
***MAT_119:**	***MAT_GENERAL_NONLINEAR_6DOF_DISCRETE_BEAM [1D] {62}**
***MAT_120:**	***MAT_GURSON [0,2] {12}**
***MAT_120_JC:**	***MAT_GURSON_JC [0,2] {12}**
***MAT_120_RCDC:**	***MAT_GURSON_RCDC [0,2] {12}**
***MAT_121:**	***MAT_GENERAL_NONLINEAR_1DOF_DISCRETE_BEAM [1D] {20}**
***MAT_122:**	***MAT_HILL_3R [2,3] {8}**
***MAT_123:**	***MAT_MODIFIED_PIECEWISE_LINEAR_PLASTICITY [0,2,3] {11}**
***MAT_124:**	***MAT_PLASTICITY_COMPRESSION_TENSION [0,1H,2,3,5,8B] {7}**
***MAT_125:**	***MAT_KINEMATIC_HARDENING_TRANSVERSELY_ANISOTROPIC [2] {11}**
***MAT_126:**	***MAT_MODIFIED_HONEYCOMB [0] {20}**
***MAT_127:**	***MAT_ARRUDA_BOYCE_RUBBER [0,5] {49}**
***MAT_128:**	***MAT_HEART_TISSUE [0] {15}**
***MAT_129:**	***MAT_LUNG_TISSUE [0] {49}**
***MAT_130:**	***MAT_SPECIAL_ORTHOTROPIC [2] {35}**
***MAT_131:**	***MAT_ISOTROPIC_SMEARED_CRACK [0,5,8B] {15}**
***MAT_132:**	***MAT_ORTHOTROPIC_SMEARED_CRACK [0] {61}**
***MAT_133:**	***MAT_BARLAT_YLD2000 [2,3] {9}**
***MAT_135:**	***MAT_WTM_STM [2,3] {30}**
***MAT_135_PLC:**	***MAT_WTM_STM_PLC [2,3] {30}**
***MAT_136:**	***MAT_CORUS_VEGTER [2] {5}**
***MAT_138:**	***MAT_COHESIVE_MIXED_MODE [7] {0}**
***MAT_139:**	***MAT_MODIFIED_FORCE_LIMITED [1B] {35}**
***MAT_140:**	***MAT_VACUUM [0,8A] {0}**
***MAT_141:**	***MAT_RATE_SENSITIVE_POLYMER [0,8B] {6}**
***MAT_142:**	***MAT_TRANSVERSELY_ANISOTROPIC_CRUSHABLE_FOAM [0] {12}**
***MAT_143:**	***MAT_WOOD_{*OPTION*} [0] {37}**
***MAT_144:**	***MAT_PITZER_CRUSHABLE FOAM [0,8B] {7}**
***MAT_145:**	***MAT_SCHWER_MURRAY_CAP_MODEL [0,5] {50}**
***MAT_146:**	***MAT_1DOF_GENERALIZED_SPRING [1D] {1}**
***MAT_147**	***MAT_FHWA_SOIL [0,5,8B] {15}**
***MAT_147_N:**	***MAT_FHWA_SOIL_NEBRASKA [0,5,8B] {15}**
***MAT_148:**	***MAT_GAS_MIXTURE [0,8A] {14}**
***MAT_151:**	***MAT_EMMI [0,5,8B] {23}**
***MAT_153:**	***MAT_DAMAGE_3 [0,1H,2,3]**
***MAT_154:**	***MAT_DESHPANDE_FLECK_FOAM [0,8B] {10}**
***MAT_155:**	***MAT_PLASTICITY_COMPRESSION_TENSION_EOS [0,5,8B] {16}**

***MAT_156:**	***MAT_MUSCLE [1T] {0}**
***MAT_157:**	***MAT_ANISOTROPIC_ELASTIC_PLASTIC [2] {5}**
***MAT_158:**	***MAT_RATE_SENSITIVE_COMPOSITE_FABRIC [2,3] {54}**
***MAT_159:**	***MAT_CSCM_{*OPTION*} [0] {22}**
***MAT_161:**	***MAT_COMPOSITE_MSC [0] {34}**
***MAT_162:**	***MAT_COMPOSITE_DMG_MSC [0,2] {40}**
***MAT_163**	***MAT_MODIFIED_CRUSHABLE_FOAM [0,8B] {10}**
***MAT_164:**	***MAT_BRAIN_LINEAR_VISCOELASTIC [0] {14}**
***MAT_165:**	***MAT_PLASTIC_NONLINEAR_KINEMATIC [0,2,8B] {8}**
***MAT_166:**	***MAT_MOMENT_CURVATURE_BEAM [1B] {54}**
***MAT_167:**	***MAT_MCCORMICK [0,8B] {8}**
***MAT_168:**	***MAT_POLYMER [0,8B] {60}**
***MAT_169:**	***MAT_ARUP_ADHESIVE [0] {20}**
***MAT_170:**	***MAT_RESULTANT_ANISOTROPIC [2] {67}**
***MAT_171:**	***MAT_STEEL_CONCENTRIC_BRACE [1B] {33}**
***MAT_172:**	***MAT_CONCRETE_EC2 [1H,2] {35}**
***MAT_173:**	***MAT_MOHR_COULOMB [0] {31}**
***MAT_174:**	***MAT_RC_BEAM [1H] {26}**
***MAT_175:**	***MAT_VISCOELASTIC_THERMAL [0,2,5,8B] {86}**
***MAT_176:**	***MAT_QUASILINEAR_VISCOELASTIC [0,2,5,8B] {81}**
***MAT_177:**	***MAT_HILL_FOAM [0] {12}**
***MAT_178:**	***MAT_VISCOELASTIC_HILL_FOAM [0] {92}**
***MAT_179:**	***MAT_LOW_DENSITY_SYNTHETIC_FOAM_{*OPTION*} [0] {77}**
***MAT_181:**	***MAT_SIMPLIFIED_RUBBER/FOAM_{*OPTION*} [0,2] {39}**
***MAT_183:**	***MAT_SIMPLIFIED_RUBBER_WITH_DAMAGE [0,2] {44}**
***MAT_184:**	***MAT_COHESIVE_ELASTIC [7] {0}**
***MAT_185:**	***MAT_COHESIVE_TH [7] {0}**
***MAT_186:**	***MAT_COHESIVE GENERAL [7] {6}**
***MAT_187:**	***MAT_SAMP-1 [0,2] {38}**
***MAT_188:**	***MAT_THERMO_ELASTO_VISCOPLASTIC_CREEP [0,2] {27}**
***MAT_189:**	***MAT_ANISOTROPIC_THERMOELASTIC [0,8B] {21}**
***MAT_190:**	***MAT_FLD_3-PARAMETER_BARLAT [2] {36}**
***MAT_191:**	***MAT_SEISMIC_BEAM [1B] {36}**
***MAT_192:**	***MAT_SOIL_BRICK [0] {71}**
***MAT_193:**	***MAT_DRUCKER_PRAGER [0] {74}**
***MAT_194:**	***MAT_RC_SHEAR_WALL [2] {36}**
***MAT_195:**	***MAT_CONCRETE_BEAM [1H] {5}**

***MAT_196:**	***MAT_GENERAL_SPRING_DISCRETE_BEAM [1D] {25}**
***MAT_197:**	***MAT_SEISMIC_ISOLATOR [1D] {10}**
***MAT_198:**	***MAT_JOINTED_ROCK [0] {31}**
***MAT_220:**	***MAT_ORTHOTROPIC_ADVANCED_DAMAGE [0] {33}**
***MAT_221:**	***MAT_ORTHOTROPIC_SIMPLIFIED_DAMAGE [0] {17}**
***MAT_224:**	***MAT_TABULATED_JOHNSON_COOK [0,2] {11}**
***MAT_230:**	***MAT_PML_ELASTIC [0] {24}**
***MAT_231:**	***MAT_PML_ACOUSTIC [6] {35}**
***MAT_232:**	***MAT_BIOT_HYSTERETIC [0,2] {30}**
***MAT_237:**	***MAT_PML_HYSTERETIC [0] {54}**
***MAT_255**	*** MAT_PIECEWISE_LINEAR_PLASTIC_THERMAL**

For the discrete (type 6) beam elements, which are used to model complicated dampers and multi-dimensional spring-damper combinations, the following material types are available:

***MAT_066:**	***MAT_LINEAR_ELASTIC_DISCRETE_BEAM [1D]**
***MAT_067:**	***MAT_NONLINEAR_ELASTIC_DISCRETE_BEAM [1D]**
***MAT_068:**	***MAT_NONLINEAR_PLASTIC_DISCRETE_BEAM [1D]**
***MAT_069:**	***MAT_SID_DAMPER_DISCRETE_BEAM [1D]**
***MAT_070:**	***MAT_HYDRAULIC_GAS_DAMPER_DISCRETE_BEAM [1D]**
***MAT_071:**	***MAT_CABLE_DISCRETE_BEAM [1D]**
***MAT_074:**	***MAT_ELASTIC_SPRING_DISCRETE_BEAM [1D]**
***MAT_093:**	***MAT_ELASTIC_6DOF_SPRING_DISCRETE_BEAM [1D]**
***MAT_094:**	***MAT_INELASTIC_SPRING_DISCRETE_BEAM [1D]**
***MAT_095:**	***MAT_INELASTIC_6DOF_SPRING_DISCRETE_BEAM [1D]**
***MAT_119:**	***MAT_GENERAL_NONLINEAR_6DOF_DISCRETE_BEAM [1D]**
***MAT_121:**	***MAT_GENERAL_NONLINEAR_1DOF_DISCRETE_BEAM [1D]**
***MAT_146:**	***MAT_1DOF_GENERALIZED_SPRING [1D]**
***MAT_196:**	***MAT_GENERAL_SPRING_DISCRETE_BEAM [1D]**
***MAT_197:**	***MAT_SEISMIC_ISOLATOR [1D]**

For the discrete springs and dampers the following material types are available.

***MAT_S01:**	***MAT_SPRING_ELASTIC**
***MAT_S02:**	***MAT_DAMPER_VISCOUS**
***MAT_S03:**	***MAT_SPRING_ELASTOPLASTIC**
***MAT_S04:**	***MAT_SPRING_NONLINEAR_ELASTIC**
***MAT_S05:**	***MAT_DAMPER_NONLINEAR_VISCOUS**
***MAT_S06:**	***MAT_SPRING_GENERAL_NONLINEAR**
***MAT_S07:**	***MAT_SPRING_MAXWELL**

***MAT_S08:**	***MAT_SPRING_INELASTIC**
***MAT_S13:**	***MAT_SPRING_TRILINEAR_DEGRADING**
***MAT_S14:**	***MAT_SPRING_SQUAT_SHEARWALL**
***MAT_S15:**	***MAT_SPRING_MUSCLE**

For ALE solids the following material types are available:

***MAT_ALE_01:**	***MAT_ALE_VACUUM (SAME AS *MAT_140)**
***MAT_ALE_02:**	***MAT_ALE_VISCOUS (SIMILAR TO *MAT_009)**
***MAT_ALE_03:**	***MAT_ALE_GAS_MIXTURE (SAME AS *MAT_148)**

For the seatbelts one material is available.

***MAT_B01:**	***MAT_SEATBELT**

For thermal materials in a coupled structural/thermal or thermal only analysis, six materials are available. These materials are related to the structural material via the *PART card. Thermal materials are defined only for solid and shell elements.

	MAT_THERMAL_*OPTION
***MAT_T01:**	***MAT_THERMAL_ISOTROPIC**
***MAT_T02:**	***MAT_THERMAL_ORTHOTROPIC**
***MAT_T03:**	***MAT_THERMAL_ISOTROPIC_TD**
***MAT_T04:**	***MAT_THERMAL_ORTHOTROPIC_TD**
***MAT_T05:**	***MAT_THERMAL_ISOTROPIC_PHASE_CHANGE**
***MAT_T06:**	***MAT_THERMAL_ISOTROPIC_TD_LC**
***MAT_T11-T15:**	***MAT_THERMAL_USER_DEFINED**

MATERIAL MODEL REFERENCE TABLES

The tables provided on the following pages list the material models, some of their attributes, and the general classes of physical materials to which the numerical models might be applied.

If a material model includes any of the following attributes, a "Y" will appear in the respective column of the table:

SRATE - Strain-rate effects
FAIL - Failure criteria
EOS - Equation-of-State required for 3D solids and 2D continuum elements
THERM - Thermal effects
ANISO - Anisotropic/orthotropic
DAM - Damage effects
TENS - Tension handled differently than compression in some manner

Potential applications of the material models, in terms of classes of physical materials, are abbreviated in the table as follows:

GN - General
CM - Composite
CR - Ceramic
FL - Fluid
FM - Foam
GL - Glass
HY - Hydrodynamic material
MT - Metal
PL - Plastic
RB - Rubber
SL - Soil, concrete, or rock
AD - Adhesive
BIO - Biological material
CIV - Civil Engineering component

Material Number and Description	SRATE	FAIL	EOS	THERM	ANISO	DAM	TENS	Applications
1 Elastic								GN, FL
2 Orthotropic Elastic (Anisotropic-solids)					Y			CM, MT
3 Plastic Kinematic/Isotropic	Y	Y						CM, MT, PL
4 Elastic Plastic Thermal				Y				MT, PL
5 Soil and Foam							Y	FM, SL
6 Linear Viscoelastic	Y							RB
7 Blatz-Ko Rubber								RB
8 High Explosive Burn			Y					HY
9 Null Material	Y	Y	Y				Y	FL, HY
10 Elastic Plastic Hydro(dynamic)		Y	Y				Y	HY, MT
11 Steinberg: Temp. Dependent Elastoplastic	Y	Y	Y	Y			Y	HY, MT
12 Isotropic Elastic Plastic								MT
13 Isotropic Elastic Plastic with Failure		Y					Y	MT
14 Soil and Foam with Failure		Y					Y	FM, SL
15 Johnson/Cook Plasticity Model	Y	Y	Y	Y		Y	Y	HY, MT
16 Pseudo Tensor Geological Model	Y	Y	Y			Y	Y	SL
17 Oriented Crack (Elastoplastic w/ Fracture)		Y	Y		Y		Y	HY, MT, PL, CR
18 Power Law Plasticity (Isotropic)	Y							MT, PL
19 Strain Rate Dependent Plasticity	Y	Y						MT, PL
20 Rigid								
21 Orthotropic Thermal (Elastic)				Y	Y			GN
22 Composite Damage		Y			Y		Y	CM
23 Temperature Dependent Orthotropic				Y	Y			CM
24 Piecewise Linear Plasticity (Isotropic)	Y	Y						MT, PL
25 Inviscid Two Invariant Geologic Cap		Y					Y	SL
26 Honeycomb	Y	Y			Y		Y	CM, FM, SL
27 Mooney-Rivlin Rubber							Y	RB

Material Number and Description	SRATE	FAIL	EOS	THERM	ANISO	DAM	TENS	Applications
28 Resultant Plasticity								MT
29 Force Limited Resultant Formulation							Y	
30 Shape Memory								MT
31 Frazer-Nash Rubber							Y	RB
32 Laminated Glass (Composite)		Y						CM, GL
33 Barlat Anisotropic Plasticity (YLD96)	Y				Y			CR, MT
34 Fabric					Y		Y	fabric
35 Plastic-Green Naghdi Rate	Y							MT
36 Three-Parameter Barlat Plasticity	Y				Y			MT
37 Transversely Anisotropic Elastic Plastic					Y			MT
38 Blatz-Ko Foam								FM, PL
39 FLD Transversely Anisotropic					Y			MT
40 Nonlinear Orthotropic		Y		Y	Y		Y	CM
41-50 User Defined Materials	Y	Y	Y	Y	Y	Y	Y	GN
51 Bamman (Temp/Rate Dependent Plasticity)	Y			Y				GN
52 Bamman Damage	Y	Y		Y		Y		MT
53 Closed cell foam (Low density polyurethane)								FM
54 Composite Damage with Chang Failure		Y			Y	Y	Y	CM
55 Composite Damage with Tsai-Wu Failure		Y			Y	Y	Y	CM
57 Low Density Urethane Foam	Y	Y					Y	FM
58 Laminated Composite Fabric		Y			Y	Y	Y	CM, fabric
59 Composite Failure (Plasticity Based)		Y			Y		Y	CM, CR
60 Elastic with Viscosity (Viscous Glass)	Y			Y				GL
61 Kelvin-Maxwell Viscoelastic	Y							FM
62 Viscous Foam (Crash dummy Foam)	Y							FM
63 Isotropic Crushable Foam							Y	FM
64 Rate Sensitive Powerlaw Plasticity	Y							MT

Material Number and Description	SRATE	FAIL	EOS	THERM	ANISO	DAM	TENS	Applications
65 Zerilli-Armstrong (Rate/Temp Plasticity)	Y		Y	Y			Y	MT
66 Linear Elastic Discrete Beam	Y				Y			
67 Nonlinear Elastic Discrete Beam	Y				Y		Y	
68 Nonlinear Plastic Discrete Beam	Y	Y			Y			
69 SID Damper Discrete Beam	Y							
70 Hydraulic Gas Damper Discrete Beam	Y							
71 Cable Discrete Beam (Elastic)							Y	cable
72 Concrete Damage (incl. Release III)	Y	Y	Y			Y	Y	SL
73 Low Density Viscous Foam	Y	Y					Y	FM
74 Elastic Spring Discrete Beam	Y	Y					Y	
75 Bilkhu/Dubois Foam							Y	FM
76 General Viscoelastic (Maxwell Model)	Y			Y			Y	RB
77 Hyperelastic and Ogden Rubber	Y						Y	RB
78 Soil Concrete		Y				Y	Y	SL
79 Hysteretic Soil (Elasto-Perfectly Plastic)		Y					Y	SL
80 Ramberg-Osgood								SL
81 Plasticity with Damage	Y	Y				Y		MT, PL
82 Plasticity with Damage Ortho	Y	Y			Y	Y		
83 Fu Chang Foam	Y	Y				Y	Y	FM
84 Winfrith Concrete (w/ rate effects)	Y						Y	FM, SL
85 Winfrith Concrete							Y	SL
86 Orthotropic Viscoelastic	Y				Y			RB
87 Cellular Rubber	Y						Y	RB
88 MTS	Y		Y	Y				MT
89 Plasticity Polymer	Y						Y	PL
90 Acoustic							Y	FL
91 Soft Tissue	Y	Y			Y		Y	BIO

Material Number and Description	SRATE	FAIL	EOS	THERM	ANISO	DAM	TENS	Applications
93 Elastic 6DOF Spring Discrete Beam	Y	Y			Y		Y	
94 Inelastic Spring Discrete Beam	Y	Y					Y	
95 Inelastic 6DOF Spring Discrete Beam	Y	Y			Y		Y	
96 Brittle Damage	Y	Y			Y	Y	Y	SL
97 General Joint Discrete Beam								
98 Simplified Johnson Cook	Y	Y						MT
99 Simpl. Johnson Cook Orthotropic Damage	Y	Y			Y	Y		MT
100 Spotweld	Y	Y				Y	Y	MT (spotwelds)
101 GE Plastic Strain Rate	Y	Y					Y	PL
102 Inv Hyperbolic Sin	Y			Y				MT, PL
103 Anisotropic Viscoplastic	Y	Y			Y			MT
103P Anisotropic Plastic					Y			MT
104 Damage 1	Y	Y			Y	Y		MT
105 Damage 2	Y	Y				Y		MT
106 Elastic Viscoplastic Thermal	Y			Y				PL
107 Modified Johnson Cook	Y	Y		Y		Y		MT
108 Ortho Elastic Plastic					Y			
110 Johnson Holmquist Ceramics	Y	Y				Y	Y	CR, GL
111 Johnson Holmquist Concrete	Y	Y				Y	Y	SL
112 Finite Elastic Strain Plasticity	Y							PL
113 Transformation Induced Plasticity (TRIP)				Y				MT
114 Layered Linear Plasticity	Y	Y						MT, PL, CM
115 Unified Creep								
116 Composite Layup					Y			CM
117 Composite Matrix					Y			CM
118 Composite Direct					Y			CM
119 General Nonlinear 6DOF Discrete Beam	Y	Y			Y		Y	

Material Number and Description	SRATE	FAIL	EOS	THERM	ANISO	DAM	TENS	Applications
120 Gurson	Y	Y				Y	Y	MT
121 General Nonlinear 1DOF Discrete Beam	Y	Y					Y	
122 Hill 3RC					Y			MT
123 Modified Piecewise Linear Plasticity	Y	Y						MT, PL
124 Plasticity Compression Tension	Y	Y					Y	MT, PL
125 Kinematic Hardening Transversely Aniso.					Y			MT
126 Modified Honeycomb	Y	Y			Y	Y	Y	CM, FM, SL
127 Arruda Boyce Rubber	Y							RB
128 Heart Tissue					Y		Y	BIO
129 Lung Tissue	Y						Y	BIO
130 Special Orthotropic					Y			
131 Isotropic Smeared Crack		Y				Y	Y	MT, CM
132 Orthotropic Smeared Crack		Y			Y	Y		MT, CM
133 Barlat YLD2000	Y				Y			MT
135 Weak and Strong Texture Model	Y	Y			Y			MT
136 Corus Vegter					Y			MT
138 Cohesive Mixed Mode		Y			Y	Y	Y	AD
139 Modified Force Limited						Y	Y	
140 Vacuum								
141 Rate Sensitve Polymer	Y							PL
142 Transversely Anisotropic Crushable Foam					Y		Y	FM
143 Wood	Y	Y			Y	Y	Y	(wood)
144 Pitzer Crushable Foam	Y						Y	FM
145 Schwer Murray Cap Model	Y	Y				Y	Y	SL
146 1DOF Generalized Spring	Y							
147 FWHA Soil	Y					Y	Y	SL
147N FHWA Soil Nebraska	Y					Y	Y	SL

Material Number and Description	SRATE	FAIL	EOS	THERM	ANISO	DAM	TENS	Applications
148 Gas Mixture				Y				FL
151 Evolving Microstructural Model of Inelast.	Y	Y		Y	Y	Y		MT
153 Damage 3	Y	Y				Y		MT, PL
154 Deshpande Fleck Foam		Y						FM
155 Plasticity Compression Tension EOS	Y	Y	Y				Y	(ice)
156 Muscle	Y						Y	BIO
157 Anisotropic Elastic Plastic					Y			MT, CM
158 Rate-Sensitive Composite Fabric	Y	Y			Y	Y	Y	CM
159 CSCM	Y	Y				Y	Y	SL
161,162 Composite MSC	Y	Y			Y	Y	Y	CM
163 Modified Crushable Foam	Y						Y	FM
164 Brain Linear Viscoelastic	Y							BIO
165 Plastic Nonlinear Kinematic		Y						MT
166 Moment Curvature Beam	Y	Y					Y	CIV
167 McCormick	Y							MT
168 Polymer				Y			Y	PL
169 Arup Adhesive	Y	Y			Y		Y	AD
170 Resultant Anisotropic					Y			PL
171 Steel Concentric Brace						Y	Y	CIV
172 Concrete EC2		Y		Y			Y	SL, MT
173 Mohr Coulomb					Y		Y	SL
174 RC Beam						Y	Y	SL
175 Viscoelastic Thermal	Y			Y			Y	RB
176 Quasilinear Viscoelastic	Y	Y				Y	Y	BIO
177 Hill Foam							Y	FM
178 Viscoelastic Hill Foam (Ortho)	Y						Y	FM
179 Low Density Synthetic Foam	Y	Y			Y	Y	Y	FM

Material Number and Description	SRATE	FAIL	EOS	THERM	ANISO	DAM	TENS	Applications
181 Simplified Rubber/Foam	Y	Y				Y	Y	RB, FM
183 Simplified Rubber with Damage	Y	Y				Y	Y	RB
184 Cohesive Elastic		Y					Y	AD
185 Cohesive TH		Y			Y	Y	Y	AD
186 Cohesive General		Y			Y	Y	Y	AD
187 Semi-Analytical Model for Polymers – 1	Y	Y				Y		PL
188 Thermo Elasto Viscoelastic Creep	Y			Y				MT
189 Anisotropic Thermoelastic				Y	Y			
190 Flow limit diagram 3-Parameter Barlat		Y			Y		Y	MT
191 Seismic Beam							Y	CIV
192 Soil Brick								SL
193 Drucker Prager							Y	SL
194 RC Shear Wall		Y				Y	Y	CIV
195 Concrete Beam	Y	Y				Y	Y	CIV
196 General Spring Discrete Beam	Y						Y	
197 Seismic Isolator	Y	Y			Y		Y	CIV
198 Jointed Rock		Y			Y		Y	SL
230 Elastic Perfectly Matched Layer (PML)	Y							SL
231 Acoustic PML								FL
232 Biot Linear Hysteretic Material	Y							SL
237 Biot Hysteretic PML	Y							SL
A01 ALE Vacuum								
A02 ALE Viscous			Y				Y	FL
A03 ALE Gas Mixture				Y				FL
S1 Spring Elastic (Linear)								
S2 Damper Viscous (Linear)	Y							
S3 Spring Elastoplastic (Isotropic)								
S4 Spring Nonlinear Elastic	Y						Y	
S5 Damper Nonlinear Viscous	Y						Y	
S6 Spring General Nonlinear							Y	
S7 Spring Maxwell (3-Parameter Viscoelastic)	Y							

Material Number and Description	SRATE	FAIL	EOS	THERM	ANISO	DAM	TENS	Applications
S8 Spring Inelastic (Tension or Compression)							Y	
S13 Spring Trilinear Degrading		Y				Y		CIV
S14 Spring Squat Shearwall						Y		CIV
S15 Spring Muscle	Y						Y	BIO
B1 Seatbelt							Y	
T01 Thermal Isotropic				Y				Heat transfer
T02 Thermal Orthotropic				Y	Y			Heat transfer
T03 Thermal Isotropic (Temp Dependent)				Y				Heat transfer
T04 Thermal Orthotropic (Temp Dependent)				Y	Y			Heat transfer
T05 Thermal Isotropic (Phase Change)				Y				Heat transfer
T06 Thermal Isotropic (Temp dep-load curve)				Y				Heat transfer
T11 Thermal User Defined				Y				Heat transfer

*MAT_ADD_EROSION

Many of the constitutive models in LS-DYNA do not allow failure and erosion. The ADD_EROSION option provides a way of including failure in these models although the option can also be applied to constitutive models with other failure/erosion criterion. Each of the criterion defined here are applied independently, and once any one of them is satisfied, the element is deleted from the calculation. NOTE: *In the R4 release of 971, this option applies to nonlinear element formulations including the 2D continuum, 3D solid elements, 3D shell elements, and the thick shell elements types 1 and 2.* In addition to erosion, damage initiation and evolution models are available as described in the remarks.

Define the following two cards:

Card 1

Variable	MID	EXCL	MXPRES	MNEPS	EFFEPS	VOLEPS	NUMFIP	NCS
Type	A8	F	F	F	F	F	F	F
Default	none	none	0.0	0.0	0.0	0.0	1.0	1.0

Card 2

Variable	MNPRES	SIGP1	SIGVM	MXEPS	EPSSH	SIGTH	IMPULSE	FAILTM
Type	F	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	none	none

The following card is optional:

Card 3 1 2 3 4 5 6 7 8

Variable	IDAM	DMGTYP	LCSDG	ECRIT	DMGEXP	DCRIT	FADEXP	LCREGD
Type	A8	F	F	F	F	F	F	F
Default	0.0	0.0	0.0	0.0	1.0	0.0	1.0	0.0

If IDAM.GT.0 define the following card:

Card 4 1 2 3 4 5 6 7 8

Variable	SIZFLG	REFSZ	NAHSV					
Type	F	F	F					
Default	0.0	0.0	0.0					

If IDAM.LT.0 then define -IDAM set of cards on the following format:

Card 4 1 2 3 4 5 6 7 8

Variable	DITYP	P1	P2					
Type	F	F	F					
Default	0.0	0.0	0.0					

Card 4	1	2	3	4	5	6	7	8
Variable	DETYP	DCTYP	Q1					
Type	F	F	F					
Default	0.0	0.0	0.0					

VARIABLE	DESCRIPTION
MID	Material identification for which this erosion definition applies. A unique number or label not exceeding 8 characters must be specified.
EXCL	The exclusion number, which applies to the values defined on Card 2. When any of the failure constants are set to the exclusion number, the associated failure criteria calculations are bypassed (which reduces the cost of the failure model). For example, to prevent a material from going into tension, the user should specify an unusual value for the exclusion number, e.g., 1234., set $P_{\min}$ to 0.0 and all the remaining constants to 1234. The default value is 0.0, which eliminates all criteria from consideration that have their constants set to 0.0 or left blank in the input file.
MXPRES	Maximum pressure at failure, $P_{\max}$. If the value is exactly zero, it is automatically excluded to maintain compatibility with old input files.
MNEPS	Minimum principal strain at failure, $\varepsilon_{\min}$. If the value is exactly zero, it is automatically excluded to maintain compatibility with old input files.
EFFEPS	Maximum effective strain at failure, $\varepsilon_{eff} = \sqrt{2/3\ \varepsilon_{ij}^{dev}\varepsilon_{ij}^{dev}}$. If the value is exactly zero, it is automatically excluded to maintain compatibility with old input files. If the value is negative, then |EFFEPS| is the effective plastic strain to failure.
VOLEPS	Volumetric strain at failure, $\varepsilon_{vol} = \varepsilon_{11} + \varepsilon_{22} + \varepsilon_{33}$. VOLEPS can be a positive or negative number depending on whether the failure is in tension or compression, respectively. If the value is exactly zero, it is automatically excluded to maintain compatibility with old input files.
NUMFIP	Number of failed integration points prior to element deletion. The default is unity.

VARIABLE	DESCRIPTION
	LT.0.0: |NUMFIP| is percentage of integration points/layers which must fail before element fails (only GISSMO).
NCS	Number of failure conditions to satisfy before failure occurs. For example, if SIGP1 and SIGVM are defined and if NCS=2, both failure criteria must be met before element deletion can occur. The default is set to unity.
MNPRES	Minimum pressure at failure, $P_{\min}$.
SIGP1	Principal stress at failure, $\sigma_{\max}$.
SIGVM	Equivalent stress at failure, $\bar{\sigma}_{\max}$.
MXEPS	Maximum principal strain at failure, $\varepsilon_{\max}$. The maximum principal strain at failure is made a function of the effective strain rate by setting MXEPS to the negative of the appropriate load curve ID.
EPSSH	Shear strain at failure, $\gamma_{\max}$.
SIGTH	Threshold stress, σ_0.
IMPULSE	Stress impulse for failure, K_f.
FAILTM	Failure time. When the problem time exceeds the failure time, the material is removed.
IDAM	Flag for damage model. EQ.0: no damage model is used. EQ.1: GISSMO damage model. LT.0: -IDAM represents the number of damage initiation and evolution criteria to be applied
DMGTYP	For GISSMO damage type the following applies. EQ.0: Damage is accumulated, no coupling to flow stress, no failure. EQ.1: Damage is accumulated, element failure occurs for D=1. Coupling of damage to flow stress depending on parameters, see remarks below. For IDAM.LT.0 the following applies. EQ.0: No action is taken EQ.1: Damage history is initiated based on values of initial plastic strains and initial strain tensor, this is to be used in multistage analyses

VARIABLE	DESCRIPTION
	when damage history is unavailable from previous steps. This relies on having a zero initial stress and a non-zero initial strain state.
LCSDG	Load curve ID defining equivalent plastic strain to failure vs. triaxiality.
ECRIT	Critical plastic strain (material instability), see below. LT.0.0: |ECRIT| is load curve ID defining critical equivalent plastic strain vs. triaxiality. EQ.0.0: Fixed value DCRIT defining critical damage is read (see below) GT.0.0: Fixed value for stress-state independent critical equivalent plastic strain.
DMGEXP	Exponent for nonlinear damage accumulation, see remarks.
DCRIT	Damage threshold value (critical damage). If a Load curve of critical plastic strain or fixed value is given by ECRIT, input is ignored.
FADEXP	Exponent for damage-related stress fadeout. LT.0.0: |FADEXP| is load curve ID defining element-size dependent fading exponent. GT.0.0: Constant fading exponent.
LCREGD	Load curve ID defining element size dependent regularization factors for equivalent plastic strain to failure.
SIZFLG	Flag for method of element size determination. EQ.0 (default): Element size is determined in undeformed configuration as square root of element area (shells), or cubic root of element volume (solids), respectively. EQ.1: Element size is updated every time step, and determined as mean edge length. (This option was added to ensure comparability with *MAT_120, and is not recommended for general purpose).
REFSZ	Reference element size, for which an additional output of damage will be generated. This is necessary to ensure the applicability of resulting damage quantities when transferred to different mesh sizes.
NAHSV	Number of history variables from damage model which should be stored in standard material history array for Postprocessing. See remarks.
DITYP	Damage initiation type EQ.0.0: Ductile EQ.1.0: Shear EQ.2.0: MSFLD

VARIABLE	DESCRIPTION
P1	Damage initiation parameter DITYP.EQ.0.0: Load curve/table ID representing plastic strain at onset of damage as function of stress triaxiality and optionally plastic strain rate. DITYP.EQ.1.0: Load curve/table ID representing plastic strain at onset of damage as function of shear influence and optionally plastic strain rate. DITYP.EQ.2.0: Load curve/table ID representing plastic strain at onset of damage as function of ratio of principal plastic strain rates and optionally plastic strain rate.
P2	Damage initiation parameter DITYP.EQ.0.0: Not used DITYP.EQ.1.0: Pressure influence coefficient k_S DITYP.EQ.2.0: Not used
DETYP	Damage evolution type EQ.0.0: Linear
DCTYP	Damage composition option for multiple criteria EQ.0.0: Maximum EQ.1.0: Multiplicative
Q1	Damage evolution parameter DETYP.EQ.0.0: Plastic displacement at failure

The criteria for failure besides failure time are:

1. $P \geq P_{\max}$, where P is the pressure (positive in compression), and $P_{\max}$ is the maximum pressure at failure.

2. $\varepsilon_3 \leq \varepsilon_{\min}$, where ε_3 is the minimum principal strain, and $\varepsilon_{\min}$ is the minimum principal strain at failure.

3. $P \leq P_{\min}$, where P is the pressure (positive in compression), and $P_{\min}$ is the minimum pressure at failure.

4. $\sigma_1 \geq \sigma_{\max}$, where σ_1 is the maximum principal stress, and $\sigma_{\max}$ is the maximum principal stress at failure.

5. $\sqrt{\frac{3}{2}\sigma_{ij}^{'}\sigma_{ij}^{'}} \geq \bar{\sigma}_{\max}$, where $\sigma_{ij}^{'}$ are the deviatoric stress components, and $\bar{\sigma}_{\max}$ is the equivalent stress at failure.

6. $\varepsilon_1 \geq \varepsilon_{\max}$, where ε_1 is the maximum principal strain, and $\varepsilon_{\max}$ is the maximum principal strain at failure.

7. $\gamma_1 \geq \gamma_{\max}$, where γ_1 is the maximum shear strain = $(\varepsilon_1 - \varepsilon_3)/2$, and $\gamma_{\max}$ is the shear strain at failure.

8. The Tuler-Butcher criterion,

$$\int_0^t [\max(0, \sigma_1 - \sigma_0)]^2 \, dt \geq K_f ,$$

where σ_1 is the maximum principal stress, σ_0 is a specified threshold stress, $\sigma_1 \geq \sigma_0 \geq 0$, and K_f is the stress impulse for failure. Stress values below the threshold value are too low to cause fracture even for very long duration loadings.

Remarks on Damage Models:

GISSMO

The GISSMO damage model is a phenomenological formulation that allows for an incremental description of damage accumulation, including softening and failure. It is intended to provide a maximum in variability for the description of damage for a variety of metallic materials (e.g. *MAT_024, *MAT_036, …). The input of parameters is based on tabulated data, allowing the user to directly convert test data to numerical input.

The model is based on an incremental formulation of damage accumulation:

$$\Delta D = \frac{DMGEXP}{\varepsilon_f} D^{\left(1-\frac{1}{DMGEXP}\right)} \Delta\varepsilon_p$$

with

D: Damage value $(0 \leq D \leq 1)$. For numerical reasons, D is initialized to a value of 1.E-20 for all damage types in the first time step

ε_f: Equivalent plastic strain to failure, determined from LCSDG as a function of the current triaxiality value η

$\Delta\varepsilon_p$: Equivalent plastic strain increment

For constant values of failure strain , this damage rate can be integrated to get a relation of damage and actual equivalent plastic strain:

$$D = \left(\frac{\varepsilon_p}{\varepsilon_f}\right)^{DMGEXP} \quad \text{for } \varepsilon_f = const. \text{ only!}$$

Triaxiality η as a measure of the current stress state is defined as

$\eta = \frac{\sigma_H}{\sigma_M}$ with hydrostatic stress σ_H and equivalent von Mises stress σ_M .

For DMGTYP.EQ.0, damage is accumulated according to the description above, yet no softening and failure is taken into account. Thus, parameters ECRIT, DCRIT and FADEXP will not have any influence. This option can be used to calculate pre-damage in multi-stage deformations without influencing the simulation results.

For DMGTYP.EQ.1, elements will be deleted if $D \geq 1$.

Depending on the set of parameters given by ECRIT (or DCRIT) and FADEXP, a Lemaitre-type coupling of damage and stress *(effective stress concept)* can be used.

Three principal ways of damage definition can be used:

1.) Input of a fixed value of critical plastic strain (ECRIT.GT.0.)
As soon as the magnitude of plastic strain reaches this value, the current damage parameter D is stored as critical damage DCRIT and the damage coupling flag ist set to unity, in order to facilitate an identification of critical elements in postprocessing. From this point on, damage is coupled to the stress tensor using the following relation:

$$\sigma = \tilde{\sigma} \left(1 - \left(\frac{D - DCRIT}{1 - DCRIT} \right)^{FADEXP} \right)$$

This leads to a continuous reduction of stress, up to the load-bearing capacity completely vanishing as D reaches unity. The fading exponent FADEXP can be defined element size dependent, to allow for the consideration of an element-size dependent amount of energy to be dissipated during element fade-out.

2.) Input of a load curve defining critical plastic strain vs. triaxiality (ECRIT.LT.0.), pointing to load curve ID |ECRIT|. This allows for a definition of triaxiality-dependent material instability, which takes account of that instability and localization will occur depending on the actual load case. This offers the possibility to use a transformed Forming Limit Diagram as an input for the expected onset of softening and localization. Using this load curve, the instability measure F is accumulated using the following relation, which is similar to the accumulation of damage D except for the instability curve is used as an input:

$$\Delta F = \frac{DMGEXP}{\varepsilon_{p,loc}} F^{\left(1 - \frac{1}{DMGEXP}\right)} \Delta\varepsilon_p$$

with F: Instability measure (0≤F≤1).
$\varepsilon_{p,loc}$: Equivalent plastic strain to instability, determined from ECRIT
$\Delta\varepsilon_p$: Equivalent plastic strain increment

As soon as the instability measure F reaches unity, the current value of damage D in the respective element is stored. Damage will from this point on be coupled to the flow stress using the relation described above.

3.) If no input for ECRIT is made, parameter DCRIT will be considered.
Coupling of Damage to the stress tensor starts if this value *(damage threshold)* is exceeded (0≤DCRIT≤1). Coupling of damage to stress is done using the relation described above.

This input allows for the use of extreme values also – for example, DCRIT.EQ.0.0 would lead to no coupling at all, and element deletion under full load (brittle fracture).

History Variables:

History variables of the GISSMO damage model are written to the Postprocessing database only if NAHSV>0. The damage history variables start at position ND, which is displayed in d3hsp file, e.g. “first damage history variable = 6” means that ND=6.

Variable	*Description*		
ND	Damage parameter D (1.E-20 ≤ D ≤ 1)		
ND+1	Damage threshold DCRIT		
ND+2	Domain flag for damage coupling:	=0	no coupling
		=1	coupling
ND+3	Triaxiality variable σ_H / σ_M		
ND+4	Equivalent plastic strain		
ND+5	Regularization factor for failure strain (determined from LCREGD)		
ND+6	Exponent for stress fading FADEXP		
ND+7	Calculated element size		
ND+8	Instability measure F		
ND+9	Resultant damage parameter D for element size REFSZ		
ND+10	Resultant damage threshold DCRIT for element size REFSZ		

Damage initiation and evolution criteria

As an alternative to GISSMO, the user may invoke an arbitrary number of damage initiation and evolution criteria, the number of course in practice being limited by the number of available criteria. With this option the following theory applies.

Assuming that *n* initiation/evolution types have been defined according to the information above, *n* being the same as –IDAM above, damage initiation and evolution history variables $\omega_D^i \in [0,\infty[$ and $D^i \in [0,1]$, *i=1,…n,* are introduced for each integration point. These are initially set to zero

and then evolve with the deformation of the elements according to rules associated with the specific damage initiation and evolution type chosen, see below for details. The variables can be post-processed just as ordinary material history variables and their positions in the history variables array is given in d3hsp, search for the string *Damage history listing*. The damage initiation variables do not influence the results but just serve as an indicator for the onset of damage. The damage evolution variables govern the damage in the material and are used to form the global damage $D \in [0,1]$. When multiple criteria are active, *n>1*, each individual criterion can be of maximum, $i \in I_{\max}$, or multiplicative, $i \in I_{\text{mult}}$, type, this is defined by the DCTYP parameter. The global damage variable is defined as

$$D = \max(D_{\max}, D_{\text{mult}})$$

where

$$D_{\max} = \max_{i \in I_{\max}} D^i$$
$$D_{\text{mult}} = 1 - \prod_{i \in I_{\text{mult}}} (1 - D^i) .$$

The damage variable relates the macroscopic (damaged) and microscopic (true) stress according to

$$\sigma = (1 - D)\tilde{\sigma} .$$

Once the damage has reached the level of D_{erode} (=0.99 by default) the stress is set to zero and the integration point is assumed failed, thus not processed after that. When NUMFIP integration points have failed the element is eroded and removed from the finite element model.

Now to the evolution of the individual damage initiation and evolution history variables, and for the sake of clarity we skip the superscript *i* from now on.

The variables ω_D governs the onset of damage and evolves independently of eachother and according to the following.

Ductile (DITYP.EQ.0):

For the ductile initiation option a function $\varepsilon_D^p = \varepsilon_D^p(\eta, \dot{\varepsilon}^p)$ represents the plastic strain at onset of damage (P1). This is a function of stress triaxiality defined as

$$\eta = -p / q$$

with *p* being the pressure and *q* the von Mises equivalent stress. Optionally this can be defined as a table with the second dependency being on the effective plastic strain rate $\dot{\varepsilon}^p$. The damage initiation history variable evolves according to

$$\omega_D = \int_0^{\varepsilon^p} \frac{d\varepsilon^p}{\varepsilon_D^p}.$$

Shear (DITYP.EQ.1):

For the shear initiation option a function $\varepsilon_D^p = \varepsilon_D^p(\theta, \dot{\varepsilon}^p)$ represents the plastic strain at onset of damage (P1). This is a function of a shear stress function defined as

$$\theta = (q + k_S\, p) / \tau$$

with *p* being the pressure, *q* the von Mises equivalent stress and τ the maximum shear stress defined as a function of the principal stress values

$$\tau = \left(\sigma_{\text{major}} - \sigma_{\text{minor}}\right) / 2 .$$

Introduced here is also the pressure influence parameter k_S (P2). Optionally this can be defined as a table with the second dependency being on the effective plastic strain rate $\dot{\varepsilon}^p$. The damage initiation history variable evolves according to

$$\omega_D = \int_0^{\varepsilon^p} \frac{d\varepsilon^p}{\varepsilon_D^p}.$$

MSFLD (DITYP.EQ.2):

For the MSFLD initiation option a function $\varepsilon_D^p = \varepsilon_D^p(\alpha, \dot{\varepsilon}^p)$ represents the plastic strain at onset of damage (P1). This is a function of the ratio of principal plastic strain rates defined as

$$\alpha = \dot{\varepsilon}^p_{\text{minor}} / \dot{\varepsilon}^p_{\text{major}} .$$

The MSFLD criterion is only relevant for shells and the principal strains should be interpreted as the in-plane principal strains. For simplicity the plastic strain evolution in this formula is assumed to stem from an associated von Mises flow rule and whence

$$\alpha = s_{\text{minor}} / s_{\text{major}}$$

with *s* being the deviatoric stress. This assures that the calculation of α is in a sense robust at the expense of being slightly off for materials with anisotropic yield functions and/or non-associated flow rules. Optionally this can be defined as a table with the second dependency being on the effective plastic strain rate $\dot{\varepsilon}^p$, for $\dot{\varepsilon}^p = 0$ the value of ε_D^p is set to a large number to prevent onset of damage for no plastic evolution. The damage initiation history variable evolves according to

$$\omega_D = \max_{t \le T} \frac{\varepsilon^p}{\varepsilon_D^p},$$

which should be interpreted as the maximum value up to this point in time. An important note with this initiation option is that the damage initiation variable is evaluated using the strains and stresses at the mid-surface of the shell and thus bending effects are not taken into account.

For the evolution of the associated damage variable *D* we introduce the plastic displacement u^P which evolves according to

$$\dot{u}^p = \begin{cases} 0 & \omega_D < 1 \\ l\dot{\varepsilon}^p & \omega_D \geq 1 \end{cases}$$

with *l* being a characteristic length of the element. The following defines the evolution of the damage variable.

Linear (DETYP.EQ.0):

With this option the damage variable evolves linearly with the plastic displacement

$$\dot{D} = \dot{u}^p / u_f^p$$

with u_f^p being the plastic displacement at failure (Q1).

***MAT_ADD_PERMEABILITY**

For consolidation calculations.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	PERM						
Type	I	F						
Default	None	None						

VARIABLE	DESCRIPTION
MID	Material identification – must be same as the structural material.
PERM	Permeability

The units of PERM are length/time (volume flow rate of water per unit area per gradient of head of excess pore pressure head).

See notes under *CONTROL_PORE_FLUID

*MAT_ADD_THERMAL_EXPANSION

The ADD_THERMAL_EXPANSION option is used to occupy an arbitrary material model in LS-DYNA with a thermal expansion property. This option applies to all nonlinear solid, shell, thick shell and beam elements and all material models except those models which use resultant formulations such as *MAT_RESULTANT_PLASTICITY and *MAT_SPECIAL_ORTHOTROPIC. Orthotropic expansion effects are supported for anisotropic materials.

Define the following card

Card 1	1	2	3	4	5	6	7	8
Variable	PID	LCID	MULT	LCID	MULTY	LCID	MULTZ	
Type	I	I	F	I	F	I	F	
Default	none	none	1.0	LCID	MULT	LCID	MULT	

VARIABLE	DESCRIPTION
PID	Part ID for which the thermal expansion property applies
LCID	Load curve ID defining thermal expansion coefficient in local x-direction as a function of temperature. If zero, the thermal expansion coefficient in local x-direction given by constant MULT.
MULT	Scale factor scaling load curve given by LCID
LCIDY	Load curve ID defining thermal expansion coefficient in local y-direction as a function of temperature. If zero, the thermal expansion coefficient in local y-direction given by constant MULTY, if MULTY=0 as well, the properties in x-direction are used.
MULTY	Scale factor scaling load curve given by LCIDY
LCIDZ	Load curve ID defining thermal expansion coefficient in local z-direction as a function of temperature. If zero, the thermal expansion coefficient in local z-direction given by constant MULTZ, if MULTZ=0 as well, the properties in x-direction are used.
MULTZ	Scale factor scaling load curve given by LCIDZ

Remarks:

When invoking the isotropic thermal expansion property (no use of the local y and z parameters) for a material, the stress update is based on the elastic strain rates given by

$$\dot{\varepsilon}_{ij}^{e} = \dot{\varepsilon}_{ij} - \alpha(T)\dot{T}\delta_{ij}$$

rather than on the total strain rates $\dot{\varepsilon}_{ij}$. For a material with the stress based on the deformation gradient F_{ij}, the elastic part of the deformation gradient is used for the stress computations

$$F_{ij}^{e} = J_T^{-1/3} F_{ij}$$

where J_T is the thermal jacobian. The thermal jacobian is updated using the rate given by

$$\dot{J}_T = 3\alpha(T)\dot{T}J_T .$$

For orthotropic properties, which apply only to materials with anisotropy, these equations are generalized to

$$\dot{\varepsilon}_{ij}^{e} = \dot{\varepsilon}_{ij} - \alpha_k(T)\dot{T}q_{ik}q_{jk}$$

and

$$F_{ij}^{e} = F_{ik}\beta_l^{-1}Q_{kl}Q_{jl}$$

where the β_i are updated as

$$\dot{\beta}_i = \alpha_i(T)\dot{T}\beta_i .$$

Here q_{ij} represents the matrix with material directions with respect to the current configuration whereas Q_{ij} are the corresponding directions with respect to the initial configuration. For (shell) materials with multiple layers of different anisotropy directions, the mid surface layer determines the orthotropy for the thermal expansion.

*MAT_NONLOCAL

In nonlocal failure theories the failure criterion depends on the state of the material within a radius of influence which surrounds the integration point. An advantage of nonlocal failure is that mesh size sensitivity on failure is greatly reduced leading to results which converge to a unique solution as the mesh is refined. Without a nonlocal criterion, strains will tend to localize randomly with mesh refinement leading to results which can change significantly from mesh to mesh. The nonlocal failure treatment can be a great help in predicting the onset and the evolution of material failure. This option can be used with two and three-dimensional solid elements, and three-dimensional shell elements. The implementation is available for under integrated elements, which have one integration point at their center. Shells are assumed to have multiple integration points through their thickness. This is a new option and should be used with caution. This option applies to a subset of elastoplastic materials that include a damage - based failure criterion.

Define the following cards:

Card 1

Variable	IDNL	PID	P	Q	L	NFREQ		
Type	I	I	I	I	F	I		
Default	none	none	none	none	none	none		

Card 2

Variable	NL1	NL2	NL3	NL4	NL5	NL6	NL7	NL8
Type	I	I	I	I	I	I	I	I
Default	none	none	none	none	none	none	none	none

Define one card for each symmetry plane. Up to six symmetry planes can be defined. The next "*" card terminates this input.

Cards 3,...	1	2	3	4	5	6	7	8
Variable	XC1	YC1	ZC1	XC2	YC2	ZC2		
Type	F	F	F	F	F	F		
Default	none	none	none	none	none	none		

VARIABLE	DESCRIPTION
IDNL	Nonlocal material input ID.
PID	Part ID for nonlocal material.
P	Exponent of weighting function. A typical value might be 8 depending somewhat on the choice of L. See equations below.
Q	Exponent of weighting function. A typical value might be 2. See equations below.
L	Characteristic length. This length should span a few elements. See equations below.
NFREQ	Number of time steps between update of neighbors. The nearest neighbor search can add significant computational time so it is suggested that NFREQ be set to value of 10 to 100 depending on the problem. This parameter may be somewhat problem dependent.
NL1,..,NL8	Define up to eight history variable ID's for nonlocal treatment.
XC1, YC1,ZC1	Coordinate of point on symmetry plane.
XC2, YC2, ZC2	Coordinate of a point along the normal vector.

Remarks:

The memory usage for this option can vary during the duration of the calculation. It is recommended that additional memory be requested by using the *CONTROL_NONLOCAL input. Usually, a value of 10 should be okay.

For elastoplastic material models in LS-DYNA which use the plastic strain as a failure criterion, the first history variable, which does not count the six stress components, is the plastic strain. In this case the variable NL1=1 and NL2 - NL8=0. See the table below, which lists the history variable ID's for a subset of materials.

Material Model Name	Effective Plastic Strain Location	Damage Parameter Location
PLASTIC_KINEMATIC	1	N/A
JOHNSON_COOK	1	5 (shells); 7 (solids)
PIECEWISE_LINEAR_PLASTICITY	1	N/A
PLASTICITY_WITH_DAMAGE	1	2
MODIFIED_ZERILLI-ARMSTRONG	1	N/A
DAMAGE_1	1	4
DAMAGE_2	1	2
MODIFIED_PIECEWISE_LINEAR_PLAST	1	N/A
PLASTICITY_COMPRESSION_TENSION	1	N/A
JOHNSON_HOLMQUIST_CONCRETE	1	2
GURSON	1	2

In applying the nonlocal equations to shell elements, integration points lying in the same plane within the radius determined by the characteristic length are considered. Therefore, it is important to define the connectivity of the shell elements consistently within the part ID, e.g., so that the outer integration points lie on the same surface.

The equations and our implementation are based on the implementation by Worswick and Lalbin [1999] of the nonlocal theory to Pijaudier-Cabot and Bazant [1987]. Let Ω_r be the neighborhood of radius, L, of element e_r and $\{e_i\}_{i=1,\ldots,N_r}$ the list of elements included in Ω_r, then

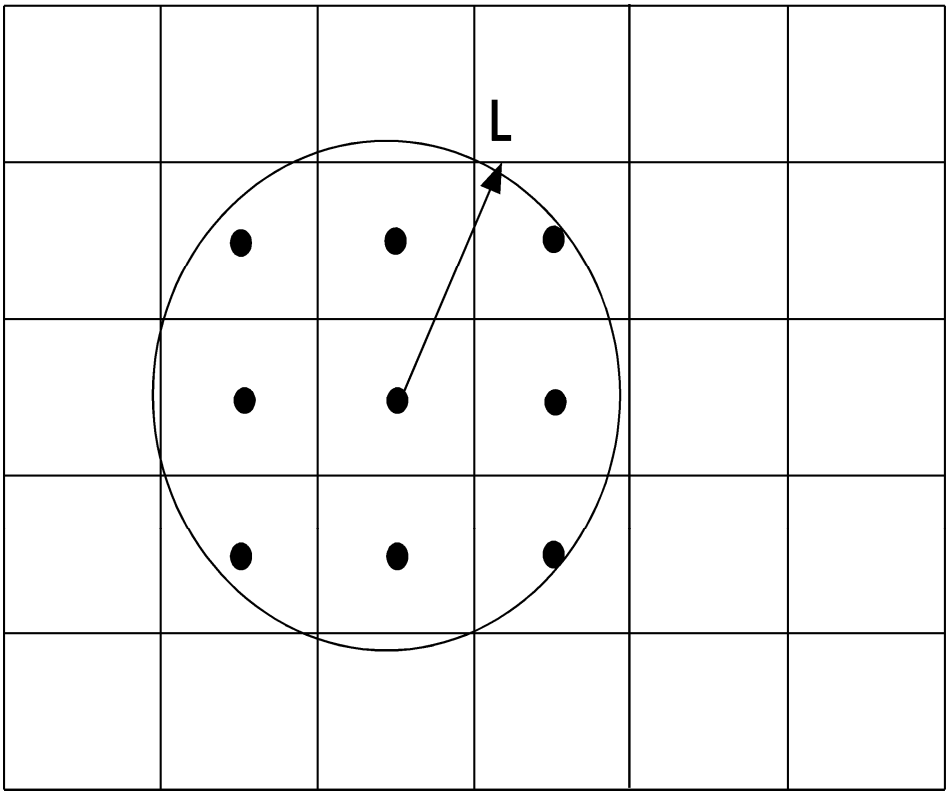
$$\dot{f}_r = \dot{f}(x_r) = \frac{1}{W_r}\int_{\Omega_r} \dot{f}_{local} w(x_r - y)dy \approx \frac{1}{W_r}\sum_{i=1}^{N_r} \dot{f}^i_{local} w_{ri} V_i$$

where

$$W_r = W(x_r) = \int w(x_r - y)dy \approx \sum_{i=1}^{N_r} w_{ri} V_i$$

$$w_{ri} = w(x_r - y_i) = \frac{1}{\left[1 + \left(\frac{\|x_r - y_i\|}{L}\right)^p\right]^q}$$

Here $\dot{f}_r$ and x_r are respectively the nonlocal rate of increase of damage and the center of the element e_r, and $\dot{f}^i_{local}$, V_i and y_i are respectively the local rate of increase of damage, the volume and the center of element e_i.



***MAT_ELASTIC_**{*OPTION*}

This is Material Type 1. This is an isotropic elastic material and is available for beam, shell, and solid elements in LS-DYNA. A specialization of this material allows the modeling of fluids.

Available options include:

<BLANK>

FLUID

such that the keyword cards appear:

***MAT_ELASTIC or MAT_001**

***MAT_ELASTIC_FLUID or MAT_001_FLUID**

The fluid option is valid for solid elements only.

Define the following card for all options:

Card	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	DA	DB	K	
Type	A8	F	F	F	F	F	F	
Default	none	none	none	none	0.0	0.0	0.0	

Define the following extra card for the FLUID option:

Card	1	2	3	4	5	6	7	8
Variable	VC	CP						
Type	F	F						
Default	none	1.0E+20						

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.
DA	Axial damping factor (used for Belytschko-Schwer beam, type 2, only).
DB	Bending damping factor (used for Belytschko-Schwer beam, type 2, only).
K	Bulk Modulus (define for fluid option only).
VC	Tensor viscosity coefficient, values between .1 and .5 should be okay.
CP	Cavitation pressure (default = 1.0e+20).

Remarks:

The axial and bending damping factors are used to damp down numerical noise. The update of the force resultants, F_i , and moment resultants, M_i , includes the damping factors:

$$F_i^{n+1} = F_i^n + \left(1 + \frac{DA}{\Delta t}\right)\Delta F_i^{n+\frac{1}{2}}$$

$$M_i^{n+1} = M_i^n + \left(1 + \frac{DB}{\Delta t}\right)\Delta M_i^{n+\frac{1}{2}}$$

The history variable labeled as "plastic strain" by LS-PREPOST is actually volumetric strain in the case of *MAT_ELASTIC.

For the fluid option the bulk modulus (K) has to be defined as Young's modulus, and Poisson's ratio is ignored. With the fluid option fluid-like behavior is obtained where the bulk modulus, K, and pressure rate, p, are given by:

$$K = \frac{E}{3(1-2\nu)}$$

$$\dot{p} = -K\ \dot{\varepsilon}_{ii}$$

and the shear modulus is set to zero. A tensor viscosity is used which acts only the deviatoric stresses, S_{ij}^{n+1}, given in terms of the damping coefficient as:

$$S_{ij}^{n+1} = VC \cdot \Delta L \cdot a \cdot \rho \dot{\varepsilon}_{ij}^{'}$$

where ΔL is a characteristic element length, a is the fluid bulk sound speed, ρ is the fluid density, and $\dot{\varepsilon}_{ij}^{'}$ is the deviatoric strain rate.

***MAT_*OPTION* TROPIC_ELASTIC**

This is Material Type 2. This material is valid for modeling the elastic-orthotropic behavior of solids, shells, and thick shells. An anisotropic option is available for solid elements. For orthotropic solids an isotropic frictional damping is available.

Available options include:

ORTHO

ANISO

such that the keyword cards appear:

***MAT_ORTHOTROPIC_ELASTIC or MAT_002** (4 cards follow)

***MAT_ANISOTROPIC_ELASTIC or MAT_002_ANIS** (5 cards follow)

Cards 1 and 2 for the ORTHO option.

Card 1

1	2	3	4	5	6	7	8
Variable	MID	RO	EA	EB	EC	PRBA	PRCA	PRCB
Type	A8	F	F	F	F	F	F	F

Card 2

Variable	GAB	GBC	GCA	AOPT	G	SIGF		
Type	F	F	F	F	F	F		

Cards 1, 2, and 3 for the ANISO option.

Card 1

	1	2	3	4	5	6	7	8
Variable	MID	RO	C11	C12	C22	C13	C23	C33
Type	A8	F	F	F	F	F	F	F

Card 2

Variable	C14	C24	C34	C44	C15	C25	C35	C45
Type	F	F	F	F	F	F	F	F

Card 3

Variable	C55	C16	C26	C36	C46	C56	C66	AOPT
Type	F	F	F	F	F	F	F	F

Cards 3/4 and 4/5 for the ORTHO/ANISO options.

Card 3/4

	1	2	3	4	5	6	7	8
Variable	XP	YP	ZP	A1	A2	A3	MACF	
Type	F	F	F	F	F	F	I	

Card 4/5

Variable	V1	V2	V3	D1	D2	D3	BETA	REF
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.

Define for the ORTHO option only:

EA	E_a, Young's modulus in a-direction.
EB	E_b, Young's modulus in b-direction.
EC	E_c, Young's modulus in c-direction (nonzero value required but not used for shells).
PRBA	ν_{ba}, Poisson's ratio ba.
PRCA	ν_{ca}, Poisson's ratio ca (if zero, defaults to PRBA for shell thickness updates).
PRCB	ν_{cb}, Poisson's ratio cb (if zero, defaults to PRBA for shell thickness updates).
GAB	G_{ab}, shear modulus ab.
GBC	G_{bc}, shear modulus bc.
GCA	G_{ca}, shear modulus ca.

Due to symmetry define the upper triangular Cij's for the ANISO option only:

C11	The 1,1 term in the 6×6 anisotropic constitutive matrix. Note that 1 corresponds to the *a* material direction
C12	The 1,2 term in the 6×6 anisotropic constitutive matrix. Note that 2 corresponds to the *b* material direction
.	.
.	.
.	.
C66	The 6,6 term in the 6×6 anisotropic constitutive matrix.

VARIABLE	DESCRIPTION

Define AOPT for both options:

AOPT — Material axes option, see Figure 2.1.

EQ.0.0: locally orthotropic with material axes determined by element nodes as shown in Figure 2.1. Nodes 1, 2, and 4 of an element are identical to the nodes used for the definition of a coordinate system as by *DEFINE_COORDINATE_NODES.

EQ.1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center; this is the a-direction. This option is for solid elements only.

EQ.2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR.

EQ.3.0: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle, BETA, from a line in the plane of the element defined by the cross product of the vector **v** with the element normal. The plane of a solid element is the midsurface between the inner surface and outer surface defined by the first four nodes and the last four nodes of the connectivity of the element, respectively.

EQ.4.0: locally orthotropic in cylindrical coordinate system with the material axes determined by a vector **v**, and an originating point, P, which define the centerline axis. This option is for solid elements only.

LT.0.0: the absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_NODES, *DEFINE_COORDINATE_SYSTEM or *DEFINE_COORDINATE_VECTOR). Available in R3 version of 971 and later.

G — Shear modulus for frequency independent damping. Frequency independent damping is based of a spring and slider in series. The critical stress for the slider mechanism is SIGF defined below. For the best results, the value of G should be 250-1000 times greater than SIGF. This option applies only to solid elements.

SIGF — Limit stress for frequency independent, frictional, damping.

XP YP ZP — Define coordinates of point **p** for AOPT = 1 and 4.

A1 A2 A3 — Define components of vector **a** for AOPT = 2.

MACF — Material axes change flag for brick elements:

EQ.1: No change, default,

EQ.2: switch material axes a and b,

EQ.3: switch material axes a and c,

EQ.4: switch material axes b and c.

VARIABLE	DESCRIPTION
V1 V2 V3	Define components of vector **v** for AOPT = 3 and 4.
D1 D2 D3	Define components of vector **d** for AOPT = 2.
BETA	Material angle in degrees for AOPT = 3, may be overridden on the element card, see *ELEMENT_SHELL_BETA or *ELEMENT_SOLID_ORTHO.
REF	Use reference geometry to initialize the stress tensor. The reference geometry is defined by the keyword: *INITIAL_FOAM_REFERENCE_GEOMETRY (see there for more details). EQ.0.0: off, EQ.1.0: on.

Remarks:

The material law that relates stresses to strains is defined as:

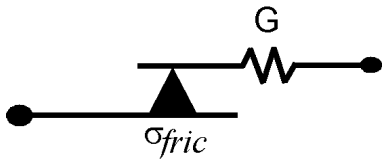
$$\underset{\sim}{C} = \underset{\sim}{T}^T \underset{\sim}{C}_L \underset{\sim}{T}$$

where $\underset{\sim}{T}$ is a transformation matrix, and $\underset{\sim}{C}_L$ is the constitutive matrix defined in terms of the material constants of the orthogonal material axes, a, b, and c. The inverse of $\underset{\sim}{C}_L$ for the orthotropic case is defined as:

$$\underset{\sim}{C}_L^{-1} = \begin{bmatrix} \frac{1}{E_a} & -\frac{\upsilon_{ba}}{E_b} & -\frac{\upsilon_{ca}}{E_c} & 0 & 0 & 0 \\ -\frac{\upsilon_{ab}}{E_a} & \frac{1}{E_b} & -\frac{\upsilon_{cb}}{E_c} & 0 & 0 & 0 \\ -\frac{\upsilon_{ac}}{E_a} & -\frac{\upsilon_{bc}}{E_b} & \frac{1}{E_c} & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1}{G_{ab}} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1}{G_{bc}} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1}{G_{ca}} \end{bmatrix}$$

Note that $\frac{\upsilon_{ab}}{E_a} = \frac{\upsilon_{ba}}{E_b}, \frac{\upsilon_{ca}}{E_c} = \frac{\upsilon_{ac}}{E_a}, \frac{\upsilon_{cb}}{E_c} = \frac{\upsilon_{bc}}{E_b}$.

The frequency independent damping is obtained by having a spring and slider in series as shown in the following sketch:



This option applies only to orthotropic solid elements and affects only the deviatoric stresses.

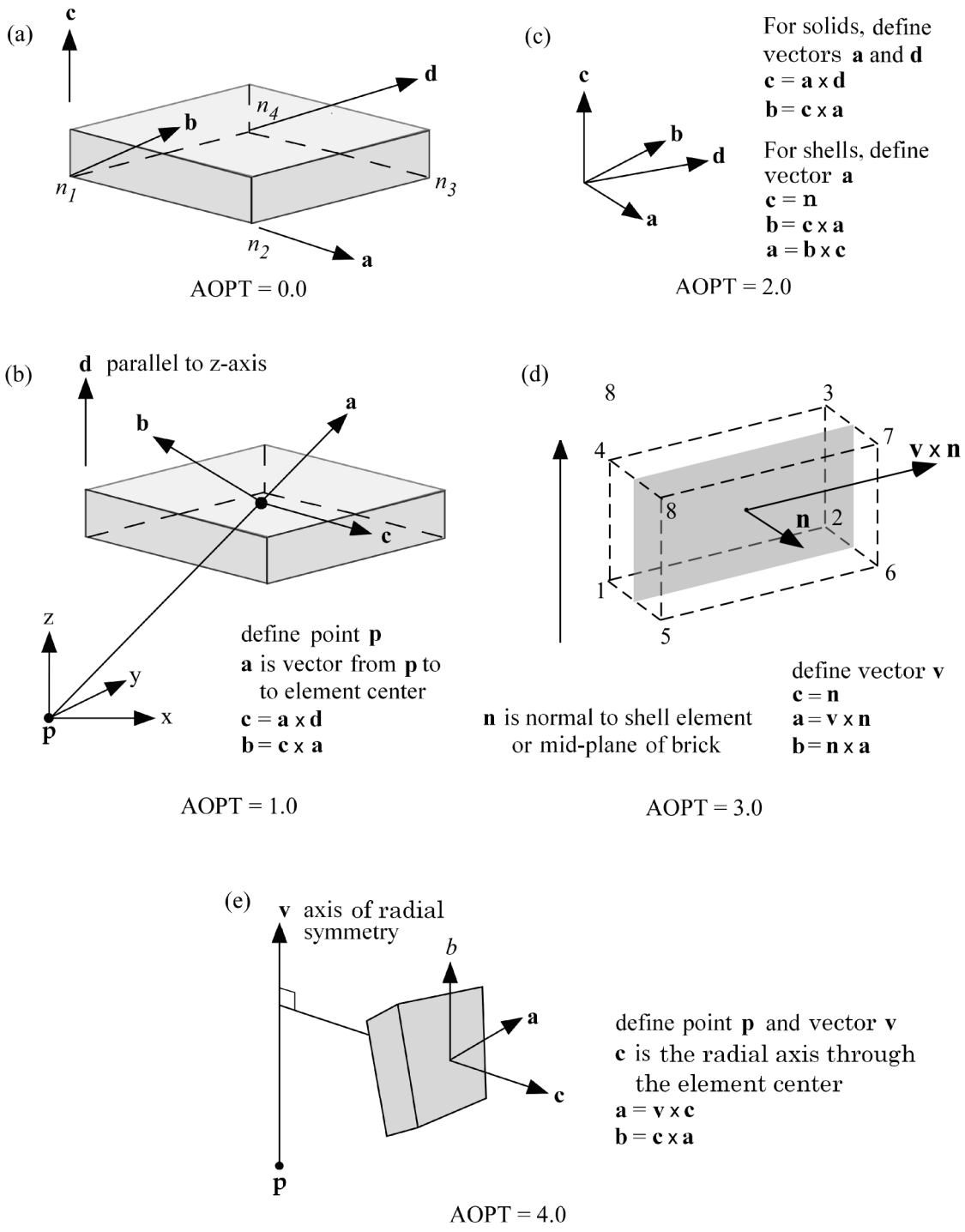


Figure 2.1. Options for determining principal material axes: (a) AOPT = 0.0, (b) AOPT = 1.0 for brick elements, (c) AOPT = 2.0, (d) AOPT = 3.0, and (e) AOPT=4.0 for brick elements.

The procedure for describing the principle material directions is explained for solid and shell elements for this material model and other anisotropic materials. We will call the material direction the **a**-**b**-**c** coordinate system. The AOPT options illustrated in Figure 2.1 can define the

a-**b**-**c** system for all elements of the parts that use the material, but this is not the final material direction. There **a**-**b**-**c** system defined by the AOPT options may be offset by a final rotation about the **c**-axis. The offset angle we call BETA.

For solid elements, the BETA angle is specified in one of two ways. When using AOPT=3, the BETA parameter defines the offset angle for all elements that use the material. The BETA parameter has no meaning for the other AOPT options. Alternatively, a BETA angle can be defined for individual solid elements as described in remark 4 for *ELEMENT_SOLID_ORTHO. The beta angle by the ORTHO option is available for all values of AOPT, and it overrides the BETA angle on the *MAT card for AOPT=3.

The directions determined by the material AOPT options may be overridden for individual elements as described in remark 2 for *ELEMENT_SOLID_ORTHO. However, be aware that for materials with AOPT=3, the final **a**-**b**-**c** system will be the system defined on the element card rotated about **c**-axis by the BETA angle specified on the *MAT card.

There are two fundamental differences between shell and solid element orthotropic materials. First, the **c**-direction is always normal to a shell element such that the **a**-direction and **b**-directions are within the plane of the element. Second, for most anisotropic materials, shell elements may have unique fiber directions within each layer through the thickness of the element so that a layered composite can be modeled with a single element.

Because shell elements have their **c**-axes defined by the element normal, AOPT=1 and AOPT=4 are not available for shells. Also, AOPT=2 requires only the vector **a** be defined since **d** is not used. The shell procedure projects the inputted **a**-direction onto each element surface.

Similar to solid elements, the **a**-**b**-**c** direction determined by AOPT is then modified by a rotation about the **c**-axis which we will call ϕ. For those materials that allow a unique rotation angle for each integration point through the element thickness, the rotation angle is calculated by

$$\phi_i = \beta + \beta_i$$

where β is a rotation for the element, and β_i is the rotation for the i'th layer of the element. The β angle can be input using the BETA parameter on the *MAT data, or will be overridden for individual elements if the BETA keyword option for *ELEMENT_SHELL is used. The β_i angles are input using the ICOMP=1 option of *SECTION_SHELL. If β or β_i is omitted, they are assumed to be zero.

All anisotropic shell materials have the BETA option on the *MAT card available for both AOPT=0 and AOPT=3, except for materials 91 and 92 which have it available for all values of AOPT, 0, 2, and 3.

All anisotropic shell materials allow a BETA angle for each integration point through the thickness, β_i, except for materials 2, 86, 91, 92, and 117. This limitation however does not preclude the use of these materials for layered composites.

The most general way to model a layered composite is to use *PART_COMPOSITE to define a material model, thickness, and material angle, β_i, for each layer of a shell element. The same capability is available through the IRID option on *SECTION_SHELL to specify a user-defined integration rule in conjunction with the PID option on *INTEGRATION_SHELL. With both methods, each layer has its own material defined and can thus have its own material direction. The *PART_COMPOSITE method is more user-friendly and is recommended.

*MAT_PLASTIC_KINEMATIC

This is Material Type 3. This model is suited to model isotropic and kinematic hardening plasticity with the option of including rate effects. It is a very cost effective model and is available for beam (Hughes-Liu and Truss), shell, and solid elements.

Card 1

Variable	MID	RO	E	PR	SIGY	ETAN	BETA	
Type	A8	F	F	F	F	F	F	
Default	none	none	none	none	none	0.0	0.0	

Card 2

Variable	SRC	SRP	FS	VP				
Type	F	F	F	F				
Default	not used	not used	not used	0.0				

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.
SIGY	Yield stress.
ETAN	Tangent modulus, see Figure 3.1.
BETA	Hardening parameter, $0 < \beta' < 1$. See comments below.
SRC	Strain rate parameter, C, for Cowper Symonds strain rate model, see below. If zero, rate effects are not considered.

VARIABLE	DESCRIPTION
SRP	Strain rate parameter, P, for Cowper Symonds strain rate model, see below. If zero, rate effects are not considered.
FS	Failure strain for eroding elements.
VP	Formulation for rate effects: EQ.0.0: Scale yield stress (default), EQ.1.0: Viscoplastic formulation

Remarks:

Strain rate is accounted for using the Cowper and Symonds model which scales the yield stress with the factor

$$1+\left(\frac{\dot{\varepsilon}}{C}\right)^{1/p}$$

where $\dot{\varepsilon}$ is the strain rate. A fully viscoplastic formulation is optional which incorporates the Cowper and Symonds formulation within the yield surface. An additional cost is incurred but the improvement allows for dramatic results. To ignore strain rate effects set both SRC and SRP to zero.

Kinematic, isotropic, or a combination of kinematic and isotropic hardening may be specified by varying β' between 0 and 1. For β' equal to 0 and 1, respectively, kinematic and isotropic hardening are obtained as shown in Figure 3.1. For isotropic hardening, $\beta' = 1$, Material Model 12, *MAT_ISOTROPIC_ELASTIC_PLASTIC, requires less storage and is more efficient. Whenever possible, Material 12 is recommended for solid elements, but for shell elements it is less accurate and thus Material 12 is not recommended in this case.

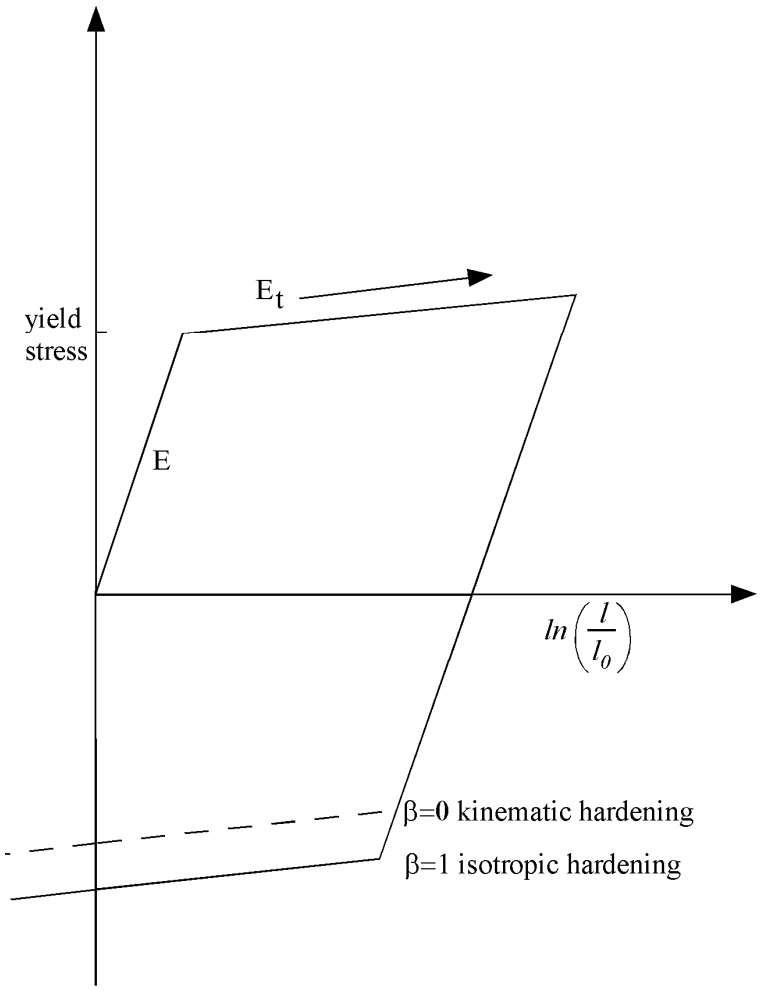


Figure 3.1. Elastic-plastic behavior with kinematic and isotropic hardening where l_0 and l are undeformed and deformed lengths of uniaxial tension specimen. E_t is the slope of the bilinear stress strain curve.

***MAT_ELASTIC_PLASTIC_THERMAL**

This is Material Type 4. Temperature dependent material coefficients can be defined. A maximum of eight temperatures with the corresponding data can be defined. A minimum of two points is needed. When this material type is used it is necessary to define nodal temperatures by activating a coupled analysis or by using another option to define the temperatures such as *LOAD_THERMAL_LOAD_CURVE, or *LOAD_THERMAL_VARIABLE.

Card 1

	1	2	3	4	5	6	7	8
Variable	MID	RO						
Type	A8	F						

Card 2

Variable	T1	T2	T3	T4	T5	T6	T7	T8
Type	F	F	F	F	F	F	F	F

Card 3

Variable	E1	E2	E3	E4	E5	E6	E7	E8
Type	F	F	F	F	F	F	F	F

Card 4

Variable	PR1	PR2	PR3	PR4	PR5	PR6	PR7	PR8
Type	F	F	F	F	F	F	F	F

No defaults are assumed.

Card 5

	1	2	3	4	5	6	7	8
Variable	ALPHA1	ALPHA2	ALPHA3	ALPHA4	ALPHA5	ALPHA6	ALPHA7	ALPHA8
Type	F	F	F	F	F	F	F	F

Card 6

Variable	SIGY1	SIGY2	SIGY3	SIGY4	SIGY5	SIGY6	SIGY7	SIGY8
Type	F	F	F	F	F	F	F	F

Card 7

Variable	ETAN1	ETAN2	ETAN3	ETAN4	ETAN5	ETAN6	ETAN7	ETAN8
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
TI	Temperatures. The minimum is 2, the maximum is 8.
EI	Corresponding Young's moduli at temperature TI.
PRI	Corresponding Poisson's ratios.
ALPHAI	Corresponding coefficients of thermal expansion.
SIGYI	Corresponding yield stresses.
EPI	Corresponding plastic hardening moduli.

Remarks:

At least two temperatures and their corresponding material properties must be defined. The analysis will be terminated if a material temperature falls outside the range defined in the input. If a thermoelastic material is considered, do not define SIGY and ETAN. The coefficient of thermal expansion is defined as the instantaneous value. Thus, the thermal strain rate becomes:

$$\dot{\varepsilon}_{ij}^{T} = \alpha \dot{T} \delta_{ij}$$

*MAT_SOIL_AND_FOAM

This is Material Type 5. This is a very simple model and works in some ways like a fluid. It should be used only in situations when soils and foams are confined within a structure or when geometric boundaries are present.

Card 1

1	2	3	4	5	6	7	8
MID	RO	G	K	A0	A1	A2	PC
A8	F	F	F	F	F	F	F

Card 2

Variable	VCR	REF						
Type	F	F						

Card 3

Variable	EPS1	EPS2	EPS3	EPS4	EPS5	EPS6	EPS7	EPS8
Type	F	F	F	F	F	F	F	F

Card 4

Variable	EPS9	EPS10						
Type	F	F						

Card 5

	1	2	3	4	5	6	7	8
Variable	P1	P2	P3	P4	P5	P6	P7	P8
Type	F	F	F	F	F	F	F	F

Card 6

Variable	P9	P10						
Type	F	F						

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
G	Shear modulus.
K	Bulk modulus for unloading used for VCR=0.0.
A0	Yield function constant for plastic yield function below.
A1	Yield function constant for plastic yield function below.
A2	Yield function constant for plastic yield function below.
PC	Pressure cutoff for tensile fracture (< 0).
VCR	Volumetric crushing option: EQ.0.0: on, EQ.1.0: loading and unloading paths are the same.
REF	Use reference geometry to initialize the pressure. The reference geometry is defined by the keyword:*INITIAL_FOAM_REFERENCE_GEOMETRY. This option does not initialize the deviatoric stress state. EQ.0.0: off, EQ.1.0: on.

VARIABLE	DESCRIPTION
EPS1,.....	Volumetric strain values (natural logarithmic values), see comments below. A maximum of 10 values are allowed and a minimum of 2 values are necessary. The tabulated values must completely cover the expected values in the analysis. If the first value is not for a volumetric strain value of zero then the point (0.0,0.0) will be automatically generated and up to a further nine additional values may be defined.
P1, P2,..PN	Pressures corresponding to volumetric strain values.

Remarks:

Pressure is positive in compression. Volumetric strain is given by the natural log of the relative volume and is negative in compression. Relative volume is a ratio of the current volume to the initial volume at the start of the calculation. The tabulated data should be given in order of increasing compression. If the pressure drops below the cutoff value specified, it is reset to that value. For a detailed description we refer to Kreig [1972].

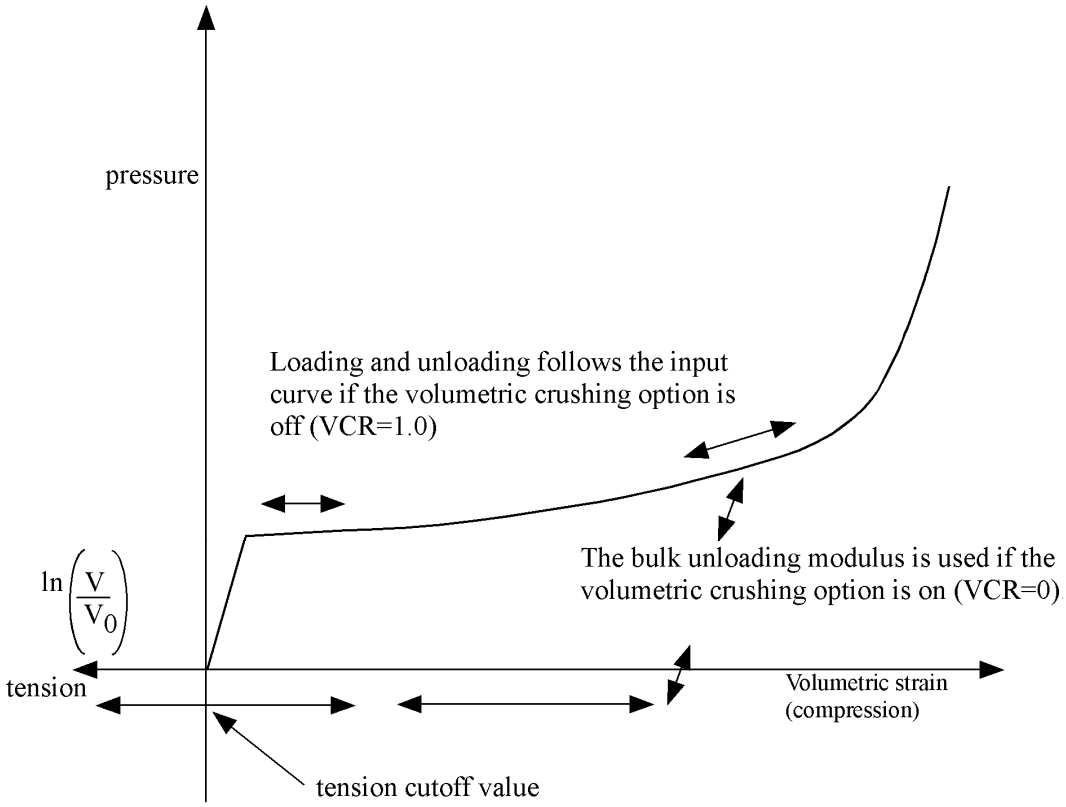


Figure 5.1. Pressure versus volumetric strain curve for soil and crushable foam model. The volumetric strain is given by the natural logarithm of the relative volume, V.

The deviatoric perfectly plastic yield function, ϕ, is described in terms of the second invariant J_2,

$$J_2 = \frac{1}{2} s_{ij} s_{ij},$$

pressure, p, and constants a_0, a_1, and a_2 as:

$$\phi = J_2 - \left[a_0 + a_1 p + a_2 p^2 \right].$$

On the yield surface $J_2 = \frac{1}{3}\sigma_y^2$ where σ_y is the uniaxial yield stress, i.e.,

$$\sigma_y = \left[\; 3\left(a_0 + a_1 p + a_2 p^2 \right) \right]^{1/2}$$

There is no strain hardening on this surface.

To eliminate the pressure dependence of the yield strength, set:

$$a_1 = a_2 = 0 \qquad a_0 = \frac{1}{3}\sigma_y^2.$$

This approach is useful when a von Mises type elastic-plastic model is desired for use with the tabulated volumetric data.

The history variable labeled as “plastic strain” by LS-Prepost is actually $\ln(V/V_0)$ in the case of *MAT_SOIL_AND_FOAM.

*MAT_VISCOELASTIC

This is Material Type 6. This model allows the modeling of viscoelastic behavior for beams (Hughes-Liu), shells, and solids. Also see *MAT_GENERAL_VISCOELASTIC for a more general formulation.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	BULK	G0	GI	BETA		
Type	A8	F	F	F	F	F		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
BULK	Elastic bulk modulus.
G0	Short-time shear modulus, see equations below.
GI	Long-time (infinite) shear modulus, G_∞.
BETA	Decay constant.

Remarks:

The shear relaxation behavior is described by [Hermann and Peterson, 1968]:

$$G(t) = G_\infty + (G_0 - G_\infty)\, e^{-\beta t}$$

A Jaumann rate formulation is used

$$\overset{\nabla}{\sigma}'_{ij} = 2 \int_0^t G(t-\tau)\, D'_{ij}(\tau)\, d\tau$$

where the prime denotes the deviatoric part of the stress rate, $\overset{\nabla}{\sigma}_{ij}$, and the strain rate, D_{ij}.

***MAT_BLATZ-KO_RUBBER**

This is Material Type 7. This one parameter material allows the modeling of nearly incompressible continuum rubber. The Poisson's ratio is fixed to 0.463.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	G	REF				
Type	A8	F	F	F				

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
G	Shear modulus.
REF	Use reference geometry to initialize the stress tensor. The reference geometry is defined by the keyword:*INITIAL_FOAM_REFERENCE_GEOMETRY (see there for more details). EQ.0.0: off, EQ.1.0: on.

Remarks:

The second Piola-Kirchhoff stress is computed as

$$S_{ij} = G\left[\frac{1}{V}C_{ij} - V^{\left(\frac{1}{1-2v}\right)}\delta_{ij}\right]$$

where V is the relative volume defined as being the ratio of the current volume to the initial volume, C_{ij} is the right Cauchy-Green strain tensor, and v is Poisson's ratio, which is set to .463 internally. This stress measure is transformed to the Cauchy stress, σ_{ij}, according to the relationship

$$\sigma_{ij} = V^{-1}F_{ik}F_{jl}S_{lk}$$

where F_{ij} is the deformation gradient tensor. Also see Blatz and Ko [1962].

*MAT_HIGH_EXPLOSIVE_BURN

This is Material Type 8. It allows the modeling of the detonation of a high explosive. In addition an equation of state must be defined. See Wilkins [1969] and Giroux [1973].

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	D	PCJ	BETA	K	G	SIGY
Type	A8	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
D	Detonation velocity.
PCJ	Chapman-Jouget pressure.
BETA	Beta burn flag, BETA (see comments below): EQ.0.0: beta + programmed burn, EQ.1.0: beta burn only, EQ.2.0: programmed burn only.
K	Bulk modulus (BETA=2.0 only).
G	Shear modulus (BETA=2.0 only).
SIGY	σ_y, yield stress (BETA=2.0 only).

Remarks:

Burn fractions, F, which multiply the equations of states for high explosives, control the release of chemical energy for simulating detonations. At any time, the pressure in a high explosive element is given by:

$$p = Fp_{eos}(V, E)$$

where p_{eos}, is the pressure from the equation of state (either types 2, 3, or 14), V is the relative volume, and E is the internal energy density per unit initial volume.

In the initialization phase, a lighting time t_l is computed for each element by dividing the distance from the detonation point to the center of the element by the detonation velocity *D*. If

multiple detonation points are defined, the closest detonation point determines t_l. The burn fraction F is taken as the maximum

$$F = \max(F_1, F_2)$$

where

$$F_1 = \begin{cases} \dfrac{2\left(t - t_l\right) DA_{e_{\max}}}{3v_e} & \text{if } t > t_l \\ \\ 0 & \text{if } t \le t_l \end{cases}$$

$$F_2 = \beta = \frac{1 - V}{1 - V_{CJ}}$$

where V_{CJ} is the Chapman-Jouguet relative volume and t is current time. If F exceeds 1, it is reset to 1. This calculation of the burn fraction usually requires several time steps for F to reach unity, thereby spreading the burn front over several elements. After reaching unity, F is held constant. This burn fraction calculation is based on work by Wilkins [1964] and is also discussed by Giroux [1973].

If the beta burn option is used, BETA=1.0, any volumetric compression will cause detonation and

$$F = F_2$$

and F_1 is not computed.

If programmed burn is used, BETA=2.0, the explosive model will behave as an elastic perfectly plastic material if the bulk modulus, shear modulus, and yield stress are defined. Therefore, with this option the explosive material can compress without causing detonation.

As an option, the high explosive material can behave as an elastic perfectly-plastic solid prior to detonation. In this case we update the stress tensor, to an elastic trial stress, $^*s_{ij}^{n+1}$,

$$^*s_{ij}^{n+1} = s_{ij}^n + s_{ip}\Omega_{pj} + s_{jp}\Omega_{pi} + 2G\dot{\varepsilon}'_{ij}\, dt$$

where G is the shear modulus, and $\dot{\varepsilon}'_{ij}$ is the deviatoric strain rate. The von Mises yield condition is given by:

$$\phi = J_2 - \frac{\sigma_y^2}{3}$$

where the second stress invariant, J_2, is defined in terms of the deviatoric stress components as

$$J_2 = \frac{1}{2} s_{ij} s_{ij}$$

and the yield stress is σ_y. If yielding has occurred, i.e., $\phi > 0$, the deviatoric trial stress is scaled to obtain the final deviatoric stress at time n+1:

$$s_{ij}^{n+1} = \frac{\sigma_y}{\sqrt{3J_2}} * s_{ij}^{n+1}$$

If $\phi \le 0$, then

$$s_{ij}^{n+1} = *s_{ij}^{n+1}$$

Before detonation pressure is given by the expression

$$p^{n+1} = K\left(\frac{1}{V^{n+1}} - 1\right)$$

where K is the bulk modulus. Once the explosive material detonates:

$$s_{ij}^{n+1} = 0$$

and the material behaves like a gas.

***MAT_NULL**

This is Material Type 9. This material allows equations of state to be considered without computing deviatoric stresses. Optionally, a viscosity can be defined. Also, erosion in tension and compression is possible.

Sometimes it is advantageous to model contact surfaces via shell elements which are not part of the structure, but are necessary to define areas of contact within nodal rigid bodies or between nodal rigid bodies.

Beams and shells that use this material type are completely bypassed in the element processing; however, the mass of the null shell elements is computed and added to the nodal points which define the connectivity. However, the mass of null beams is ignored if the value of the density is less than 1.e-11. The Young's modulus and Poisson's ratio are used only for setting the contact interface stiffnesses, and it is recommended that reasonable values be input.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	PC	MU	TEROD	CEROD	YM	PR
Type	A8	F	F	F	F	F	F	F
Defaults	none	none	0.0	0.0	0.0	0.0	0.0	0.0

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
PC	Pressure cutoff (≤ 0.0). See Remark 4.
MU	Dynamic viscosity coefficient μ (optional). See Remark 1.
TEROD	Relative volume, $\frac{V}{V_0}$, for erosion in tension. Typically, use values greater than unity. If zero, erosion in tension is inactive.
CEROD	Relative volume, $\frac{V}{V_0}$, for erosion in compression. Typically, use values less than unity. If zero, erosion in compression is inactive.

VARIABLE	DESCRIPTION
YM	Young's modulus (used for null beams and shells only)
PR	Poisson's ratio (used for null beams and shells only)

Remarks:

1. The null material must be used with an equation-of-state. Pressure cutoff is negative in tension. A (deviatoric) viscous stress of the form

$$\sigma'_{ij} = 2\mu\dot{\varepsilon}'_{ij}$$

$$\left[\frac{N}{m^2}\right] \sim \left[\frac{N}{m^2}s\right]\left[\frac{1}{s}\right]$$

   is computed for nonzero μ where $\dot{\varepsilon}'_{ij}$ is the deviatoric strain rate. μ is the dynamic viscosity. For example, in SI unit system, μ may have a unit of [Pa*s].

2. Null material has no shear stiffness and hourglass control must be used with great care. In some applications, the default hourglass coefficient might lead to significant energy losses. In general for fluid(s), the hourglass coefficient QM should be small (in the range 1.0E-4 to 1.0E-6 in the SI unit system for the standard default IHQ choice).

3. The Null material has no yield strength and behaves in a fluid-like manner.

4. The cut-off pressure, PC, must be defined to allow for a material to "numerically" cavitate. In other words, when a material undergoes dilatation above certain magnitude, it should no longer be able to resist this dilatation. Since dilatation stress or pressure is negative, setting PC limit to a very small negative number would allow for the material to cavitate once the pressure in the material goes below this negative value.

***MAT_ELASTIC_PLASTIC_HYDRO_**{*OPTION*}

This is Material Type 10. This material allows the modeling of an elastic-plastic hydrodynamic material.

Available options include:

<**BLANK**>

SPALL

The keyword card can appear in two ways:

***MAT_ELASTIC_PLASTIC_HYDRO or MAT_010**

***MAT_ELASTIC_PLASTIC_HYDRO_SPALL or MAT_010_SPALL**

Card 1

Variable	MID	RO	G	SIGY	EH	PC	FS	CHARL
Type	A8	F	F	F	F	F	F	F
Default	none	none	none	0.0	0.0	-∞	0.0	0.0

Define this card if and only if the SPALL option is specified.

Optional

Variable	A1	A2	SPALL					
Type	F	F	F					

Card 2

Variable	EPS1	EPS2	EPS3	EPS4	EPS5	EPS6	EPS7	EPS8
Type	F	F	F	F	F	F	F	F

Card 3	1	2	3	4	5	6	7	8
Variable	EPS9	EPS10	EPS11	EPS12	EPS13	EPS14	EPS15	EPS16
Type	F	F	F	F	F	F	F	F

Card 4

Variable	ES1	ES2	ES3	ES4	ES5	ES6	ES7	ES8
Type	F	F	F	F	F	F	F	F

Card 5

Variable	ES9	ES10	ES11	ES12	ES13	ES14	ES15	ES16
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
G	Shear modulus.
SIGY	Yield stress, see comment below.
EH	Plastic hardening modulus, see definition below.
PC	Pressure cutoff (≤ 0.0). If zero, a cutoff of $-\infty$ is assumed.
FS	Failure strain for erosion.
CHARL	Characteristic element thickness for deletion. This applies to 2D solid elements that lie on a boundary of a part. If the boundary element thins down due to stretching or compression, and if it thins to a value less than CHARL, the element will be deleted. The primary application of this option is to predict the break-up of axisymmetric shaped charge jets.

VARIABLE	DESCRIPTION
A1	Linear pressure hardening coefficient.
A2	Quadratic pressure hardening coefficient.
SPALL	Spall type: EQ.0.0: default set to "1.0", EQ.1.0: p ≧ PC, EQ.2.0: if $\sigma_{max} \geq$ -PC element spalls and tension, p < 0, is never allowed, EQ.3.0: p < PC element spalls and tension, p < 0, is never allowed.
EPS	Effective plastic strain (True). Define up to 16 values. Care must be taken that the full range of strains expected in the analysis is covered. Linear extrapolation is used if the strain values exceed the maximum input value.
ES	Effective stress. Define up to 16 values.

Remarks:

If ES and EPS are undefined, the yield stress and plastic hardening modulus are taken from SIGY and EH. In this case, the bilinear stress-strain curve shown in Figure 3.1. is obtained with hardening parameter, β=1. The yield strength is calculated as

$$\sigma_y = \sigma_0 + E_h \bar{\varepsilon}^p + (a_1 + pa_2)\max[p, 0]$$

The quantity E_h is the plastic hardening modulus defined in terms of Young's modulus, E, and the tangent modulus, E_t, as follows

$$E_h = \frac{E_t E}{E - E_t} .$$

and p is the pressure taken as positive in compression.

If ES and EPS are specified, a curve like that shown in Figure 10.1. may be defined. Effective stress is defined in terms of the deviatoric stress tensor, s_{ij}, as:

$$\bar{\sigma} = \left(\frac{3}{2} s_{ij} \, s_{ij} \right)^{1/2}$$

and effective plastic strain by:

$$\bar{\varepsilon}^p = \int_0^t \left(\frac{2}{3} \, D_{ij}^p \, D_{ij}^p \right)^{1/2} dt ,$$

where t denotes time and D_{ij}^p is the plastic component of the rate of deformation tensor. In this case the plastic hardening modulus on Card 1 is ignored and the yield stress is given as

$$\sigma_y = f\left(\bar{\varepsilon}^p\right),$$

where the value for $f\left(\bar{\varepsilon}^p\right)$ is found by interpolation from the data curve.

A choice of three spall models is offered to represent material splitting, cracking, and failure under tensile loads. The pressure limit model, SPALL=1, limits the hydrostatic tension to the specified value, p_{cut}. If pressures more tensile than this limit are calculated, the pressure is reset to p_{cut}. This option is not strictly a spall model, since the deviatoric stresses are unaffected by the pressure reaching the tensile cutoff, and the pressure cutoff value, p_{cut}, remains unchanged throughout the analysis. The maximum principal stress spall model, SPALL=2, detects spall if the maximum principal stress, σ_{max}, exceeds the limiting value $-p_{cut}$. Note that the negative sign is required because p_{cut} is measu red positive in compression, while σ_{max} is positive in tension. Once spall is detected with this model, the deviatoric stresses are reset to zero, and no hydrostatic tension ($p<0$) is permitted. If tensile pressures are calculated, they are reset to 0 in the spalled material. Thus, the spalled material behaves as a rubble or incohesive material. The hydrostatic tension spall model, SPALL=3, detects spall if the pressure becomes more tensile than the specified limit, p_{cut}. Once spall is detected the deviatoric stresses are reset to zero, and nonzero values of pressure are required to be compressive (positive). If hydrostatic tension ($p<0$) is subsequently calculated, the pressure is reset to 0 for that element.

This model is applicable to a wide range of materials, including those with pressure-dependent yield behavior. The use of 16 points in the yield stress versus effective plastic strain curve allows complex post-yield hardening behavior to be accurately represented. In addition, the incorporation of an equation of state permits accurate modeling of a variety of different materials. The spall model options permit incorporation of material failure, fracture, and disintegration effects under tensile loads.

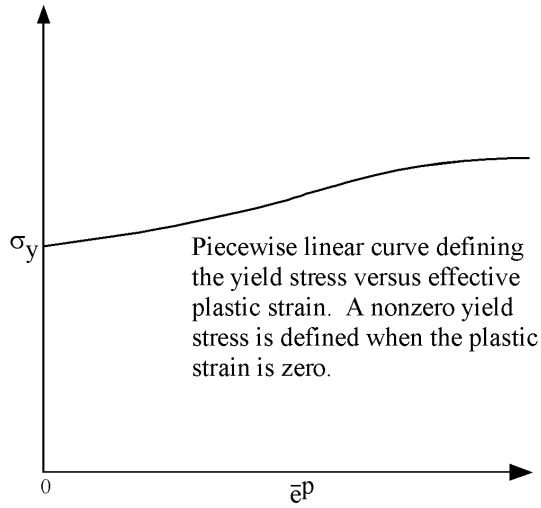


Figure 10.1. Effective stress versus effective plastic strain curve.

***MAT_STEINBERG**

This is Material Type 11. This material is available for modeling materials deforming at very high strain rates ($>10^5$) and can be used with solid elements. The yield strength is a function of temperature and pressure. An equation of state determines the pressure.

Card 1

1	2	3	4	5	6	7	8
MID	RO	G0	SIGO	BETA	N	GAMA	SIGM
A8	F	F	F	F	F	F	F

Card 2

B	BP	H	F	A	TMO	GAMO	SA
F	F	F	F	F	F	F	F

Card 3

PC	SPALL	RP	FLAG	MMN	MMX	ECO	EC1
F	F	F	F	F	F	F	F

Card 4

EC2	EC3	EC4	EC5	EC6	EC7	EC8	EC9
F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
G0	Basic shear modulus.
SIGO	σ_o, see defining equations.
BETA	β, see defining equations.
N	n, see defining equations.
GAMA	γ_i, initial plastic strain, see defining equations.
SIGM	σ_m, see defining equations.
B	b, see defining equations.
BP	b′, see defining equations.
H	h, see defining equations.
F	f, see defining equations.
A	Atomic weight (if = 0.0, R′ must be defined).
TMO	T_{mo}, see defining equations.
GAMO	γ_o, see defining equations.
SA	a, see defining equations.
PC	Pressure cutoff (default=-1.e+30)
SPALL	Spall type: EQ. 0.0: default set to "2.0", EQ. 1.0: $p \geqq PC$, EQ. 2.0: if $\sigma_{max} \geq$ -PC element spalls and tension, $p < 0$, is never allowed, EQ. 3.0: $p <$ PC element spalls and tension, $p < 0$, is never allowed.
RP	R′. If $R' \neq 0.0$, A is not defined.

VARIABLE	DESCRIPTION

FLAG	Set to 1.0 for μ coefficients for the cold compression energy fit. Default is η.
MMN	μ_{min} or η_{min}. Optional μ or η minimum value.
MMX	μ_{max} or η_{max}. Optional μ or η maximum value.
EC0,...EC9	Cold compression energy coefficients (optional).

Remarks:

Users who have an interest in this model are encouraged to study the paper by Steinberg and Guinan which provides the theoretical basis. Another useful reference is the KOVEC user's manual.

In terms of the foregoing input parameters, we define the shear modulus, G, before the material melts as:

$$G = G_0\left[1 + bpV^{1/3} - h\left(\frac{E_i - E_c}{3R'} - 300\right)\right]e^{-fE_i/E_m - E_i}$$

where p is the pressure, V is the relative volume, E_c is the cold compression energy:

$$E_c(x) = \int_0^x p\,dx - \frac{900\,R'\exp(ax)}{(1-x)^{2(\gamma_0 - a - 1/2)}},$$

$$x = 1 - V,$$

and E_m is the melting energy:

$$E_m(x) = E_c(x) + 3R'T_m(x)$$

which is in terms of the melting temperature $T_m(x)$:

$$T_m(x) = \frac{T_{mo}\exp(2ax)}{V^{2(\gamma_o - a - 1/3)}}$$

and the melting temperature at $\rho = \rho_o$, T_{mo}.

In the above equation R′ is defined by

$$R' = \frac{R\rho}{A}$$

where R is the gas constant and A is the atomic weight. If R′ is not defined, LS-DYNA computes it with R in the cm-gram-microsecond system of units.

The yield strength σ_y is given by:

$$\sigma_y = \sigma_0' \left[1 + b'pV^{1/3} - h\left(\frac{E_i - E_c}{3R'} - 300 \right) \right] e^{-f E_i / E_m - E_i}$$

if E_m exceeds E_i. Here, σ_0' is given by:

$$\sigma_0' = \sigma_0 \left[1 + \beta\left(\gamma_i + \bar{\varepsilon}^p \right) \right]^n$$

where σ_0 is the initial yield stress and γ_i is the initial plastic strain. If the work-hardened yield stress σ_0' exceeds σ_m, σ_0' is set equal to σ_m. After the materials melt, σ_y and G are set to one half their initial value.

If the coefficients EC0,...,EC9 are not defined above, LS-DYNA will fit the cold compression energy to a ten term polynomial expansion either as a function of μ or η depending on the input variable, FLAG, as:

$$E_c\left(\eta^i\right) = \sum_{i=0}^{9} EC_i \eta^i$$

$$E_c\left(\mu^i\right) = \sum_{i=0}^{9} EC_i \mu^i$$

where EC_i is the ith coefficient and:

$$\eta = \frac{\rho}{\rho_o}$$

$$\mu = \frac{\rho}{\rho_o} - 1$$

A linear least squares method is used to perform the fit.

A choice of three spall models is offered to represent material splitting, cracking, and failure under tensile loads. The pressure limit model, SPALL=1, limits the hydrostatic tension to the specified value, p_{cut}. If pressures more tensile than this limit are calculated, the pressure is reset to p_{cut}. This option is not strictly a spall model, since the deviatoric stresses are unaffected by the pressure reaching the tensile cutoff, and the pressure cutoff value, p_{cut}, remains unchanged throughout the analysis. The maximum principal stress spall model, SPALL=2, detects spall if the maximum principal stress, σ_{max}, exceeds the limiting value $-p_{cut}$. Note that the negative sign is required because p_{cut} is measured positive in compression, while σ_{max} is positive in tension. Once spall is detected with this model, the deviatoric stresses are reset to zero, and no hydrostatic tension ($p<0$) is permitted. If tensile pressures are calculated, they are reset to 0 in the spalled material. Thus, the spalled material behaves as a rubble or incohesive material. The hydrostatic tension spall model, SPALL=3, detects spall if the pressure becomes

more tensile than the specified limit, p_{cut}. Once spall is detected the deviatoric stresses are reset to zero, and nonzero values of pressure are required to be compressive (positive). If hydrostatic tension ($p<0$) is subsequently calculated, the pressure is reset to 0 for that element.

This model is applicable to a wide range of materials, including those with pressure-dependent yield behavior. In addition, the incorporation of an equation of state permits accurate modeling of a variety of different materials. The spall model options permit incorporation of material failure, fracture, and disintegration effects under tensile loads.

*MAT_STEINBERG_LUND

This is Material Type 11. This material is a modification of the Steinberg model above to include the rate model of Steinberg and Lund [1989]. An equation of state determines the pressure.

The keyword cards can appear in two ways:

***MAT_STEINBERG_LUND or MAT_011_LUND**

Card 1

Variable	MID	RO	G0	SIGO	BETA	N	GAMA	SIGM
Type	A8	F	F	F	F	F	F	F

Card 2

Variable	B	BP	H	F	A	TMO	GAMO	SA
Type	F	F	F	F	F	F	F	F

Card 3

Variable	PC	SPALL	RP	FLAG	MMN	MMX	ECO	EC1
Type	F	F	F	F	F	F	F	F

Card 4

Variable	EC2	EC3	EC4	EC5	EC6	EC7	EC8	EC9
Type	F	F	F	F	F	F	F	F

Card 5	1	2	3	4	5	6	7	8
Variable	UK	C1	C2	YP	YA	YM		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
G0	Basic shear modulus.
SIGO	σ_o, see defining equations.
BETA	β, see defining equations.
N	n, see defining equations.
GAMA	γ_i, initial plastic strain, see defining equations.
SIGM	σ_m, see defining equations.
B	b, see defining equations.
BP	b′, see defining equations.
H	h, see defining equations.
F	f, see defining equations.
A	Atomic weight (if = 0.0, R′ must be defined).
TMO	T_{mo}, see defining equations.
GAMO	γ_o, see defining equations.
SA	a, see defining equations.
PC	p_{cut} or $-\sigma_f$ (default=-1.e+30)

VARIABLE	DESCRIPTION
SPALL	Spall type: EQ. 0.0: default set to "2.0", EQ. 1.0: $p \geqq p_{min}$, EQ. 2.0: if $\sigma_{max} \geq -p_{min}$ element spalls and tension, $p < 0$, is never allowed, EQ. 3.0: $p < -p_{min}$ element spalls and tension, $p < 0$, is never allowed.
RP	R′. If R′ ≠ 0.0, A is not defined.
FLAG	Set to 1.0 for μ coefficients for the cold compression energy fit. Default is η.
MMN	μ_{min} or η_{min}. Optional μ or η minimum value.
MMX	μ_{max} or η_{max}. Optional μ or η maximum value.
EC0,...EC9	Cold compression energy coefficients (optional).
UK	Activation energy for rate dependent model.
C1	Exponent prefactor in rate dependent model.
C2	Coefficient of drag term in rate dependent model.
YP	Peierls stress for rate dependent model.
YA	Athermal yield stress for rate dependent model.
YMAX	Work hardening maximum for rate model.

Remarks:

This model is similar in theory to the *MAT_STEINBERG above but with the addition of rate effects. When rate effects are included, the yield stress is given by:

$$\sigma_y = \left\{ Y_T\left(\dot{\varepsilon}_p, T\right) + Y_A f\left(\varepsilon_p\right) \right\} \frac{G\left(p,T\right)}{G_0}$$

There are two imposed limits on the yield stress. The first is on the thermal yield stress:

$$Y_A f\left(\varepsilon_p\right) = Y_A \left[1 + \beta\left(\gamma_i + \varepsilon^p\right) \right]^n \leq Y_{max}$$

and the second is on the thermal part:

$$Y_T \leq Y_P$$

*MAT_ISOTROPIC_ELASTIC_PLASTIC

This is Material Type 12. This is a very low cost isotropic plasticity model for three-dimensional solids. In the plane stress implementation for shell elements, a one-step radial return approach is used to scale the Cauchy stress tensor to if the state of stress exceeds the yield surface. This approach to plasticity leads to inaccurate shell thickness updates and stresses after yielding. This is the only model in LS-DYNA for plane stress that does not default to an iterative approach.

Card	1	2	3	4	5	6	7	8
Variable	MID	RO	G	SIGY	ETAN	BULK		
Type	A8	F	F	F	F	F		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
G	Shear modulus.
SIGY	Yield stress.
ETAN	Plastic hardening modulus.
BULK	Bulk modulus, K.

Remarks:

Here the pressure is integrated in time

$$\dot{p} = -K \ \dot{\varepsilon}_{ii}$$

where $\dot{\varepsilon}_{ii}$ is the volumetric strain rate.

*MAT_ISOTROPIC_ELASTIC_FAILURE

This is Material Type 13. This is a non-iterative plasticity with simple plastic strain failure model.

Card 1

	1	2	3	4	5	6	7	8
Variable	MID	RO	G	SIGY	ETAN	BULK		
Type	A8	F	F	F	F	F		
Default	None	none	none	none	0.0	none		

Card 2

Variable	EPF	PRF	REM	TREM				
Type	F	F	F	F				
Default	None	0.0	0.0	0.0				

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
G	Shear modulus.
SIGY	Yield stress.
ETAN	Plastic hardening modulus.
BULK	Bulk modulus.
EPF	Plastic failure strain.
PRF	Failure pressure (≤ 0.0).

VARIABLE	DESCRIPTION
REM	Element erosion option: EQ.0.0: failed element eroded after failure, NE.0.0: element is kept, no removal except by Δt below.
TREM	Δt for element removal: EQ.0.0: Δt is not considered (default), GT.0.0: element eroded if element time step size falls below Δt.

Remarks:

When the effective plastic strain reaches the failure strain or when the pressure reaches the failure pressure, the element loses its ability to carry tension and the deviatoric stresses are set to zero, i.e., the material behaves like a fluid. If Δt for element removal is defined the element removal option is ignored.

The element erosion option based on Δt must be used cautiously with the contact options. Nodes to surface contact is recommended with all nodes of the eroded brick elements included in the node list. As the elements are eroded the mass remains and continues to interact with the master surface.

*MAT_SOIL_AND_FOAM_FAILURE

This is Material Type 14. The input for this model is the same as for *MATERIAL_SOIL_AND_FOAM (Type 5); however, when the pressure reaches the failure pressure, the element loses its ability to carry tension. It should be used only in situations when soils and foams are confined within a structure or when geometric boundaries are present.

*MAT_JOHNSON_COOK

This is Material Type 15. The Johnson/Cook strain and temperature sensitive plasticity is sometimes used for problems where the strain rates vary over a large range and adiabatic temperature increases due to plastic heating cause material softening. When used with solid elements this model requires an equation-of-state. If thermal effects and damage are unimportant, the much less expensive *MAT_SIMPLIFIED_JOHNSON_COOK model is recommended. The simplified model can be used with beam elements.

Card 1

1	2	3	4	5	6	7	8
Variable	MID	RO	G	E	PR	DTF	VP	RATEOP
Type	A8	F	F	F	F	F	F	F
Default	none	none	none	none	none	0.0	0.0	0.0

Card 2

Variable	A	B	N	C	M	TM	TR	EPSO
Type	F	F	F	F	F	F	F	F
Default	none	0.0	0.0	0.0	none	none	none	none

Card 3

Variable	CP	PC	SPALL	IT	D1	D2	D3	D4
Type	F	F	F	F	F	F	F	F
Default	none	0.0	2.0	0.0	0.0	0.0	0.0	0.0

Card 4	1	2	3	4	5	6	7	8
Variable	D5	C2/P		EFMIN				
Type	F	F		F				
Default	0.0	0.0		0.000001				

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
G	Shear modulus
E	Young's Modulus (shell elements only)
PR	Poisson's ratio (shell elements only)
DTF	Minimum time step size for automatic element deletion (shell elements). The element will be deleted when the solution time step size drops below DTF*TSSFAC where TSSFAC is the time step scale factor defined by *CONTROL_TIMESTEP.
VP	Formulation for rate effects: EQ.0.0: Scale yield stress (default), EQ.1.0: Viscoplastic formulation.
RATEOP	Optional forms of strain-rate term: EQ.0.0: Log-Linear Johnson-Cook (default), EQ.1.0: Log-Quadratic Huh-Kang (2 parameters), EQ.2.0: Exponential Allen-Rule-Jones, EQ.3.0: Exponential Cowper-Symonds (2 parameters).
A	See equations below.
B	See equations below.
N	See equations below.
C	See equations below.
M	See equations below.

VARIABLE	DESCRIPTION
TM	Melt temperature
TR	Room temperature
EPSO	Quasi-static threshold strain rate. Ideally, this value represents the highest strain rate for which no rate adjustment to the flow stress is needed, and is input in units of 1/model time units. For example, if strain rate effects on the flow stress first become apparent at strain rates greater than 1E-02 seconds^{-1} and the system of units for the model input is kg, mm, msec, then EPSO should be set to 1E-05 [msec^{-1}]
CP	Specific heat (superseded by heat capacity in *MAT_THERMAL_*OPTION* if a coupled thermal/structural analysis)
PC	Failure stress or pressure cutoff (PC < 0.0)
SPALL	Spall type: EQ. 0.0: default set to "2.0", EQ. 1.0: Tensile pressure is limited by PC, i.e., p is always ≥ PC, EQ. 2.0: σ_{max} ≥ -PC triggers shell element deletion and tensile stresses to be reset to zero in solid elements. Only compressive stresses are subsequently allowed in solids, EQ. 3.0: p < PC triggers shell element deletion and pressure to be reset to zero in solid elements. Tensile pressure is subsequently disallowed in solids.
IT	Plastic strain iteration option. This input applies to solid elements only since it is always necessary to iterate for the shell element plane stress condition. EQ. 0.0: no iterations (default), EQ. 1.0: accurate iterative solution for plastic strain. Much more expensive than default.
D1-D5	Failure parameters, see equations below.
C2/P	Optional strain-rate parameter for Huh-Kang (C2) or Cowper-Symonds (P) forms; see equations below.
EFMIN	The minimum required strain for failure.

Remarks:

Johnson and Cook express the flow stress as

$$\sigma_y = \left(A + B\bar{\varepsilon}^{p^n}\right)\left(1 + c\ln\dot{\varepsilon}^*\right)\left(1 - T^{*m}\right)$$

where

A, B, C, n, and m = input constants

$\bar{\varepsilon}^p$ effective plastic strain

For VP=0, $\dot{\varepsilon}* = \dfrac{\dot{\bar{\varepsilon}}}{EPSO}$ effective total strain rate normalized by quasi-static threshold rate

For VP=1, $\dot{\varepsilon}* = \dfrac{\dot{\bar{\varepsilon}}^p}{EPSO}$ effective plastic strain rate normalized by quasi-static threshold rate

T* = homologous temperature = $\dfrac{T - T_{room}}{T_{melt} - T_{room}}$

The quantity $T - T_{room}$ is stored as extra history variable 5.

Constants for a variety of materials are provided in Johnson and Cook [1983]. A fully viscoplastic formulation is optional (VP) which incorporates the rate equations within the yield surface. An additional cost is incurred but the improvement is that results can be dramatic.

Due to nonlinearity in the dependence of flow stress on plastic strain, an accurate value of the flow stress requires iteration for the increment in plastic strain. However, by using a Taylor series expansion with linearization about the current time, we can solve for σ_y with sufficient accuracy to avoid iteration.

The strain at fracture is given by

$$\varepsilon^f = \left[D_1 + D_2 \exp D_3 \sigma^* \right] \left[1 + D_4 \ln \dot{\varepsilon}^* \right] \left[1 + D_5 T^* \right]$$

where σ* is the ratio of pressure divided by effective stress

$$\sigma^* = \frac{p}{\sigma_{eff}}$$

Fracture occurs when the damage parameter

$$D = \sum \frac{\Delta \bar{\varepsilon}^p}{\varepsilon^f}$$

reaches the value of 1. D is stored as extra history variable 4 in shell elements and extra history variable 6 in solid elements.

A choice of three spall models is offered to represent material splitting, cracking, and failure under tensile loads. The pressure limit model limits the minimum hydrostatic pressure to the specified value, $p \geq p_{min}$. If pressures more tensile than this limit are calculated, the pressure is reset to p_{min}. This option is not strictly a spall model since the deviatoric stresses are

unaffected by the pressure reaching the tensile cutoff and the pressure cutoff value p_{min} remains unchanged throughout the analysis. The maximum principal stress spall model detects spall if the maximum principal stress, σ_{max}, exceeds the limiting value σ_p. Once spall in solids is detected with this model, the deviatoric stresses are reset to zero and no hydrostatic tension is permitted. If tensile pressures are calculated, they are reset to 0 in the spalled material. Thus, the spalled material behaves as rubble. The hydrostatic tension spall model detects spall if the pressure becomes more tensile than the specified limit, p_{min}. Once spall in solids is detected with this model, the deviatoric stresses are set to zero and the pressure is required to be compressive. If hydrostatic tension is calculated then the pressure is reset to 0 for that element.

In addition to the above failure criterion, this material model also supports a shell element deletion criterion based on the maximum stable time step size for the element, Δt_{max}. Generally, Δt_{max} goes down as the element becomes more distorted. To assure stability of time integration, the global LS-DYNA time step is the minimum of the Δt_{max} values calculated for all elements in the model. Using this option allows the selective deletion of elements whose time step Δt_{max} has fallen below the specified minimum time step, Δt_{crit}. Elements which are severely distorted often indicate that material has failed and supports little load, but these same elements may have very small time steps and therefore control the cost of the analysis. This option allows these highly distorted elements to be deleted from the calculation, and, therefore, the analysis can proceed at a larger time step, and, thus, at a reduced cost. Deleted elements do not carry any load, and are deleted from all applicable slide surface definitions. Clearly, this option must be judiciously used to obtain accurate results at a minimum cost.

Material type 15 is applicable to the high rate deformation of many materials including most metals. Unlike the Steinberg-Guinan model, the Johnson-Cook model remains valid down to lower strain rates and even into the quasistatic regime. Typical applications include explosive metal forming, ballistic penetration, and impact.

Optional Strain Rate Forms

The standard Johnson-Cook strain rate term is linear in the logarithim of the strain rate:

$$1+C\ln\dot{\varepsilon}^*$$

Some additional data fiting capability can be obtained by using the quadratic form proposed by Huh & Kang [2002]:

$$1+C\ln\dot{\varepsilon}^*+C_2\left(\ln\dot{\varepsilon}^*\right)^2$$

Two additional exponential forms are available, one due to Allen, Rule & Jones [1997]

$$\left(\dot{\varepsilon}^*\right)^C$$

and the other a Cowper-Symonds-like [1958] form:

$$1+\left(\frac{\dot{\varepsilon}_{eff}^{p}}{C}\right)^{\frac{1}{P}}$$

The three additional rate forms (RATEOP=1,2, or 3) are currently available for solid & shell elements but only when the viscoplastic rate option is active (VP=1). See Huh and Kang [2002], Allen, Rule, and Jones [1997], and Cowper and Symonds [1958].

***MAT_PSEUDO_TENSOR**

This is Material Type 16. This model has been used to analyze buried steel reinforced concrete structures subjected to impulsive loadings.

Card 1

	1	2	3	4	5	6	7	8
Variable	MID	RO	G	PR				
Type	A8	F	F	F				
Default	none	none	none	none				

Card 2

Variable	SIGF	A0	A1	A2	A0F	A1F	B1	PER
Type	F	F	F	F	F	F	F	F
Default	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0

Card 3

Variable	ER	PRR	SIGY	ETAN	LCP	LCR		
Type	F	F	F	F	F	F		
Default	0.0	0.0	none	0.0				

Card 4

	1	2	3	4	5	6	7	8
Variable	X1	X2	X3	X4	X5	X6	X7	X8
Type	F	F	F	F	F	F	F	F
Default								

Card 5

Variable	X9	X10	X11	X12	X13	X14	X15	X16
Type	F	F	F	F	F	F	F	F
Default								

Card 6

Variable	YS1	YS2	YS3	YS4	YS5	YS6	YS7	YS8
Type	F	F	F	F	F	F	F	F
Default								

Card 7

Variable	YS9	YS10	YS11	YS12	YS13	YS14	YS15	YS16
Type	F	F	F	F	F	F	F	F
Default								

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
G	Shear modulus.
PR	Poisson's ratio.
SIGF	Tensile cutoff (maximum principal stress for failure).
A0	Cohesion.
A1	Pressure hardening coefficient.
A2	Pressure hardening coefficient.
A0F	Cohesion for failed material.
A1F	Pressure hardening coefficient for failed material.
B1	Damage scaling factor (or exponent in Mode II.C).
PER	Percent reinforcement.
ER	Elastic modulus for reinforcement.
PRR	Poisson's ratio for reinforcement.
SIGY	Initial yield stress.
ETAN	Tangent modulus/plastic hardening modulus.
LCP	Load curve ID giving rate sensitivity for principal material, see *DEFINE_CURVE.
LCR	Load curve ID giving rate sensitivity for reinforcement, see *DEFINE_CURVE.
X	Effective plastic strain, damage, or pressure. See discussion below.
YS	Yield stress (Mode I) or scale factor (Mode II.B or II.C).

Remarks:

This model can be used in two major modes - a simple tabular pressure-dependent yield surface, and a potentially complex model featuring two yield versus pressure functions with the means of migrating from one curve to the other. For both modes, load curve LCP is taken to be a

strain rate multiplier for the yield strength. Note that this model must be used with equation-of-state type 8 or 9.

Response Mode I. Tabulated Yield Stress Versus Pressure

This model is well suited for implementing standard geologic models like the Mohr-Coulomb yield surface with a Tresca limit, as shown in Figure 16.1. Examples of converting conventional triaxial compression data to this type of model are found in (Desai and Siriwardane, 1984). Note that under conventional triaxial compression conditions, the LS-DYNA input corresponds to an ordinate of $\sigma_1 - \sigma_3$ rather than the more widely used $\frac{\sigma_1 - \sigma_3}{2}$, where σ_1 is the maximum principal stress and σ_3 is the minimum principal stress.

This material combined with equation-of-state type 9 (saturated) has been used very successfully to model ground shocks and soil-structure interactions at pressures up to 100kbars (approximately 1.5 x 106 psi).

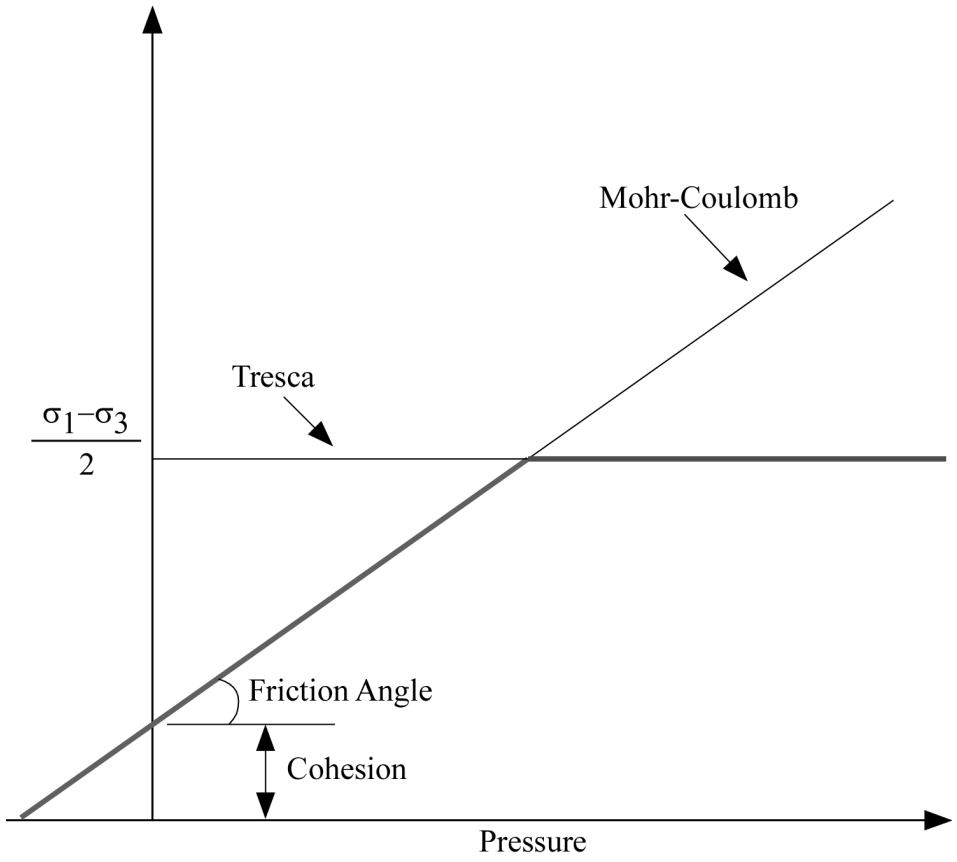


Figure 16.1. Mohr-Coulomb surface with a Tresca limit.

To invoke Mode I of this model, set a_0, a_1, a_2, b_1, a_{0f}, and a_{1f} to zero. The tabulated values of pressure should then be specified on cards 4 and 5, and the corresponding values of yield stress should be specified on cards 6 and 7. The parameters relating to reinforcement properties, initial yield stress, and tangent modulus are not used in this response mode, and should be set to zero.

Simple tensile failure

Note that a_{1f} is reset internally to 1/3 even though it is input as zero; this defines a failed material curve of slope 3*p*, where *p* denotes pressure (positive in compression). In this case the yield strength is taken from the tabulated yield vs. pressure curve until the maximum principal stress(σ_1) in the element exceeds the tensile cut-off (σ_{cut}). For every time step that $\sigma_1 > \sigma_{cut}$ the yield strength is scaled back by a fraction of the distance between the two curves until after 20 time steps the yield strength is defined by the failed curve. The only way to inhibit this feature is to set σ_{cut} arbitrarily large.

Response Mode II. Two Curve Model with Damage and Failure

This approach uses two yield versus pressure curves of the form

$$\sigma_y = a_0 + \frac{p}{a_1 + a_2 p}$$

The upper curve is best described as the maximum yield strength curve and the lower curve is the failed material curve. There are a variety of ways of moving between the two curves and each is discussed below.

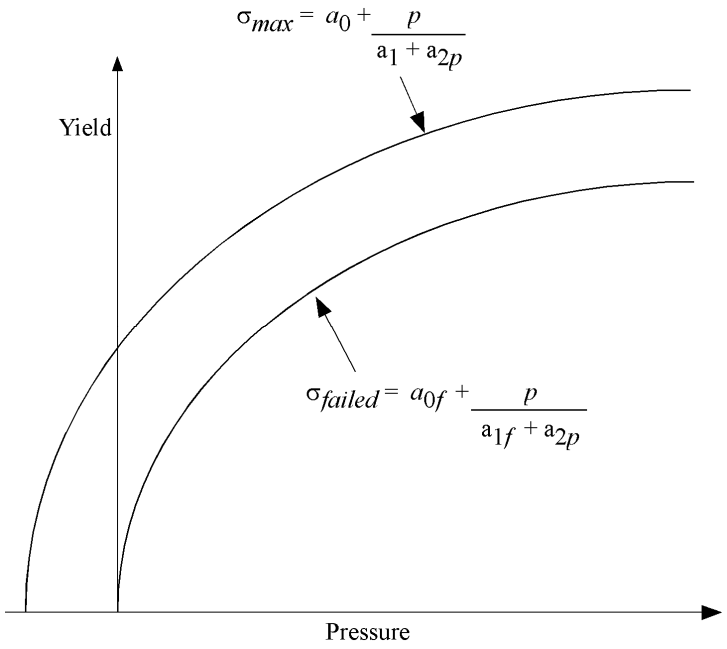


Figure 16.2. Two-curve concrete model with damage and failure.

MODE II. A: Simple tensile failure

Define a_0, a_1, a_2, a_{0f} and a_{1f}, set b_1 to zero, and leave cards 4 through 7 blank. In this case the yield strength is taken from the maximum yield curve until the maximum principal stress (σ_1) in the element exceeds the tensile cut-off (σ_{cut}). For every time step that $\sigma_1 > \sigma_{cut}$ the yield strength is scaled back by a fraction of the distance between the two curves until after 20 time steps the yield strength is defined by the failure curve.

Mode II.B: Tensile failure plus plastic strain scaling

Define a_0, a_1, a_2, a_{0f} and a_{1f}, set b_1 to zero, and user cards 4 through 7 to define a scale factor, η, versus effective plastic strain. LS-DYNA evaluates η at the current effective plastic strain and then calculated the yield stress as

$$\sigma_{yield} = \sigma_{failed} + \eta(\sigma_{\max} - \sigma_{failed})$$

where $\sigma_{\max}$ and σ_{failed} are found as shown in Figure 16.2. This yield strength is then subject to scaling for tensile failure as described above. This type of model allows the description of a strain hardening or softening material such as concrete.

Mode II.C: Tensile failure plus damage scaling

The change in yield stress as a function of plastic strain arises from the physical mechanisms such as internal cracking, and the extent of this cracking is affected by the hydrostatic pressure when the cracking occurs. This mechanism gives rise to the "confinement" effect on concrete behavior. To account for this phenomenon, a "damage" function was defined and incorporated. This damage function is given the form:

$$\lambda = \int_0^{\varepsilon^p} \left(1 + \frac{p}{\sigma_{cut}}\right)^{-b_1} d\varepsilon^p$$

Define a_0, a_1, a_2, a_{0f} and a_{1f}, and b_1. Cards 4 though 7 now give η as a function of λ and scale the yield stress as

$$\sigma_{yield} = \sigma_{failed} + \eta\left(\sigma_{\max} - \sigma_{failed}\right)$$

and then apply any tensile failure criteria.

Mode II Concrete Model Options

Material Type 16 Mode II provides for the automatic internal generation of a simple "generic" model from concrete if A0 is negative then SIGF is assumed to be the unconfined concrete compressive strength, $f_c^{'}$ and –A0 is assumed to be a conversion factor from LS-DYNA pressure units to psi. (For example, if the model stress units are MPa, A0 should be set to –145.) In this case the parameter values generated internally are

$$f_c^{'} = SIGF$$

$$\sigma_{cut} = 1.7\left(\frac{f_c^{'2}}{-A0}\right)^{\frac{1}{3}}$$

$$a_0 = \frac{f_c^{'}}{4}$$

$$a_1 = \frac{1}{3}$$

$$a_2 = \frac{1}{3f_c^{'}}$$

$$a_{0f} = 0$$

$$a_{1f} = 0.385$$

Note that these a_{0f} and a_{1f} defaults will be overridden by non zero entries on Card 3. If plastic strain or damage scaling is desired, Cards 5 through 8 and $b1$ should be specified in the input. When a_0 is input as a negative quantity, the equation-of-state can be given as 0 and a trilinear EOS Type 8 model will be automatically generated from the unconfined compressive strength and Poisson's ratio. The EOS 8 model is a simple pressure versus volumetric strain model with no internal energy terms, and should give reasonable results for pressures up to 5kbar (approximately 75,000 psi).

Mixture model

A reinforcement fraction, f_r, can be defined along with properties of the reinforcement material. The bulk modulus, shear modulus, and yield strength are then calculated from a simple mixture rule, i.e., for the bulk modulus the rule gives:

$$K = (1 - f_r)K_m + f_r K_r$$

where K_m and K_r are the bulk moduli for the geologic material and the reinforcement material, respectively. This feature should be used with caution. It gives an isotropic effect in the material instead of the true anisotropic material behavior. A reasonable approach would be to use the mixture elements only where the reinforcing exists and plain elements elsewhere. When the mixture model is being used, the strain rate multiplier for the principal material is taken from load curve N1 and the multiplier for the reinforcement is taken from load curve N2.

A Suggestion

The LLNL DYNA3D manual from 1991 [Whirley and Hallquist] suggests using the damage function (Mode II.C.) in Material Type 16 with the following set of parameters:

$$a_0 = \frac{f_c^{'}}{4}$$
$$a_1 = \frac{1}{3}$$
$$a_2 = \frac{1}{3f_c^{'}}$$
$$a_{0f} = \frac{f_c^{'}}{10}$$
$$a_{1f} = 1.5$$
$$b_1 = 1.25$$

and a damage table of:

Card 4:	0.0	8.62E-06	2.15E-05	3.14E-05	3.95E-04
	5.17E-04	6.38E-04	7.98E-04		
Card 5:	9.67E-04	1.41E-03	1.97E-03	2.59E-03	3.27E-03
	4.00E-03	4.79E-03	0.909		
Card 6:	0.309	0.543	0.840	0.975	1.000
	0.790	0.630	0.469		
Card 7:	0.383	0.247	0.173	0.136	0.114
	0.086	0.056	0.0		

This set of parameters should give results consistent with Dilger, Koch, and Kowalczyk, [1984] for plane concrete. It has been successfully used for reinforced structures where the reinforcing bars were modeled explicitly with embedded beam and shell elements. The model does not incorporate the major failure mechanism - separation of the concrete and reinforcement leading to catastrophic loss of confinement pressure. However, experience indicates that this physical behavior will occur when this model shows about 4% strain.

*MAT_ORIENTED_CRACK

This is Material Type 17. This material may be used to model brittle materials which fail due to large tensile stresses.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	SIGY	ETAN	FS	PRF
Type	A8	F	F	F	F	F	F	F
Default	none	None	none	none	none	0.0	none	0.0

Optional card for crack propagation to adjacent elements (see remarks):

Card 2	1	2	3	4	5	6	7	8
Variable	SOFT	CVELO						
Type	F	F						
Default	1.0	0.0						

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.
SIGY	Yield stress.
ETAN	Plastic hardening modulus.
FS	Fracture stress.
PRF	Failure or cutoff pressure (≤ 0.0).

VARIABLE	DESCRIPTION
SOFT	Factor by which the fracture stress is reduced when a crack is coming from failed neighboring element. See remarks.
CVELO	Crack propagation velocity. See remarks.

Remarks:

This is an isotropic elastic-plastic material which includes a failure model with an oriented crack. The von Mises yield condition is given by:

$$\phi = J_2 - \frac{\sigma_y^2}{3}$$

where the second stress invariant, J_2, is defined in terms of the deviatoric stress components as

$$J_2 = \frac{1}{2} s_{ij} s_{ij}$$

and the yield stress, σ_y, is a function of the effective plastic strain, ε_{eff}^p, and the plastic hardening modulus, E_p:

$$\sigma_y = \sigma_0 + E_p \varepsilon_{eff}^p$$

The effective plastic strain is defined as:

$$\varepsilon_{eff}^p = \int_0^t d\varepsilon_{eff}^p$$

where $$d\varepsilon_{eff}^p = \sqrt{\frac{2}{3} d\varepsilon_{ij}^p d\varepsilon_{ij}^p}$$

and the plastic tangent modulus is defined in terms of the input tangent modulus, E_t, as

$$E_p = \frac{E E_t}{E - E_t}$$

Pressure in this model is found from evaluating an equation of state. A pressure cutoff can be defined such that the pressure is not allowed to fall below the cutoff value.

The oriented crack fracture model is based on a maximum principal stress criterion. When the maximum principal stress exceeds the fracture stress, σ_f, the element fails on a plane perpendicular to the direction of the maximum principal stress. The normal stress and the two shear stresses on that plane are then reduced to zero. This stress reduction is done according to a delay function that reduces the stresses gradually to zero over a small number of time steps. This

delay function procedure is used to reduce the ringing that may otherwise be introduced into the system by the sudden fracture. The number of steps for stress reduction is 20 by default (CVELO=0.0) or it is internally computed if CVELO > 0.0 is given:

$$n_{steps} = \text{int}\left(\frac{L_e}{CVELO \cdot \Delta t}\right)$$

where L_e is characteristic element length and Δt is time step size.

After a tensile fracture, the element will not support tensile stress on the fracture plane, but in compression will support both normal and shear stresses. The orientation of this fracture surface is tracked throughout the deformation, and is updated to properly model finite deformation effects. If the maximum principal stress subsequently exceeds the fracture stress in another direction, the element fails isotropically. In this case the element completely loses its ability to support any shear stress or hydrostatic tension, and only compressive hydrostatic stress states are possible. Thus, once isotropic failure has occurred, the material behaves like a fluid.

This model is applicable to elastic or elastoplastic materials under significant tensile or shear loading when fracture is expected. Potential applications include brittle materials such as ceramics as well as porous materials such as concrete in cases where pressure hardening effects are not significant.

Crack propagation behavior to adjacent elements can be controlled via parameter SOFT for thin, shell-like structures (e.g. only 2 or 3 solids over thickness). Additionally, LS-DYNA has to know where the plane or solid element midplane is at each integration point for projection of crack plane on this element midplane. Therefore, element numbering has to be as shown in Figure 17.1. Only solid element type 1 is supported with that option at the moment.

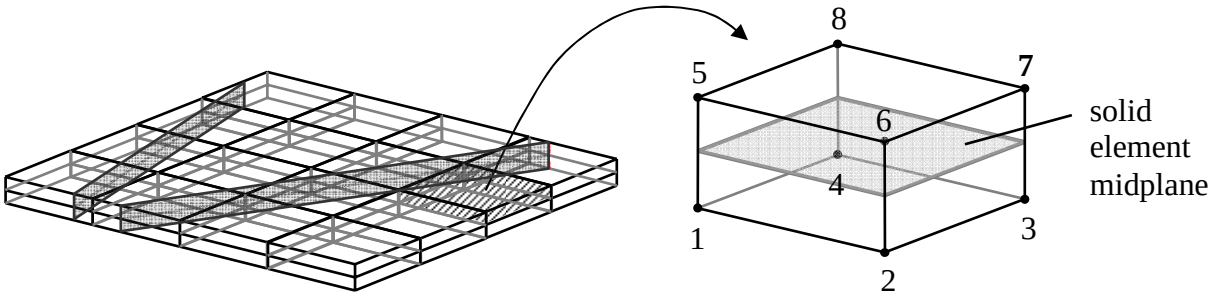


Figure 17.1. Thin structure (2 elements over thickness) with cracks and necessary element numbering

*MAT_POWER_LAW_PLASTICITY

This is Material Type 18. This is an isotropic plasticity model with rate effects which uses a power law hardening rule.

Card 1

Variable	MID	RO	E	PR	K	N	SRC	SRP
Type	A8	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	0.0	0.0

Card 2

Variable	SIGY	VP	EPSF					
Type	F	F	F					
Default	0.0	0.0	0.0					

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.
K	Strength coefficient.
N	Hardening exponent.
SRC	Strain rate parameter, C, if zero, rate effects are ignored.
SRP	Strain rate parameter, P, if zero, rate effects are ignored.

VARIABLE	DESCRIPTION
SIGY	Optional input parameter for defining the initial yield stress, σ_y. Generally, this parameter is not necessary and the strain to yield is calculated as described below. LT.0.02: $\varepsilon_{yp} = SIGY$ GE.0.02: See below.
EPSF	Plastic failure strain for element deletion.
VP	Formulation for rate effects: EQ.0.0: Scale yield stress (default), EQ.1.0: Viscoplastic formulation.

Remarks:

Elastoplastic behavior with isotropic hardening is provided by this model. The yield stress, σ_y, is a function of plastic strain and obeys the equation:

$$\sigma_y = k\ \varepsilon^n = k\left(\varepsilon_{yp} + \bar{\varepsilon}^p\right)^n$$

where ε_{yp} is the elastic strain to yield and $\bar{\varepsilon}^p$ is the effective plastic strain (logarithmic). If SIGY is set to zero, the strain to yield if found by solving for the intersection of the linearly elastic loading equation with the strain hardening equation:

$$\sigma = E\varepsilon$$
$$\sigma = k\ \varepsilon^n$$

which gives the elastic strain at yield as:

$$\varepsilon_{yp} = \left(\frac{E}{k}\right)^{\left[\frac{1}{n-1}\right]}$$

If SIGY yield is nonzero and greater than 0.02 then:

$$\varepsilon_{yp} = \left(\frac{\sigma_y}{k}\right)^{\left[\frac{1}{n}\right]}$$

Strain rate is accounted for using the Cowper and Symonds model which scales the yield stress with the factor

$$1 + \left(\frac{\dot{\varepsilon}}{C}\right)^{1/p}$$

where $\dot{\varepsilon}$ is the strain rate. A fully viscoplastic formulation is optional which incorporates the Cowper and Symonds formulation within the yield surface. An additional cost is incurred but the improvement is results can be dramatic.

*MAT_STRAIN_RATE_DEPENDENT_PLASTICITY

This is Material Type 19. A strain rate dependent material can be defined. For an alternative, see Material Type 24. Required is a curve for the yield stress versus the effective strain rate. Optionally, Young's modulus and the tangent modulus can also be defined versus the effective strain rate. Also, optional failure of the material can be defined either by defining a von Mises stress at failure as a function of the effective strain rate (valid for solids/shells/thick shells) or by defining a minimum time step size (only for shells).

Card 1

Variable	MID	RO	E	PR	VP			
Type	A8	F	F	F	F			
Default	none	none	none	none	0.0			

Card 2

Variable	LC1	ETAN	LC2	LC3	LC4	TDEL	RDEF	
Type	F	F	F	F	F	F	F	
Default	none	0.0	0.0	0.0	0.0	0.0	0.0	

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.
VP	Formulation for rate effects: EQ.0.0: Scale yield stress (default), EQ.1.0: Viscoplastic formulation (recommended).

VARIABLE	DESCRIPTION
LC1	Load curve ID defining the yield stress σ_0 as a function of the effective strain rate.
ETAN	Tangent modulus, E_t
LC2	Load curve ID defining Young's modulus as a function of the effective strain rate (available only when VP=0; not recommended).
LC3	Load curve ID defining tangent modulus as a function of the effective strain rate (optional).
LC4	Load curve ID defining von Mises stress at failure as a function of the effective strain rate (optional).
TDEL	Minimum time step size for automatic element deletion. Use for shells only.
RDEF	Redefinition of failure curve: EQ.1.0: Effective plastic strain, EQ.2.0: Maximum principal stress and absolute value of minimum principal stress, EQ.3.0: Maximum principal stress (release 5 of v.971)

Remarks:

In this model, a load curve is used to describe the yield strength σ_0 as a function of effective strain rate $\dot{\bar{\varepsilon}}$ where

$$\dot{\bar{\varepsilon}} = \left(\frac{2}{3} \dot{\varepsilon}'_{ij} \, \dot{\varepsilon}'_{ij} \right)^{1/2}$$

and the prime denotes the deviatoric component. The yield stress is defined as

$$\sigma_y = \sigma_0 \left(\dot{\bar{\varepsilon}} \right) + E_p \bar{\varepsilon}^p$$

where $\bar{\varepsilon}^p$ is the effective plastic strain and E_p is given in terms of Young's modulus and the tangent modulus by

$$E_p = \frac{E E_t}{E - E_t}.$$

Both Young's modulus and the tangent modulus may optionally be made functions of strain rate by specifying a load curve ID giving their values as a function of strain rate. If these load curve ID's are input as 0, then the constant values specified in the input are used.

Note that all load curves used to define quantities as a function of strain rate must have the same number of points at the same strain rate values. This requirement is used to allow vectorized interpolation to enhance the execution speed of this constitutive model.

This model also contains a simple mechanism for modeling material failure. This option is activated by specifying a load curve ID defining the effective stress at failure as a function of strain rate. For solid elements, once the effective stress exceeds the failure stress the element is deemed to have failed and is removed from the solution. For shell elements the entire shell element is deemed to have failed if all integration points through the thickness have an effective stress that exceeds the failure stress. After failure the shell element is removed from the solution.

In addition to the above failure criterion, this material model also supports a shell element deletion criterion based on the maximum stable time step size for the element, $\Delta t_{\max}$. Generally, $\Delta t_{\max}$ goes down as the element becomes more distorted. To assure stability of time integration, the global LS-DYNA time step is the minimum of the $\Delta t_{\max}$ values calculated for all elements in the model. Using this option allows the selective deletion of elements whose time step $\Delta t_{\max}$ has fallen below the specified minimum time step, Δt_{crit}. Elements which are severely distorted often indicate that material has failed and supports little load, but these same elements may have very small time steps and therefore control the cost of the analysis. This option allows these highly distorted elements to be deleted from the calculation, and, therefore, the analysis can proceed at a larger time step, and, thus, at a reduced cost. Deleted elements do not carry any load, and are deleted from all applicable slide surface definitions. Clearly, this option must be judiciously used to obtain accurate results at a minimum cost.

A fully viscoplastic formulation is optional which incorporates the rate formulation within the yield surface. An additional cost is incurred but the improvement is results can be dramatic.

*MAT_RIGID

This is Material 20. Parts made from this material are considered to belong to a rigid body (for each part ID). Also, the coupling of a rigid body with MADYMO and CAL3D can be defined via this material. Alternatively, a VDA surface can be attached as surface to model the geometry, e.g., for the tooling in metalforming applications. Also, global and local constraints on the mass center can be optionally defined. Optionally, a local consideration for output and user-defined airbag sensors can be chosen.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	N	COUPLE	M	ALIAS RE
Type	A8	F	F	F	F	F	F	C/F
Default	none	none	none	none	0	0	0	Blank none

Card 2

Variable	CMO	CON1	CON2					
Type	F	F	F					
Default	0	0	0					

Optional for output (Must be included but may be left blank).

Card 3	1	2	3	4	5	6	7	8
Variable	LCO or A1	A2	A3	V1	V2	V3		
Type	F	F	F	F	F	F		
Default	0	0	0	0	0	0		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
E	Young's modulus. Reasonable values have to be chosen for contact analysis (choice of penalty), see Remarks below.
PR	Poisson's ratio. Reasonable values have to be chosen for contact analysis (choice of penalty), see Remarks below.
N	MADYMO3D 5.4 coupling flag, n: EQ.0: use normal LS-DYNA rigid body updates, GT.0: the rigid body is coupled to MADYMO 5.4 ellipsoid number n, LT.0: the rigid body is coupled to MADYMO 5.4 plane number |n|.
COUPLE	Coupling option if applicable: EQ.-1: attach VDA surface in ALIAS (defined in the eighth field) and automatically generate a mesh for viewing the surface in LS-PREPOST. MADYMO 5.4 / CAL3D coupling option: EQ.0: the undeformed geometry input to LS-DYNA corresponds to the local system for MADYMO 5.4 / CAL3D. The finite element mesh is input, EQ.1: the undeformed geometry input to LS-DYNA corresponds to the global system for MADYMO 5.4 / CAL3D, EQ.2: generate a mesh for the ellipsoids and planes internally in LS-DYNA.
M	MADYMO3D 5.4 coupling flag, m: EQ.0: use normal LS-DYNA rigid body updates, EQ.m: this rigid body corresponds to MADYMO rigid body number m. Rigid body updates are performed by MADYMO.
ALIAS	VDA surface alias name, see Appendix L.
RE	MADYMO 6.0.1 External Reference Number
CMO	Center of mass constraint option, CMO: EQ.+1.0: constraints applied in global directions, EQ. 0.0: no constraints, EQ. -1.0: constraints applied in local directions (SPC constraint).
CON1	First constraint parameter:

VARIABLE	DESCRIPTION
	If CMO=+1.0, then specify global translational constraint: EQ.0: no constraints, EQ.1: constrained x displacement, EQ.2: constrained y displacement, EQ.3: constrained z displacement, EQ.4: constrained x and y displacements, EQ.5: constrained y and z displacements, EQ.6: constrained z and x displacements, EQ.7: constrained x, y, and z displacements. If CM0=-1.0, then specify local coordinate system ID. See *DEFINE_ COORDINATE_*OPTION*: This coordinate system is fixed in time.
CON2	Second constraint parameter: If CMO=+1.0, then specify global rotational constraint: EQ.0: no constraints, EQ.1: constrained x rotation, EQ.2: constrained y rotation, EQ.3: constrained z rotation, EQ.4: constrained x and y rotations, EQ.5: constrained y and z rotations, EQ.6: constrained z and x rotations, EQ.7: constrained x, y, and z rotations. If CM0=-1.0, then specify local (SPC) constraint: EQ.000000 no constraint, EQ.100000 constrained x translation, EQ.010000 constrained y translation, EQ.001000 constrained z translation, EQ.000100 constrained x rotation, EQ.000010 constrained y rotation, EQ.000001 constrained z rotation. Any combination of local constraints can be achieved by adding the number 1 into the corresponding column.
LCO	Local coordinate system number for output. (See *DEFINE_ COORDINATE_*OPTION*.) *****Alternative method for specifying local system below.*****
A1-V3	Define two vectors **a** and **v**, fixed in the rigid body which are used for output and the user defined airbag sensor subroutines. The output parameters are in the directions **a**, **b**, and **c** where the latter are given by the cross products **c=a**×**v** and **b=c**×**a.** This input is optional.

Remarks:

The rigid material type 20 provides a convenient way of turning one or more parts comprised of beams, shells, or solid elements into a rigid body. Approximating a deformable body as rigid is a preferred modeling technique in many real world applications. For example, in sheet metal forming problems the tooling can properly and accurately be treated as rigid. In the design of restraint systems the occupant can, for the purposes of early design studies, also be treated as rigid. Elements which are rigid are bypassed in the element processing and no storage is allocated for storing history variables; consequently, the rigid material type is very cost efficient.

Two unique rigid part ID's may not share common nodes unless they are merged together using the rigid body merge option. A rigid body may be made up of disjoint finite element meshes, however. LS-DYNA assumes this is the case since this is a common practice in setting up tooling meshes in forming problems.

All elements which reference a given part ID corresponding to the rigid material should be contiguous, but this is not a requirement. If two disjoint groups of elements on opposite sides of a model are modeled as rigid, separate part ID's should be created for each of the contiguous element groups if each group is to move independently. This requirement arises from the fact that LS-DYNA internally computes the six rigid body degrees-of-freedom for each rigid body (rigid material or set of merged materials), and if disjoint groups of rigid elements use the same part ID, the disjoint groups will move together as one rigid body.

Inertial properties for rigid materials may be defined in either of two ways. By default, the inertial properties are calculated from the geometry of the constituent elements of the rigid material and the density specified for the part ID. Alternatively, the inertial properties and initial velocities for a rigid body may be directly defined, and this overrides data calculated from the material property definition and nodal initial velocity definitions.

Young's modulus, E, and Poisson's ratio, υ are used for determining sliding interface parameters if the rigid body interacts in a contact definition. Realistic values for these constants should be defined since unrealistic values may contribute to numerical problem in contact.

Constraint directions for rigid materials (CMO equal to +1 or -1) are fixed, that is, not updated, with time. To impose a constraint on a rigid body such that the constraint direction is updated as the rigid body rotates, use *BOUNDARY_PRESCRIBED_MOTION_RIGID_LOCAL.

If no constraints are specified for the rigid part (CMO=0) the nodes for the part are scanned to determine constraints on the part in global directions. If constraints are specified (CMO equal to +1 or -1) then the nodes are not scanned.

For coupling with MADYMO 5.4.1, only basic coupling is available.

The coupling flags (N and M) must match with SYSTEM and ELLIPSOID/PLANE in the MADYMO input file and the coupling option (COUPLE) must be defined.

For coupling with MADYMO 6.0.1, both basic and extended coupling are available:

(1) Basic Coupling: The external reference number (RE) must match with the external reference number in the MADYMO XML input file. The coupling option (COUPLE) must be defined.

(2) Extended Coupling: Under this option MADYMO will handle the contact between the MADYMO and LS-DYNA models. The external reference number (RE) and the coupling option (COUPLE) are not needed. All coupling surfaces that interface with the MADYMO models need to be defined in *CONTACT_COUPLING.

*MAT_ORTHOTROPIC_THERMAL

This is Material Type 21. A linearly elastic, orthotropic material with orthotropic thermal expansion.

Card 1

1	2	3	4	5	6	7	8
Variable	MID	RO	EA	EB	EC	PRBA	PRCA	PRCB
Type	A8	F	F	F	F	F	F	F

Card 2

Variable	GAB	GBC	GCA	AA	AB	AC	AOPT	MACF
Type	F	F	F	F	F	F	F	I

Card 3

Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card 4

Variable	V1	V2	V3	D1	D2	D3	BETA	REF
Type	F	F	F	F	F	F	F	

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
EA	E_a, Young's modulus in a-direction.

VARIABLE	DESCRIPTION
EB	E_b, Young's modulus in b-direction.
EC	E_c, Young's modulus in c-direction.
PRBA	ν_{ba}, Poisson's ratio, ba.
PRCA	ν_{ca}, Poisson's ratio, ca.
PRCB	ν_{cb}, Poisson's ratio, cb
GAB	G_{ab}, Shear modulus, ab.
GBC	G_{bc}, Shear modulus, bc.
GCA	G_{ca}, Shear modulus, ca.
AA	α_a, coefficients of thermal expansion in the a-direction.
AB	α_b, coefficients of thermal expansion in the b-direction.
AC	α_c, coefficients of thermal expansion in the c-direction.
AOPT	Material axes option (see MAT_*OPTION* TROPIC_ELASTIC for a more complete description): EQ. 0.0: locally orthotropic with material axes determined by element nodes 1, 2, and 4, as with *DEFINE_COORDINATE_NODES. EQ. 1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center; this is the a-direction. This option is for solid elements only. EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR. EQ. 3.0: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle, BETA, from a line in the plane of the element defined by the cross product of the vector **v** with the element normal. EQ. 4.0: locally orthotropic in cylindrical coordinate system with the material axes determined by a vector **v**, and an originating point, P, which define the centerline axis. This option is for solid elements only. LT.0.0: the absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_NODES, *DEFINE_COORDINATE_SYSTEM or *DEFINE_COORDINATE_VECTOR). Available in R3 version of 971 and later.

VARIABLE	DESCRIPTION
MACF	Material axes change flag for brick elements: EQ.1: No change, default, EQ.2: switch material axes a and b, EQ.3: switch material axes a and c, EQ.4: switch material axes b and c.
XP,YP,ZP	Coordinates of point **p** for AOPT = 1.
A1,A2,A3	Components of vector **a** for AOPT = 2.
V1,V2,V3	Components of vector **v** for AOPT = 3.
D1,D2,D3	Components of vector **d** for AOPT = 2.
BETA	Material angle in degrees for AOPT = 3, may be overridden on the element card, see *ELEMENT_SHELL_BETA or *ELEMENT_SOLID_ORTHO.
REF	Use reference geometry to initialize the stress tensor. The reference geometry is defined by the keyword:*INITIAL_FOAM_REFERENCE_GEOMETRY (see there for more details). EQ.0.0: off, EQ.1.0: on.

Remarks:

In the implementation for three-dimensional continua a total Lagrangian formulation is used. In this approach the material law that relates second Piola-Kirchhoff stress S to the Green-St. Venant strain E is

$$S = C \cdot E = T^t C_l T \cdot E$$

where T is the transformation matrix [Cook 1974].

$$T = \begin{bmatrix} l_1^2 & m_1^2 & n_1^2 & l_1 m_1 & m_1 n_1 & n_1 l_1 \\ l_2^2 & m_2^2 & n_2^2 & l_2 m_2 & m_2 n_2 & n_2 l_2 \\ l_3^2 & m_3^2 & n_3^2 & l_3 m_3 & m_3 n_3 & n_3 l_3 \\ 2l_1 l_2 & 2m_1 m_2 & 2n_1 n_2 & (l_1 m_2 + l_2 m_1) & (m_1 n_2 + m_2 n_1) & (n_1 l_2 + n_2 l_1) \\ 2l_2 l_3 & 2m_2 m_3 & 2n_2 n_3 & (l_2 m_3 + l_3 m_2) & (m_2 n_3 + m_3 n_2) & (n_2 l_3 + n_3 l_2) \\ 2l_3 l_1 & 2m_3 m_1 & 2n_3 n_1 & (l_3 m_1 + l_1 m_3) & (m_3 n_1 + m_1 n_3) & (n_3 l_1 + n_1 l_3) \end{bmatrix}$$

l_i, m_i, n_i are the direction cosines

$$x_i' = l_i x_1 + m_i x_2 + n_i x_3 \qquad for\ i = 1,2,3$$

and x_i' denotes the material axes. The constitutive matrix C_l is defined in terms of the material axes as

$$C_l^{-1} = \begin{bmatrix} \frac{1}{E_{11}} & -\frac{\upsilon_{21}}{E_{22}} & -\frac{\upsilon_{31}}{E_{33}} & 0 & 0 & 0 \\ -\frac{\upsilon_{12}}{E_{11}} & \frac{1}{E_{22}} & -\frac{\upsilon_{32}}{E_{33}} & 0 & 0 & 0 \\ -\frac{\upsilon_{13}}{E_{11}} & -\frac{\upsilon_{23}}{E_{22}} & \frac{1}{E_{33}} & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1}{G_{12}} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1}{G_{23}} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1}{G_{31}} \end{bmatrix}$$

where the subscripts denote the material axes, i.e.,

$$\upsilon_{ij} = \upsilon_{x_i' x_j'} \quad \text{and} \quad E_{ii} = E_{x_i'}$$

Since C_l is symmetric

$$\frac{\upsilon_{12}}{E_{11}} = \frac{\upsilon_{21}}{E_{22}}, \text{ etc.}$$

The vector of Green-St. Venant strain components is

$$E^t = \lfloor E_{11}, E_{22}, E_{33}, E_{12}, E_{23}, E_{31} \rfloor$$

which include the local thermal strains which are integrated in time:

$$\varepsilon_{aa}^{n+1} = \varepsilon_{aa}^n + \alpha_a \left(T^{n+1} - T^n \right)$$
$$\varepsilon_{bb}^{n+1} = \varepsilon_{bb}^n + \alpha_b \left(T^{n+1} - T^n \right)$$
$$\varepsilon_{cc}^{n+1} = \varepsilon_{cc}^n + \alpha_c \left(T^{n+1} - T^n \right)$$

After computing S_{ij} we then obtain the Cauchy stress:

$$\sigma_{ij} = \frac{\rho}{\rho_0} \frac{\partial x_i}{\partial X_k} \frac{\partial x_j}{\partial X_l} S_{kl}$$

This model will predict realistic behavior for finite displacement and rotations as long as the strains are small.

In the implementation for shell elements, the stresses are integrated in time and are updated in the corotational coordinate system. In this procedure the local material axes are assumed to remain orthogonal in the deformed configuration. This assumption is valid if the strains remain small.

*MAT_COMPOSITE_DAMAGE

This is Material Type 22. An orthotropic material with optional brittle failure for composites can be defined following the suggestion of [Chang and Chang 1987a, 1987b]. Three failure criteria are possible, see the LS-DYNA Theory Manual. By using the user defined integration rule, see *INTEGRATION_SHELL, the constitutive constants can vary through the shell thickness. For all shells, except the DKT formulation, laminated shell theory can be activated to properly model the transverse shear deformation. Lamination theory is applied to correct for the assumption of a uniform constant shear strain through the thickness of the shell. For sandwich shells where the outer layers are much stiffer than the inner layers, the response will tend to be too stiff unless lamination theory is used. To turn on lamination theory see *CONTROL_SHELL.

Card 1

1	2	3	4	5	6	7	8
Variable	MID	RO	EA	EB	EC	PRBA	PRCA	PRCB
Type	A8	F	F	F	F	F	F	F
Default	none	None	none	none	none	none		

Card 2

Variable	GAB	GBC	GCA	KFAIL	AOPT	MACF		
Type	F	F	F	F	F	I		
Default	none	None	none	0.0	0.0	0		

Card 3

Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		
Default	0.0	0.0	0.0	0.0	0.0	0.0		

Card 4

	1	2	3	4	5	6	7	8
Variable	V1	V2	V3	D1	D2	D3	BETA	
Type	F	F	F	F	F	F	F	
Default	0.0	0.0	0.0	0.0	0.0	0.0	0.0	

Card 5

Variable	SC	XT	YT	YC	ALPH	SN	SYZ	SZX
Type	F	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	none	none

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
EA	E_a, Young's modulus in a-direction.
EB	E_b, Young's modulus in b-direction.
EC	E_c, Young's modulus in c-direction.
PRBA	ν_{ba}, Poisson ratio, ba.
PRCA	ν_{ca}, Poisson ratio, ca (if zero, defaults to PRBA for shell thickness updates).
PRCB	ν_{cb}, Poisson ratio, cb (if zero, defaults to PRBA for shell thickness updates).
GAB	G_{ab}, Shear modulus, ab.
GBC	G_{bc}, Shear modulus, bc.
GCA	G_{ca}, Shear modulus, ca.

VARIABLE	DESCRIPTION
KFAIL	Bulk modulus of failed material. Necessary for compressive failure.
AOPT	Material axes option (see MAT_*OPTION* TROPIC_ELASTIC for a more complete description): EQ. 0.0: locally orthotropic with material axes determined by element nodes 1, 2, and 4, as with *DEFINE_COORDINATE_NODES. EQ. 1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center; this is the a-direction. This option is for solid elements only. EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR. EQ. 3.0: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle, BETA, from a line in the plane of the element defined by the cross product of the vector **v** with the element normal. EQ. 4.0: locally orthotropic in cylindrical coordinate system with the material axes determined by a vector **v**, and an originating point, P, which define the centerline axis. This option is for solid elements only. LT.0.0: the absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_NODES, *DEFINE_COORDINATE_SYSTEM or *DEFINE_COORDINATE_VECTOR). Available in R3 version of 971 and later.
MACF	Material axes change flag for brick elements: EQ.1: No change, default, EQ.2: switch material axes a and b, EQ.3: switch material axes a and c, EQ.4: switch material axes b and c.
XP,YP,ZP	Coordinates of point **p** for AOPT = 1.
A1,A2,A3	Components of vector **a** for AOPT = 2.
V1,V2,V3	Components of vector **v** for AOPT = 3.
D1,D2,D3	Components of vector **d** for AOPT = 2.
BETA	Material angle in degrees for AOPT = 3, may be overridden on the element card, see *ELEMENT_SHELL_BETA or *ELEMENT_SOLID_ORTHO.
SC	Shear strength, ab plane, see the LS-DYNA Theory Manual.

VARIABLE	DESCRIPTION
XT	Longitudinal tensile strength, a-axis, see the LS-DYNA Theory Manual.
YT	Transverse tensile strength, b-axis.
YC	Transverse compressive strength, b-axis (positive value).
ALPH	Shear stress parameter for the nonlinear term, see the LS-DYNA Theory Manual. Suggested range 0 – 0.5.
SN	Normal tensile strength *(solid elements only)*
SYZ	Transverse shear strength *(solid elements only)*
SZX	Transverse shear strength *(solid elements only)*

Remarks:

The number of additional integration point variables for shells written to the d3plot database is input by the optional *DATABASE_EXTENT_BINARY as variable NEIPS. These additional variables are tabulated below (*ip* = shell integration point):

History Variable	Description	Value	LS-Prepost history variable
ef(i)	*tensile fiber mode*	*1 - elastic* *0 - failed*	*1*
cm(i)	*tensile matrix mode*		*2*
ed(i)	*compressive matrix mode*		*3*

These variables can be plotted in LS-Prepost as element history variables 1, 2, and 3. The following components are stored as element component 7 instead of the effective plastic strain.

Description	Integration point
$\frac{1}{nip}\sum_{i=1}^{nip} ef(i)$	*1*
$\frac{1}{nip}\sum_{i=1}^{nip} cm(i)$	*2*
$\frac{1}{nip}\sum_{i=1}^{nip} ed(i)$	*3*

*MAT_TEMPERATURE_DEPENDENT_ORTHOTROPIC

This is Material Type 23. An orthotropic elastic material with arbitrary temperature dependency can be defined.

Card 1

1	2	3	4	5	6	7	8
Variable	MID	RO	AOPT	REF	MACF			
Type	A8	F	F	F	I			

Card 2

Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card 3

Variable	V1	V2	V3	D1	D2	D3	BETA	
Type	F	F	F	F	F	F	F	

Define one set of constants on two cards for each temperature point. Up to 48 points (96 cards) can defined. The next "*" card terminates the input.

Card 1 for Temperature Ti

1	2	3	4	5	6	7	8
Variable	EAi	EBi	ECi	PRBAi	PRCAi	PRCBi		
Type	F	F	F	F	F	F		

Card 2 for Temperature Ti	1	2	3	4	5	6	7	8
Variable	AAi	ABi	ACi	GABi	GBCi	GCAi	Ti	
Type	F	F	F	F	F	F	F	

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
AOPT	Material axes option (see MAT_*OPTION* TROPIC_ELASTIC for a more complete description): EQ. 0.0: locally orthotropic with material axes determined by element nodes 1, 2, and 4, as with *DEFINE_COORDINATE_NODES. EQ. 1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center; this is the a-direction. This option is for solid elements only. EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR. EQ. 3.0: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle, BETA, from a line in the plane of the element defined by the cross product of the vector **v** with the element normal. EQ. 4.0: locally orthotropic in cylindrical coordinate system with the material axes determined by a vector **v**, and an originating point, P, which define the centerline axis. This option is for solid elements only. LT.0.0: the absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_NODES, *DEFINE_COORDINATE_SYSTEM or *DEFINE_COORDINATE_VECTOR). Available in R3 version of 971 and later.
REF	Use reference geometry to initialize the stress tensor. The reference geometry is defined by the keyword:*INITIAL_FOAM_REFERENCE_GEOMETRY (see for more details). EQ.0.0: off, EQ.1.0: on.

VARIABLE	DESCRIPTION
MACF	Material axes change flag for brick elements: EQ.1: No change, default, EQ.2: switch material axes a and b, EQ.3: switch material axes a and c, EQ.4: switch material axes b and c.
XP,YP,ZP	Coordinates of point **p** for AOPT = 1.
A1,A2,A3	Components of vector **a** for AOPT = 2.
V1,V2,V3	Components of vector **v** for AOPT = 3.
D1,D2,D3	Components of vector **d** for AOPT = 2.
BETA	Material angle in degrees for AOPT = 3, may be overridden on the element card, see *ELEMENT_SHELL_BETA or *ELEMENT_SOLID_ORTHO.
EAi	E_a, Young's modulus in a-direction at temperature Ti.
EBi	E_b, Young's modulus in b-direction at temperature Ti.
ECi	E_c, Young's modulus in c-direction at temperature Ti.
PRBAi	ν_{ba}, Poisson's ratio ba at temperature Ti.
PRCAi	ν_{ca}, Poisson's ratio ca at temperature Ti.
PRCBi	ν_{cb}, Poisson's ratio cb at temperature Ti.
AAi	α_a, coefficient of thermal expansion in a-direction at temperature Ti.
ABi	α_b, coefficient of thermal expansion in b-direction at temperature Ti.
ACi	α_c, coefficient of thermal expansion in c-direction at temperature Ti.
GABi	G_{ab}, Shear modulus ab at temperature Ti.
GBCi	G_{bc}, Shear modulus bc at temperature Ti.
GCAi	G_{ca}, Shear modulus ca at temperature Ti.
Ti	ith temperature

Remarks:

In the implementation for three-dimensional continua a total Lagrangian formulation is used. In this approach the material law that relates second Piola-Kirchhoff stress S to the Green-St. Venant strain E is

$$S = C \cdot E = T^t C_l T \cdot E$$

where T is the transformation matrix [Cook 1974].

$$T = \begin{bmatrix} l_1^2 & m_1^2 & n_1^2 & l_1 m_1 & m_1 n_1 & n_1 l_1 \\ l_2^2 & m_2^2 & n_2^2 & l_2 m_2 & m_2 n_2 & n_2 l_2 \\ l_3^2 & m_3^2 & n_3^2 & l_3 m_3 & m_3 n_3 & n_3 l_3 \\ 2l_1 l_2 & 2m_1 m_2 & 2n_1 n_2 & (l_1 m_2 + l_2 m_1) & (m_1 n_2 + m_2 n_1) & (n_1 l_2 + n_2 l_1) \\ 2l_2 l_3 & 2m_2 m_3 & 2n_2 n_3 & (l_2 m_3 + l_3 m_2) & (m_2 n_3 + m_3 n_2) & (n_2 l_3 + n_3 l_2) \\ 2l_3 l_1 & 2m_3 m_1 & 2n_3 n_1 & (l_3 m_1 + l_1 m_3) & (m_3 n_1 + m_1 n_3) & (n_3 l_1 + n_1 l_3) \end{bmatrix}$$

l_i, m_i, n_i are the direction cosines

$$x_i' = l_i x_1 + m_i x_2 + n_i x_3 \quad for\ i = 1,2,3$$

and x_i' denotes the material axes. The temperature dependent constitutive matrix C_l is defined in terms of the material axes as

$$C_l^{-1} = \begin{bmatrix} \frac{1}{E_{11}(T)} & -\frac{\upsilon_{13}(T)}{E_{11}(T)} & -\frac{\upsilon_{31}(T)}{E_{33}(T)} & 0 & 0 & 0 \\ -\frac{\upsilon_{12}(T)}{E_{11}} & \frac{1}{E_{22}(T)} & -\frac{\upsilon_{32}(T)}{E_{33}(T)} & 0 & 0 & 0 \\ -\frac{\upsilon_{13}(T)}{E_{11}(T)} & -\frac{\upsilon_{23}(T)}{E_{22}(T)} & \frac{1}{E_{33}(T)} & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1}{G_{12}(T)} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1}{G_{23}(T)} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1}{G_{31}(T)} \end{bmatrix}$$

where the subscripts denote the material axes, i.e.,

$$\upsilon_{ij} = \upsilon_{x_i' x_j'} \quad \text{and} \quad E_{ii} = E_{x_i'}$$

Since C_l is symmetric

$$\frac{\upsilon_{12}}{E_{11}} = \frac{\upsilon_{21}}{E_{22}}, \text{ etc.}$$

The vector of Green-St. Venant strain components is

$$E^t = \lfloor E_{11}, E_{22}, E_{33}, E_{12}, E_{23}, E_{31} \rfloor$$

which include the local thermal strains which are integrated in time:

$$\varepsilon_{aa}^{n+1} = \varepsilon_{aa}^{n} + \alpha_a \left(T^{n+\frac{1}{2}} \right) \left[T^{n+1} - T^n \right]$$

$$\varepsilon_{bb}^{n+1} = \varepsilon_{bb}^{n} + \alpha_b \left(T^{n+\frac{1}{2}} \right) \left[T^{n+1} - T^n \right]$$

$$\varepsilon_{cc}^{n+1} = \varepsilon_{cc}^{n} + \alpha_c \left(T^{n+\frac{1}{2}} \right) \left[T^{n+1} - T^n \right]$$

After computing S_{ij} we then obtain the Cauchy stress:

$$\sigma_{ij} = \frac{\rho}{\rho_0} \frac{\partial x_i}{\partial X_k} \frac{\partial x_j}{\partial X_l} S_{kl}$$

This model will predict realistic behavior for finite displacement and rotations as long as the strains are small.

For shell elements, the stresses are integrated in time and are updated in the corotational coordinate system. In this procedure the local material axes are assumed to remain orthogonal in the deformed configuration. This assumption is valid if the strains remain small.

*MAT_PIECEWISE_LINEAR_PLASTICITY

This is Material Type 24. An elasto-plastic material with an arbitrary stress versus strain curve and arbitrary strain rate dependency can be defined. See also Remark below. Also, failure based on a plastic strain or a minimum time step size can be defined. For another model with a more comprehensive failure criteria see MAT_MODIFIED_PIECEWISE_LINEAR_PLASTICITY. If considering laminated or sandwich shells with non-uniform material properties (this is defined through the user specified integration rule), the model, MAT_LAYERED_LINEAR_ PLASTICITY, is recommended. If solid elements are used and if the elastic strains before yielding are finite, the model, MAT_FINITE_ELASTIC_STRAIN_PLASTICITY, treats the elastic strains using a hyperelastic formulation.

Card 1

1	2	3	4	5	6	7	8
MID	RO	E	PR	SIGY	ETAN	FAIL	TDEL
A8	F	F	F	F	F	F	F
none	none	none	none	none	0.0	10.E+20	0

Card 2

1	2	3	4	5	6	7	8
C	P	LCSS	LCSR	VP			
F	F	F	F	F			
0	0	0	0	0			

Card 3

1	2	3	4	5	6	7	8
EPS1	EPS2	EPS3	EPS4	EPS5	EPS6	EPS7	EPS8
F	F	F	F	F	F	F	F
0	0	0	0	0	0	0	0

Card 4	1	2	3	4	5	6	7	8
Variable	ES1	ES2	ES3	ES4	ES5	ES6	ES7	ES8
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.
SIGY	Yield stress.
ETAN	Tangent modulus, ignored if (LCSS.GT.0) is defined.
FAIL	Failure flag. LT.0.0: User defined failure subroutine is called to determine failure EQ.0.0: Failure is not considered. This option is recommended if failure is not of interest since many calculations will be saved. GT.0.0: Plastic strain to failure. When the plastic strain reaches this value, the element is deleted from the calculation.
TDEL	Minimum time step size for automatic element deletion.
C	Strain rate parameter, C, see formula below.
P	Strain rate parameter, P, see formula below.
LCSS	Load curve ID or Table ID (optional; supersedes SIGY, ETAN, EPS1-8, ES1-8). Load curve ID defining effective stress versus effective plastic strain. If defined EPS1-EPS8 and ES1-ES8 are ignored. The table ID defines for each strain rate value a load curve ID giving the stress versus effective plastic strain for that rate, See Figure 24.1. The stress versus effective plastic strain curve for the lowest value of strain rate is used if the strain rate falls below the minimum value. Likewise, the stress versus effective plastic strain curve for the highest value of strain

VARIABLE	DESCRIPTION
	rate is used if the strain rate exceeds the maximum value. The strain rate parameters: C and P; the curve ID, LCSR; EPS1-EPS8 and ES1-ES8 are ignored if a Table ID is defined. NOTE: The strain rate values defined in the table may be given as the natural logarithm of the strain rate. If the *first* stress-strain curve in the table corresponds to a negative strain rate, LS-DYNA assumes that the natural logarithm of the strain rate value is used. Since the tables are internally discretized to equally space the points, natural logarithms are necessary, for example, if the curves correspond to rates from 10.e-04 to 10.e+04. Computing the natural logarithm of the strain rate does slow the stress update down significantly on some computers.
LCSR	Load curve ID defining strain rate scaling effect on yield stress.
VP	Formulation for rate effects: EQ.-1.0: Cowper-Symonds with deviatoric strain rate rather than total. EQ.0.0: Scale yield stress (default), EQ.1.0: Viscoplastic formulation.
EPS1-EPS8	Effective plastic strain values (optional; supersedes SIGY, ETAN). At least 2 points should be defined. The first point must be zero corresponding to the initial yield stress. WARNING: If the first point is nonzero the yield stress is extrapolated to determine the initial yield. If this option is used SIGY and ETAN are ignored and may be input as zero.
ES1-ES8	Corresponding yield stress values to EPS1 - EPS8.

Remarks:

The stress strain behavior may be treated by a bilinear stress strain curve by defining the tangent modulus, ETAN. Alternately, a curve of effective stress vs. effective plastic strain similar to that shown in Figure 10.1 may be defined by (EPS1,ES1) - (EPS8,ES8); however, a curve ID (LCSS) may be referenced instead if eight points are insufficient. The cost is roughly the same for either approach. Note that in the special case of uniaxial stress, true stress vs. true plastic strain is equivalent to effective stress vs. effective plastic strain. The most general approach is to use the table definition (LCSS) discussed below.

Three options to account for strain rate effects are possible.

I. Strain rate may be accounted for using the Cowper and Symonds model which scales the yield stress with the factor

$$1+\left(\frac{\dot{\varepsilon}}{C}\right)^{1/p}$$

where $\dot{\varepsilon}$ is the strain rate. $\dot{\varepsilon}=\sqrt{\dot{\varepsilon}_{ij}\dot{\varepsilon}_{ij}}$. If VP=-1. The deviatoric strain rates are used instead.

If the viscoplastic option is active, VP=1.0, and if SIGY is > 0 then the dynamic yield stress is computed from the sum of the static stress, $\sigma_y^s\left(\varepsilon_{eff}^p\right)$, which is typically given by a load curve ID, and the initial yield stress, SIGY, multiplied by the Cowper-Symonds rate term as follows:

$$\sigma_y\left(\varepsilon_{eff}^p,\dot{\varepsilon}_{eff}^p\right)=\sigma_y^s\left(\varepsilon_{eff}^p\right)+SIGY\cdot\left(\frac{\dot{\varepsilon}_{eff}^p}{C}\right)^{1/p}$$

where the plastic strain rate is used. With this latter approach similar results can be obtained between this model and material model: *MAT_ANISOTROPIC_VISCOPLASTIC. If SIGY=0, the following equation is used instead where the static stress, $\sigma_y^s\left(\varepsilon_{eff}^p\right)$, must be defined by a load curve:

$$\sigma_y\left(\varepsilon_{eff}^p,\dot{\varepsilon}_{eff}^p\right)=\sigma_y^s\left(\varepsilon_{eff}^p\right)\left[1+\left(\frac{\dot{\varepsilon}_{eff}^p}{C}\right)^{1/p}\right]$$

This latter equation is always used if the viscoplastic option is off.

II. For complete generality a load curve (LCSR) to scale the yield stress may be input instead. In this curve the scale factor versus strain rate is defined.

III. If different stress versus strain curves can be provided for various strain rates, the option using the reference to a table (LCSS) can be used. Then the table input in *DEFINE_TABLE has to be used, see Figure 24.1.

A fully viscoplastic formulation is optional (variable VP) which incorporates the different options above within the yield surface. An additional cost is incurred over the simple scaling but the improvement is results can be dramatic.

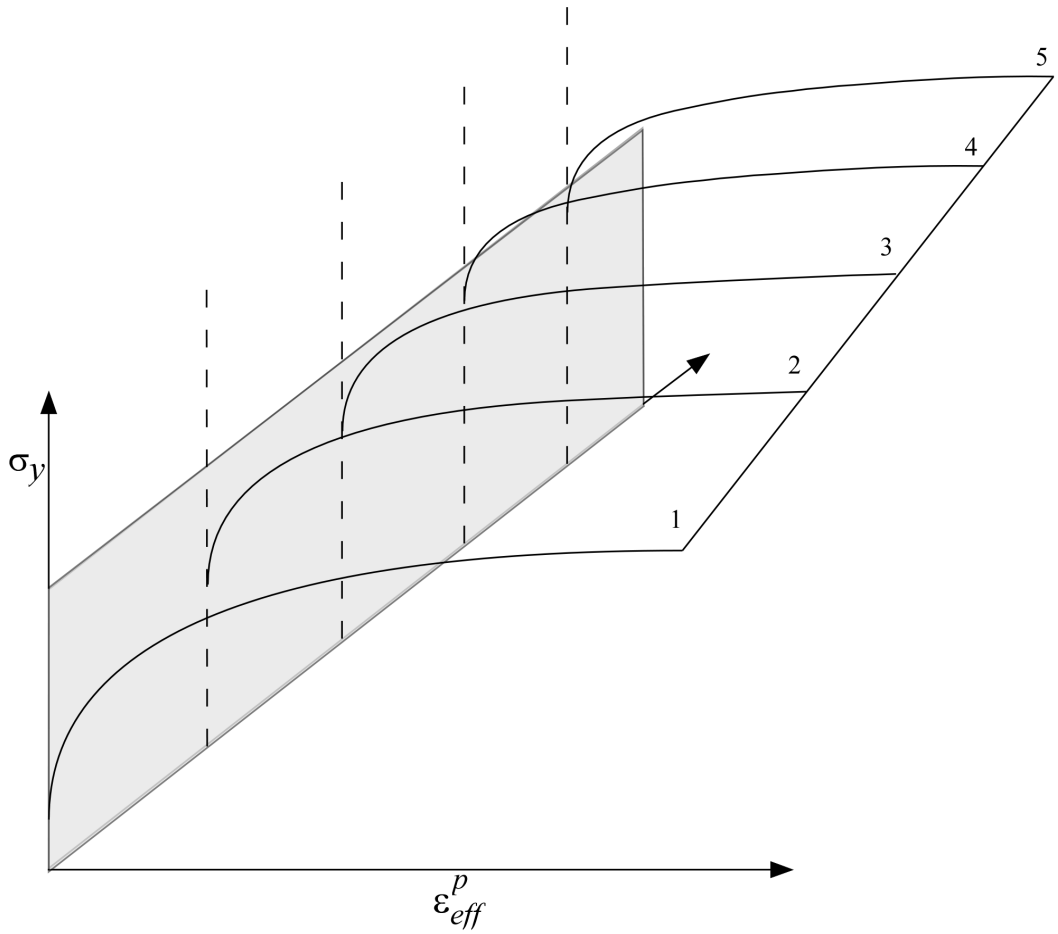


Figure 24.1. Rate effects may be accounted for by defining a table of curves. If a table ID is specified a curve ID is given for each strain rate, see *DEFINE_TABLE. Intermediate values are found by interpolating between curves. Effective plastic strain versus yield stress is expected. If the strain rate values fall out of range, extrapolation is not used; rather, either the first or last curve determines the yield stress depending on whether the rate is low or high, respectively.

*MAT_GEOLOGIC_CAP_MODEL

This is Material Type 25. This is an inviscid two invariant geologic cap model. This material model can be used for geomechanical problems or for materials as concrete, see references cited below.

Card 1

Variable	MID	RO	BULK	G	ALPHA	THETA	GAMMA	BETA
Type	A8	F	F	F	F	F	F	F

Card 2

Variable	R	D	W	X0	C	N		
Type	F	F	F	F	F	F		

Card 3

Variable	PLOT	FTYPE	VEC	TOFF				
Type	F	F	F	F				

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
BULK	Initial bulk modulus, K.
G	Initial Shear modulus.
ALPHA	Failure envelope parameter, α.
THETA	Failure envelope linear coefficient, θ.
GAMMA	Failure envelope exponential coefficient, γ.

VARIABLE	DESCRIPTION
BETA	Failure envelope exponent, β.
R	Cap, surface axis ratio.
D	Hardening law exponent.
W	Hardening law coefficient.
X0	Hardening law exponent, X_0.
C	Kinematic hardening coefficient, $\bar{c}$.
N	Kinematic hardening parameter.
PLOT	Save the following variable for plotting in LS-Prepost, to be labeled there as "effective plastic strain:" EQ.1: hardening parameter, κ, EQ.2: cap -J_1 axis intercept, X (κ), EQ.3: volumetric plastic strain ε_v^p, EQ.4: first stress invariant, J_1, EQ.5: second stress invariant, $\sqrt{J_2}$. EQ.6: not used EQ.7: not used EQ.8: response mode number EQ.9: number of iterations.
FTYPE	Formulation flag: EQ.1: soils (Cap surface may contract), EQ.2: concrete and rock (Cap doesn't contract).
VEC	Vectorization flag: EQ.0: vectorized (fixed number of iterations), EQ.1: fully iterative, If the vectorized solution is chosen, the stresses might be slightly off the yield surface; however, on vector computers a much more efficient solution is achieved.
TOFF	Tension Cut Off, TOFF < 0 (positive in compression).

Remarks:

The implementation of an extended two invariant cap model, suggested by Stojko [1990], is based on the formulations of Simo, et al. [1988, 1990] and Sandler and Rubin [1979]. In this model, the two invariant cap theory is extended to include nonlinear kinematic hardening as suggested by Isenberg, Vaughan, and Sandler [1978]. A brief discussion of the extended cap model and its parameters is given below.

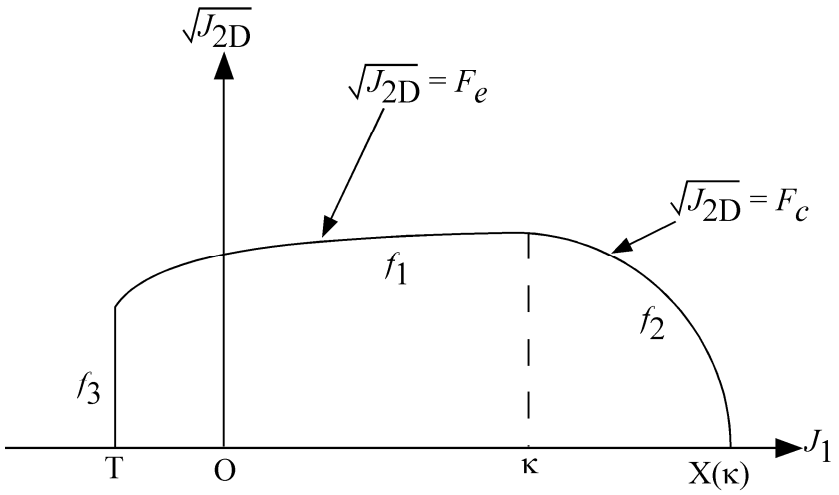


Figure 25.1. The yield surface of the two-invariant cap model in pressure $\sqrt{J_{2D}} - J_1$ space. Surface f_1 is the failure envelope, f_2 is the cap surface, and f_3 is the tension cutoff.

The cap model is formulated in terms of the invariants of the stress tensor. The square root of the second invariant of the deviatoric stress tensor, $\sqrt{J_{2D}}$ is found from the deviatoric stresses **s** as

$$\sqrt{J_{2D}} \equiv \sqrt{\frac{1}{2} S_{ij} S_{ij}}$$

and is the objective scalar measure of the distortional or shearing stress. The first invariant of the stress, J_1, is the trace of the stress tensor.

The cap model consists of three surfaces in $\sqrt{J_{2D}} - J_1$ space, as shown in Figure 25.1. First, there is a failure envelope surface, denoted f_1 in the figure. The functional form of f_1 is

$$f_1 = \sqrt{J_{2D}} - \min\left(F_e(J_1), T_{mises}\right) ,$$

where F_e is given by

$$F_e(J_1) \equiv \alpha - \gamma \exp(-\beta J_1) + \theta J_1$$

and $T_{mises} \equiv \left| X(\kappa_n) - L(\kappa_n) \right|$. This failure envelop surface is fixed in $\sqrt{J_{2D}} - J_1$ space, and therefore does not harden unless kinematic hardening is present. Next, there is a cap surface, denoted f_2 in the figure, with f_2 given by

$$f_2 = \sqrt{J_{2D}} - F_c(J_1, K)$$

where F_c is defined by

$$F_c(J_1,\kappa) \equiv \frac{1}{R}\sqrt{\left[X(\kappa)-L(\kappa)\right]^2-\left[J_1-L(\kappa)\right]^2},$$

$X(\kappa)$ is the intersection of the cap surface with the J_1 axis

$$X(\kappa) = \kappa + RF_e(\kappa),$$

and $L(\kappa)$ is defined by

$$L(\kappa) \equiv \begin{cases} \kappa \text{ if } \kappa > 0 \\ 0 \text{ if } \kappa \le 0 \end{cases}$$

The hardening parameter κ is related to the plastic volume change ε_v^p through the hardening law

$$\varepsilon_v^p = W\left\{1-\exp\left[-D\left(X(\kappa)-X_0\right)\right]\right\}$$

Geometrically, κ is seen in the figure as the J_1 coordinate of the intersection of the cap surface and the failure surface. Finally, there is the tension cutoff surface, denoted f_3 in the figure. The function f_3 is given by

$$f_3 \equiv T\text{-}J_1$$

where T is the input material parameter which specifies the maximum hydrostatic tension sustainable by the material. The elastic domain in $\sqrt{J_{2D}} - J_1$ space is then bounded by the failure envelope surface above, the tension cutoff surface on the left, and the cap surface on the right.

An additive decomposition of the strain into elastic and plastic parts is assumed:

$$\boldsymbol{\varepsilon} = \boldsymbol{\varepsilon}^e + \boldsymbol{\varepsilon}^p,$$

where $\boldsymbol{\varepsilon}^e$ is the elastic strain and $\boldsymbol{\varepsilon}^p$ is the plastic strain. Stress is found from the elastic strain using Hooke's law,

$$\boldsymbol{\sigma} = \mathbf{C}(\boldsymbol{\varepsilon} - \boldsymbol{\varepsilon}^p),$$

where $\boldsymbol{\sigma}$ is the stress and $\mathbf{C}$ is the elastic constitutive tensor.

The yield condition may be written

$$f_1(s) \le 0$$
$$f_2(s,\kappa) \le 0$$
$$f_3(s) \le 0$$

and the plastic consistency condition requires that

$$\dot{\lambda}_k f_k = 0$$
$$k = 1,2,3$$
$$\dot{\lambda}_k \ge 0$$

where λ_k is the plastic consistency parameter for surface k. If $f_k < 0$ then, $\dot{\lambda}_k = 0$ and the response is elastic. If $f_k > 0$ then surface k is active and $\dot{\lambda}_k$ is found from the requirement that $\dot{f}_k = 0$.

Associated plastic flow is assumed, so using Koiter's flow rule the plastic strain rate is given as the sum of contribution from all of the active surfaces,

$$\dot{\varepsilon}^p = \sum_{k=1}^{3} \dot{\lambda}_k \frac{\partial f_k}{\partial s}.$$

One of the major advantages of the cap model over other classical pressure-dependent plasticity models is the ability to control the amount of dilatancy produced under shear loading. Dilatancy is produced under shear loading as a result of the yield surface having a positive slope in $\sqrt{J_{2D}} - J$ space, so the assumption of plastic flow in the direction normal to the yield surface produces a plastic strain rate vector that has a component in the volumetric (hydrostatic) direction (see Figure 25.1). In models such as the Drucker-Prager and Mohr-Coulomb, this dilatancy continues as long as shear loads are applied, and in many cases produces far more dilatancy than is experimentally observed in material tests. In the cap model, when the failure surface is active, dilatancy is produced just as with the Drucker-Prager and Mohr-Coulumb models. However, the hardening law permits the cap surface to contract until the cap intersects the failure envelope at the stress point, and the cap remains at that point. The local normal to the yield surface is now vertical, and therefore the normality rule assures that no further plastic volumetric strain (dilatancy) is created. Adjustment of the parameters that control the rate of cap contractions permits experimentally observed amounts of dilatancy to be incorporated into the cap model, thus producing a constitutive law which better represents the physics to be modeled.

Another advantage of the cap model over other models such as the Drucker-Prager and Mohr-Coulomb is the ability to model plastic compaction. In these models all purely volumetric response is elastic. In the cap model, volumetric response is elastic until the stress point hits the cap surface. Therefore, plastic volumetric strain (compaction) is generated at a rate controlled by the hardening law. Thus, in addition to controlling the amount of dilatancy, the introduction of the cap surface adds another experimentally observed response characteristic of geological material into the model.

The inclusion of kinematic hardening results in hysteretic energy dissipation under cyclic loading conditions. Following the approach of Isenberg, et al. [1978] a nonlinear kinematic hardening law is used for the failure envelope surface when nonzero values of and N are specified. In this case, the failure envelope surface is replaced by a family of yield surfaces bounded by an initial yield surface and a limiting failure envelope surface. Thus, the shape of the yield surfaces described above remains unchanged, but they may translate in a plane orthogonal to the J axis,

Translation of the yield surfaces is permitted through the introduction of a "back stress" tensor, $\boldsymbol{\alpha}$ The formulation including kinematic hardening is obtained by replacing the stress $\boldsymbol{\sigma}$ with the translated stress tensor $\eta \equiv \sigma - \alpha$ in all of the above equation. The history tensor $\boldsymbol{\alpha}$ is assumed deviatoric, and therefore has only 5 unique components. The evolution of the back stress tensor is governed by the nonlinear hardening law

$$\alpha = \bar{c}\bar{F}\left(\sigma, \alpha\right)\dot{e}^{p}$$

where $\bar{c}$ is a constant, $\bar{F}$ is a scalar function of $\boldsymbol{\sigma}$ and $\boldsymbol{\alpha}$ and $\dot{e}^{p}$ is the rate of deviatoric plastic strain. The constant may be estimated from the slope of the shear stress - plastic shear strain curve at low levels of shear stress.

The function $\bar{F}$ is defined as

$$\bar{F} \equiv \max\left(0, 1 - \frac{\left(\sigma - \alpha\right) \bullet \alpha}{2NF_{e}\left(J_{1}\right)}\right)$$

where N is a constant defining the size of the yield surface. The value of N may be interpreted as the radial distant between the outside of the initial yield surface and the inside of the limit surface. In order for the limit surface of the kinematic hardening cap model to correspond with the failure envelope surface of the standard cap model, the scalar parameter a must be replaced α - N in the definition F_e.

The cap model contains a number of parameters which must be chosen to represent a particular material, and are generally based on experimental data. The parameters α, β, θ, and γ are usually evaluated by fitting a curve through failure data taken from a set of triaxial compression tests. The parameters W, D, and X_0 define the cap hardening law. The value W represents the void fraction of the uncompressed sample and D governs the slope of the initial loading curve in hydrostatic compression. The value of R is the ration of major to minor axes of the quarter ellipse defining the cap surface. Additional details and guidelines for fitting the cap model to experimental data are found in Chen and Baladi [1985].

*MAT_HONEYCOMB

This is Material Type 26. The major use of this material model is for honeycomb and foam materials with real anisotropic behavior. A nonlinear elastoplastic material behavior can be defined separately for all normal and shear stresses. These are considered to be fully uncoupled. See notes below.

Card 1

	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	SIGY	VF	MU	BULK
Type	A8	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	.05	0.0

Card 2

Variable	LCA	LCB	LCC	LCS	LCAB	LCBC	LCCA	LCSR
Type	F	F	F	F	F	F	F	F
Default	none	LCA	LCA	LCA	LCS	LCS	LCS	optional

Card 3

Variable	EAAU	EBBU	ECCU	GABU	GBCU	GCAU	AOPT	MACF
Type	F	F	F	F	F	F		I

Card 4

Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card 5	1	2	3	4	5	6	7	8
Variable	D1	D2	D3	TSEF	SSEF	V1	V2	V3
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
E	Young's modulus for compacted honeycomb material.
PR	Poisson's ratio for compacted honeycomb material.
SIGY	Yield stress for fully compacted honeycomb.
VF	Relative volume at which the honeycomb is fully compacted.
MU	μ, material viscosity coefficient. (default=.05) Recommended.
BULK	Bulk viscosity flag: EQ.0.0: bulk viscosity is not used. This is recommended. EQ.1.0: bulk viscosity is active and μ=0. This will give results identical to previous versions of LS-DYNA.
LCA	Load curve ID, see *DEFINE_CURVE, for sigma-aa versus either relative volume or volumetric strain. See notes below.
LCB	Load curve ID, see *DEFINE_CURVE, for sigma-bb versus either relative volume or volumetric strain. Default LCB=LCA. See notes below.
LCC	Load curve ID, see *DEFINE_CURVE, for sigma-cc versus either relative volume or volumetric strain. Default LCC=LCA. See notes below.
LCS	Load curve ID, see *DEFINE_CURVE, for shear stress versus either relative volume or volumetric strain. Default LCS=LCA. Each component of shear stress may have its own load curve. See notes below.

VARIABLE	DESCRIPTION
LCAB	Load curve ID, see *DEFINE_CURVE, for sigma-ab versus either relative volume or volumetric strain. Default LCAB=LCS. See notes below.
LCBC	Load curve ID, see *DEFINE_CURVE, for sigma-bc versus either relative volume or volumetric strain. Default LCBC=LCS. See notes below.
LCCA	Load curve ID, see *DEFINE_CURVE, or sigma-ca versus either relative volume or volumetric strain. Default LCCA=LCS. See notes below.
LCSR	Load curve ID, see *DEFINE_CURVE, for strain-rate effects defining the scale factor versus strain rate. This is optional. The curves defined above are scaled using this curve.
EAAU	Elastic modulus E_{aau} in uncompressed configuration.
EBBU	Elastic modulus E_{bbu} in uncompressed configuration.
ECCU	Elastic modulus E_{ccu} in uncompressed configuration.
GABU	Shear modulus G_{abu} in uncompressed configuration.
GBCU	Shear modulus G_{bcu} in uncompressed configuration.
GCAU	Shear modulus G_{cau} in uncompressed configuration.
AOPT	Material axes option (see MAT_*OPTION* TROPIC_ELASTIC for a more complete description): EQ.0.0: locally orthotropic with material axes determined by element nodes 1, 2, and 4, as with *DEFINE_COORDINATE_NODES. EQ.1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center; this is the a-direction. EQ.2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR. EQ.3.0: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle, BETA, from a line in the plane of the element defined by the cross product of the vector **v** with the element normal. The plane of a solid element is the midsurface between the inner surface and outer surface defined by the first four nodes and the last four nodes of the connectivity of the element, respectively.

EQ.4.0: locally orthotropic in cylindrical coordinate system with the material axes determined by a vector **v**, and an originating point, P, which define the centerline axis. This option is for solid elements only.
LT.0.0: the absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_NODES, *DEFINE_COORDINATE_SYSTEM or *DEFINE_COORDINATE_VECTOR). Available in R3 version of 971 and later.

VARIABLE	DESCRIPTION
MACF	Material axes change flag: EQ.1: No change, default, EQ.2: switch material axes a and b, EQ.3: switch material axes a and c, EQ.4: switch material axes b and c.
XP YP ZP	Coordinates of point **p** for AOPT = 1.
A1 A2 A3	Components of vector **a** for AOPT = 2.
D1 D2 D3	Components of vector **d** for AOPT = 2.
V1 V2 V3	Define components of vector **v** for AOPT = 3 and 4.
TSEF	Tensile strain at element failure (element will erode).
SSEF	Shear strain at element failure (element will erode).

Remarks:

For efficiency it is strongly recommended that the load curve ID's: LCA, LCB, LCC, LCS, LCAB, LCBC, and LCCA, contain exactly the same number of points with corresponding strain values on the abscissa. If this recommendation is followed the cost of the table lookup is insignificant. Conversely, the cost increases significantly if the abscissa strain values are not consistent between load curves.

The behavior before compaction is orthotropic where the components of the stress tensor are uncoupled, i.e., an *a* component of strain will generate resistance in the local *a*-direction with no coupling to the local *b* and *c* directions. The elastic moduli vary, from their initial values to the fully compacted values at V_f, linearly with the relative volume V:

$$E_{aa} = E_{aau} + \beta\left(E - E_{aau}\right)$$

$$E_{bb} = E_{bbu} + \beta\left(E - E_{bbu}\right)$$

$$E_{cc} = E_{ccu} + \beta\left(E - E_{ccu}\right)$$

$$G_{ab} = E_{abu} + \beta\left(G - G_{abu}\right)$$

$$G_{bc} = E_{bcu} + \beta\left(G - G_{bcu}\right)$$

$$G_{ca} = E_{cau} + \beta\left(G - G_{cau}\right)$$

where

$$\beta = \max\left[\min\left(\frac{1-V}{1-V_f}, 1\right), 0\right]$$

and G is the elastic shear modulus for the fully compacted honeycomb material

$$G = \frac{E}{2\left(1+\nu\right)}.$$

The relative volume, V, is defined as the ratio of the current volume to the initial volume. Typically, V=1 at the beginning of a calculation. The viscosity coefficient μ (MU) should be set to a small number (usually .02-.10 is okay). Alternatively, the two bulk viscosity coefficients on the control cards should be set to very small numbers to prevent the development of spurious pressures that may lead to undesirable and confusing results. The latter is not recommended since spurious numerical noise may develop.

The load curves define the magnitude of the average stress as the material changes density (relative volume), see Figure 26.1. Each curve related to this model must have the same number of points and the same abscissa values. There are two ways to define these curves, **a)** as a function of relative volume (V) or **b)** as a function of volumetric strain defined as:

$$\varepsilon_V = 1 - V$$

In the former, the first value in the curve should correspond to a value of relative volume slightly less than the fully compacted value. In the latter, the first value in the curve should be less than or equal to zero, corresponding to tension, and increase to full compaction. **Care should be taken when defining the curves so that extrapolated values do not lead to negative yield stresses.**

At the beginning of the stress update each element's stresses and strain rates are transformed into the local element coordinate system. For the uncompacted material, the trial stress components are updated using the elastic interpolated moduli according to:

$$\sigma_{aa}^{n+1^{trial}} = \sigma_{aa}^{n} + E_{aa}\Delta\varepsilon_{aa}$$

$$\sigma_{bb}^{n+1^{trial}} = \sigma_{bb}^{n} + E_{bb}\Delta\varepsilon_{bb}$$

$$\sigma_{cc}^{n+1^{trial}} = \sigma_{cc}^{n} + E_{cc}\Delta\varepsilon_{cc}$$

$$\sigma_{ab}^{n+1^{trial}} = \sigma_{ab}^{n} + 2G_{ab}\Delta\varepsilon_{ab}$$

$$\sigma_{bc}^{n+1^{trial}} = \sigma_{bc}^{n} + 2G_{bc}\Delta\varepsilon_{bc}$$

$$\sigma_{ca}^{n+1^{trial}} = \sigma_{ca}^{n} + 2G_{ca}\Delta\varepsilon_{ca}$$

Each component of the updated stresses is then independently checked to ensure that they do not exceed the permissible values determined from the load curves; e.g., if

$$\left|\sigma_{ij}^{n+1^{trial}}\right| > \lambda\sigma_{ij}(V)$$

then

$$\sigma_{ij}^{n+1} = \sigma_{ij}(V)\frac{\lambda\sigma_{ij}^{n+1^{trial}}}{\left|\lambda\sigma_{ij}^{n+1^{trial}}\right|}$$

On Card 2 $\sigma_{ij}(V)$ is defined by LCA for the aa stress component, LCB for the bb component, LCC for the cc component, and LCS for the ab, bc, ca shear stress components. The parameter λ is either unity or a value taken from the load curve number, LCSR, that defines λ as a function of strain-rate. Strain-rate is defined here as the Euclidean norm of the deviatoric strain-rate tensor.

For fully compacted material it is assumed that the material behavior is elastic-perfectly plastic and the stress components updated according to:

$$s_{ij}^{trial} = s_{ij}^{n} + 2G\Delta\varepsilon_{ij}^{dev^{n+1/2}}$$

where the deviatoric strain increment is defined as

$$\Delta\varepsilon_{ij}^{dev} = \Delta\varepsilon_{ij} - \frac{1}{3}\Delta\varepsilon_{kk}\delta_{ij}$$

Now a check is made to see if the yield stress for the fully compacted material is exceeded by comparing

$$s_{eff}^{trial} = \left(\tfrac{3}{2} s_{ij}^{trial} s_{ij}^{trial} \right)^{1/2}$$

the effective trial stress to the defined yield stress, SIGY. If the effective trial stress exceeds the yield stress the stress components are simply scaled back to the yield surface

$$s_{ij}^{n+1} = \frac{\sigma_y}{s_{eff}^{trial}} s_{ij}^{trial} .$$

Now the pressure is updated using the elastic bulk modulus, K

$$p^{n+1} = p^n - K \Delta \varepsilon_{kk}^{n+1/2}$$
$$K = \frac{E}{3(1-2\nu)}$$

to obtain the final value for the Cauchy stress

$$\sigma_{ij}^{n+1} = s_{ij}^{n+1} - p^{n+1} \delta_{ij}$$

After completing the stress update transform the stresses back to the global configuration.

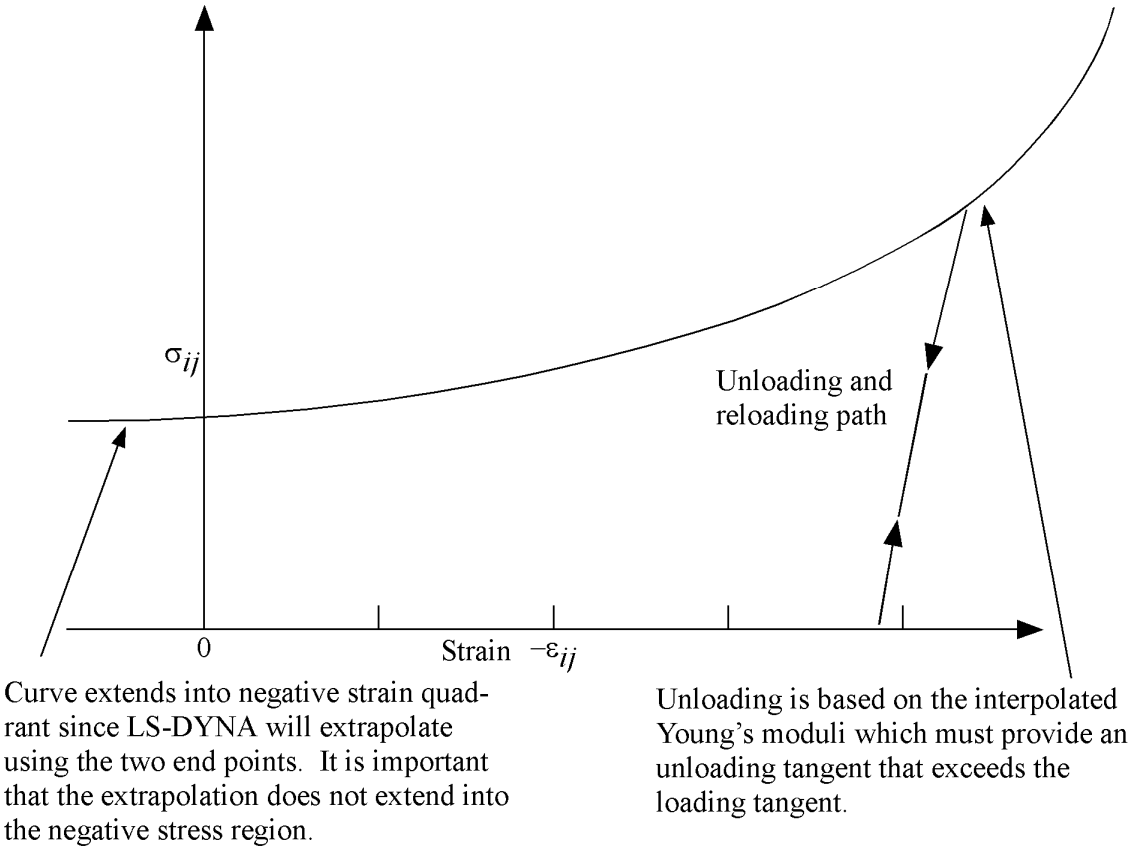


Figure 26.1. Stress quantity versus volumetric strain. Note that the "yield stress" at a volumetric strain of zero is non-zero. In the load curve definition, see *DEFINE_CURVE, the "time" value is the volumetric strain and the "function" value is the yield stress.

*MAT_MOONEY-RIVLIN_RUBBER

This is Material Type 27. A two-parametric material model for rubber can be defined.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	PR	A	B	REF		
Type	A8	F	F	F	F	F		

Card 2

Variable	SGL	SW	ST	LCID				
Type	F	F	F	F				

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
PR	Poisson's ratio (value between 0.49 and 0.5 is recommended, smaller values may not work).
A	Constant, see literature and equations defined below.
B	Constant, see literature and equations defined below.
REF	Use reference geometry to initialize the stress tensor. The reference geometry is defined by the keyword:*INITIAL_FOAM_REFERENCE_GEOMETRY (see there for more details). EQ.0.0: off, EQ.1.0: on.

If A=B=0.0, then a least square fit is computed from tabulated uniaxial data via a load curve. The following information should be defined.

SGL	Specimen gauge length l_0, see Figure 27.1.

VARIABLE	DESCRIPTION
SW	Specimen width, see Figure 27.1.
ST	Specimen thickness, see Figure 27.1.
LCID	Load curve ID, see *DEFINE_CURVE, giving the force versus actual change ΔL in the gauge length. See also Figure 27.2 for an alternative definition.

Remarks:

The strain energy density function is defined as:

$$W = A(I-3) + B(II-3) + C\left(III^{-2}-1\right) + D\left(III-1\right)^2$$

where

$$C = 0.5\ A + B$$

$$D = \frac{A(5\upsilon-2) + B(11\upsilon-5)}{2(1-2\upsilon)}$$

υ =Poisson's ratio

2(A+B) = shear modulus of linear elasticity

I, II, III = invariants of right Cauchy-Green Tensor *C*.

The load curve definition that provides the uniaxial data should give the change in gauge length, ΔL, versus the corresponding force. In compression both the force and the change in gauge length must be specified as negative values. In tension the force and change in gauge length should be input as positive values. The principal stretch ratio in the uniaxial direction, λ_1, is then given by

$$\lambda_1 = \frac{L_0 + \Delta L}{L_0}$$

with L_0 being the initial length and L being the actual length.

Alternatively, the stress versus strain curve can also be input by setting the gauge length, thickness, and width to unity (1.0) and defining the engineering strain in place of the change in gauge length and the nominal (engineering) stress in place of the force, see Figure 27.2.

The least square fit to the experimental data is performed during the initialization phase and is a comparison between the fit and the actual input is provided in the d3hsp file. It is a good idea to visually check to make sure it is acceptable. The coefficients A and B are also printed in

the output file. It is also advised to use the material driver (see Appendix K) for checking out the material model.

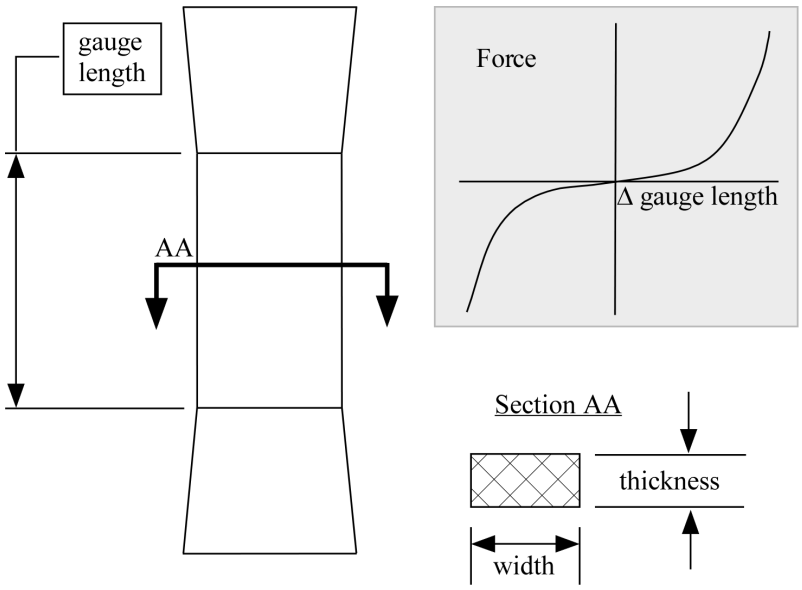


Figure 27.1. Uniaxial specimen for experimental data.

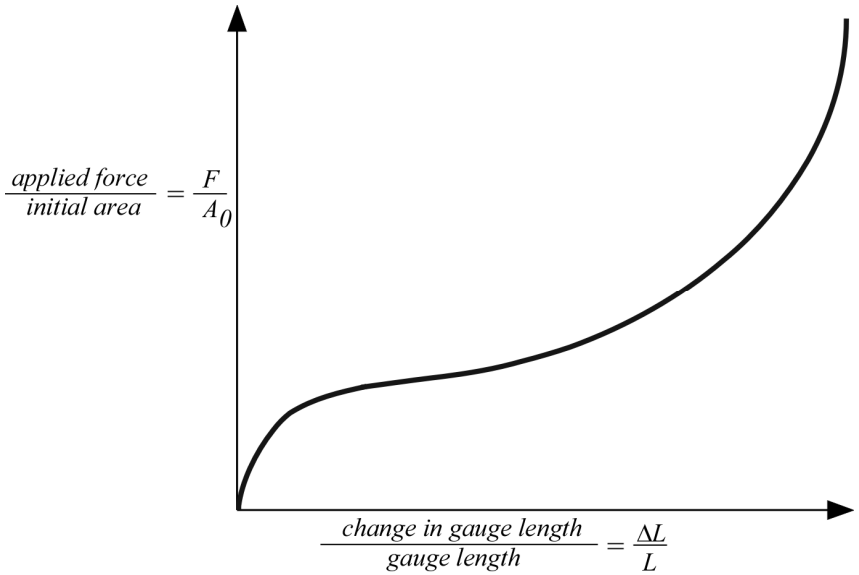


Figure 27.2. The stress versus strain curve can used instead of the force versus the change in the gauge length by setting the gauge length, thickness, and width to unity (1.0) and defining the engineering strain in place of the change in gauge length and the nominal (engineering) stress in place of the force. *MAT_077_O is a better alternative for fitting data resembling the curve above. *MAT_027 will provide a poor fit to a curve that exhibits an strong upturn in slope as strains become large.

*MAT_RESULTANT_PLASTICITY

This is Material Type 28. A resultant formulation for beam and shell elements including elasto-plastic behavior can be defined. This model is available for the Belytschko-Schwer beam, the C^0 triangular shell, the Belytschko-Tsay shell, and the fully integrated type 16 shell. For beams, the treatment is elastic-perfectly plastic, but for shell elements isotropic hardening is approximately modeled. For a detailed description we refer to the LS-DYNA Theory Manual. Since the stresses are not computed in the resultant formulation, the stresses output to the binary databases for the resultant elements are zero.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	SIGY	ETAN		
Type	A8	F	F	F	F	F		
Default	none	none	none	none	none	0.0		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
E	Young's modulus
PR	Poisson's ratio
SIGY	Yield stress
ETAN	Plastic hardening modulus (for shells only)

*MAT_FORCE_LIMITED

This is Material Type 29. With this material model, for the Belytschko-Schwer beam only, plastic hinge forming at the ends of a beam can be modeled using curve definitions. Optionally, collapse can also be modeled. See also *MAT_139.

Description: FORCE LIMITED Resultant Formulation

Card 1

Variable	MID	RO	E	PR	DF	AOPT	YTFLAG	ASOFT
Type	A8	F	F	F	F	F	F	F
Default	none	none	none	none	0.0	0.0	0.0	0.0

Card 2

Variable	M1	M2	M3	M4	M5	M6	M7	M8
Type	F	F	F	F	F	F	F	F
Default	none	0	0	0	0	0	0	0

Card 3

Variable	LC1	LC2	LC3	LC4	LC5	LC6	LC7	LC8
Type	F	F	F	F	F	F	F	F
Default	none	0	0	0	0	0	0	0

Card 4

	1	2	3	4	5	6	7	8
Variable	LPS1	SFS1	LPS2	SFS2	YMS1	YMS2		
Type	F	F	F	F	F	F		
Default	0	1.0	LPS1	1.0	1.0E+20	YMS1		

Card 5

Variable	LPT1	SFT1	LPT2	SFT2	YMT1	YMT2		
Type	F	F	F	F	F	F		
Default	0	1.0	LPT1	1.0	1.0E+20	YMT1		

Card 6

Variable	LPR	SFR	YMR					
Type	F	F	F					
Default	0	1.0	1.0E+20					

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
E	Young's modulus
PR	Poisson's ratio
DF	Damping factor, see definition in notes below. A proper control for the timestep has to be maintained by the user!

VARIABLE	DESCRIPTION
AOPT	Axial load curve option: EQ.0.0: axial load curves are force versus strain, EQ.1.0: axial load curves are force versus change in length. LT.0.0: the absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_NODES, *DEFINE_COORDINATE_SYSTEM or *DEFINE_COORDINATE_VECTOR). Available in R3 version of 971 and later.
YTFLAG	Flag to allow beam to yield in tension: EQ.0.0: beam does not yield in tension, EQ.1.0: beam can yield in tension.
ASOFT	Axial elastic softening factor applied once hinge has formed. When a hinge has formed the stiffness is reduced by this factor. If zero, this factor is ignored.
M1, M2,...,M8	Applied end moment for force versus (strain/change in length) curve. At least one must be defined. A maximum of 8 moments can be defined. The values should be in ascending order.
LC1, LC2,...,LC8	Load curve ID (see *DEFINE_CURVE) defining axial force (collapse load) versus strain/change in length (see AOPT) for the corresponding applied end moment. Define the same number as end moments. Each curve must contain the same number of points.
LPS1	Load curve ID for plastic moment versus rotation about s-axis at node 1. If zero, this load curve is ignored.
SFS1	Scale factor for plastic moment versus rotation curve about s-axis at node 1. Default = 1.0.
LPS2	Load curve ID for plastic moment versus rotation about s-axis at node 2. Default: is same as at node 1.
SFS2	Scale factor for plastic moment versus rotation curve about s-axis at node 2. Default: is same as at node 1.
YMS1	Yield moment about s-axis at node 1 for interaction calculations (default set to 1.0E+20 to prevent interaction).
YMS2	Yield moment about s-axis at node 2 for interaction calculations (default set to YMS1).
LPT1	Load curve ID for plastic moment versus rotation about t-axis at node 1. If zero, this load curve is ignored.

VARIABLE	DESCRIPTION
SFT1	Scale factor for plastic moment versus rotation curve about t-axis at node 1. Default = 1.0.
LPT2	Load curve ID for plastic moment versus rotation about t-axis at node 2. Default: is the same as at node 1.
SFT2	Scale factor for plastic moment versus rotation curve about t-axis at node 2. Default: is the same as at node 1.
YMT1	Yield moment about t-axis at node 1 for interaction calculations (default set to 1.0E+20 to prevent interactions)
YMT2	Yield moment about t-axis at node 2 for interaction calculations (default set to YMT1)
LPR	Load curve ID for plastic torsional moment versus rotation. If zero, this load curve is ignored.
SFR	Scale factor for plastic torsional moment versus rotation (default = 1.0).
YMR	Torsional yield moment for interaction calculations (default set to 1.0E+20 to prevent interaction)

Remarks:

This material model is available for the Belytschko resultant beam element only. Plastic hinges form at the ends of the beam when the moment reaches the plastic moment. The moment versus rotation relationship is specified by the user in the form of a load curve and scale factor. The points of the load curve are (plastic rotation in radians, plastic moment). Both quantities should be positive for all points, with the first point being (zero, initial plastic moment). Within this constraint any form of characteristic may be used, including flat or falling curves. Different load curves and scale factors may be specified at each node and about each of the local s and t axes.

Axial collapse occurs when the compressive axial load reaches the collapse load. Collapse load versus collapse deflection is specified in the form of a load curve. The points of the load curve are either (true strain, collapse force) or (change in length, collapse force). Both quantities should be entered as positive for all points, and will be interpreted as compressive. The first point should be (zero, initial collapse load).

The collapse load may vary with end moment as well as with deflections. In this case several load-deflection curves are defined, each corresponding to a different end moment. Each load curve should have the same number of points and the same deflection values. The end moment is defined as the average of the absolute moments at each end of the beam and is always positive.

Stiffness-proportional damping may be added using the damping factor λ. This is defined as follows:

$$\lambda = \frac{2 * \xi}{\omega}$$

where ξ is the damping factor at the reference frequency ω (in radians per second). For example if 1% damping at 2Hz is required

$$\lambda = \frac{2 * 0.01}{2\pi * 2} = 0.001592$$

If damping is used, a small timestep may be required. LS-DYNA does not check this so to avoid instability it may be necessary to control the timestep via a load curve. As a guide, the timestep required for any given element is multiplied by $0.3L/c\lambda$ when damping is present (L = element length, c = sound speed).

Moment Interaction:

Plastic hinges can form due to the combined action of moments about the three axes. This facility is activated only when yield moments are defined in the material input. A hinge forms when the following condition is first satisfied.

$$\left(\frac{M_r}{M_{ryield}}\right)^2 + \left(\frac{M_s}{M_{syield}}\right)^2 + \left(\frac{M_t}{M_{tyield}}\right)^2 \geq 1$$

where,

M_r, M_s, M_t = current moment

M_{ryield}, M_{syield}, M_{tyield} = yield moment

Note that scale factors for hinge behavior defined in the input will also be applied to the yield moments: for example, M_{syield} in the above formula is given by the input yield moment about the local axis times the input scale factor for the local s axis. For strain-softening characteristics, the yield moment should generally be set equal to the initial peak of the moment-rotation load curve.

On forming a hinge, upper limit moments are set. These are given by

$$M_{r_{upper}} = MAX\left(M_r, \frac{M_{r_{yield}}}{2}\right)$$

and similar for M_s and M_t.

Thereafter the plastic moments will be given by

M_{rp}, = min (M_{rupper}, M_{rcurve}) and similar for s and t

where

M_{rp} = current plastic moment

M_{rcurve} = moment taken from load curve at the current rotation scaled according to the scale factor.

The effect of this is to provide an upper limit to the moment that can be generated; it represents the softening effect of local buckling at a hinge site. Thus if a member is bent about is local s-axis it will then be weaker in torsion and about its local t-axis. For moments-softening curves,

the effect is to trim off the initial peak (although if the curves subsequently harden, the final hardening will also be trimmed off).

It is not possible to make the plastic moment vary with axial load.

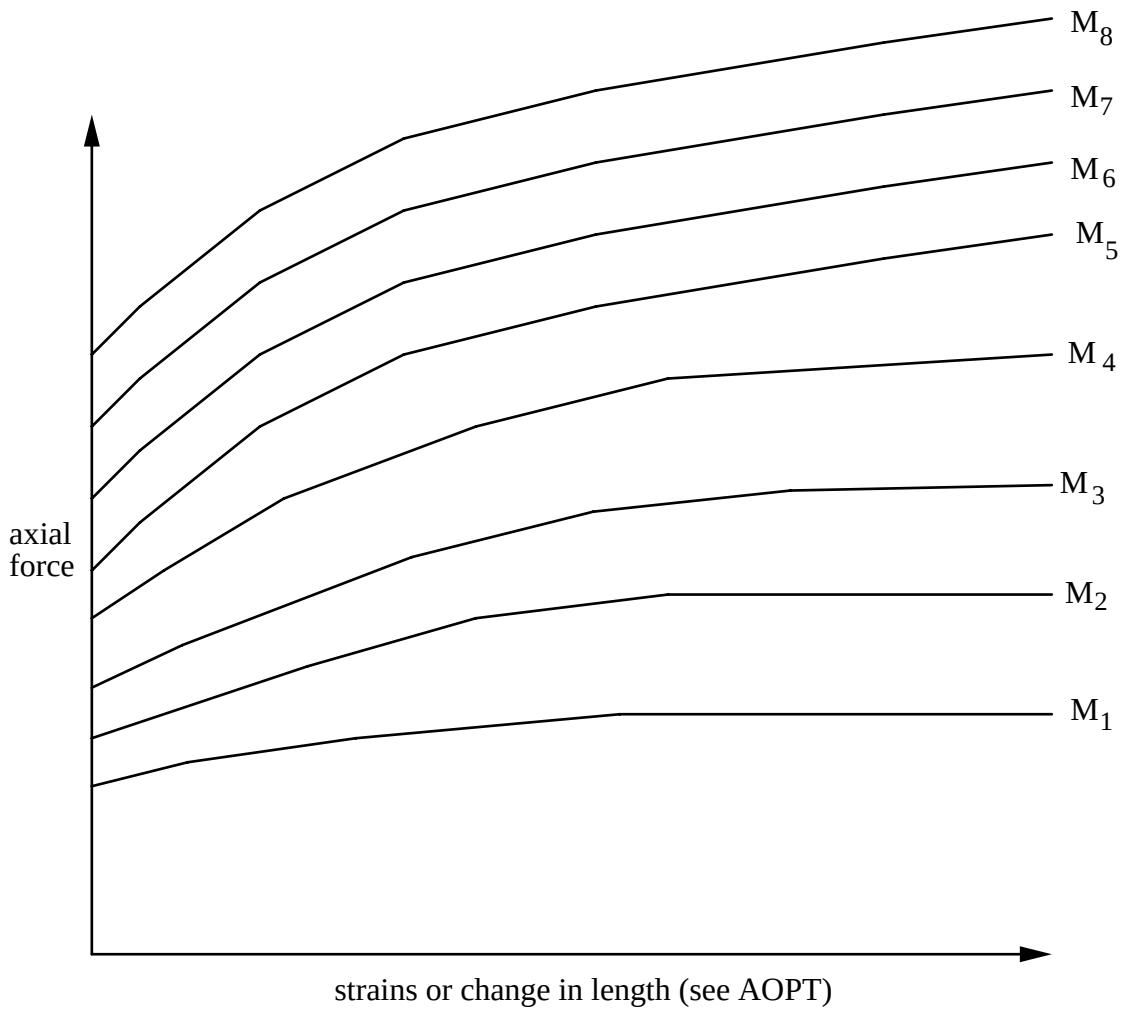


Figure 29.1. The force magnitude is limited by the applied end moment. For an intermediate value of the end moment LS-DYNA interpolates between the curves to determine the allowable force value.

*MAT_SHAPE_MEMORY

This is material type 30. This material model describes the superelastic response present in shape-memory alloys (SMA), that is the peculiar material ability to undergo large deformations with a full recovery in loading-unloading cycles (See Figure 30.1). The material response is always characterized by a hysteresis loop. See the references by Auricchio, Taylor and Lubliner [1997] and Auricchio and Taylor [1997]. This model is available for shell and solid elements. For Hughes-Liu beam elements it is available starting in Release 3 of version 971.

Card 1

	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR				
Type	A8	F	F	F				
Default	none	none	none	none				

Card 2

Variable	SIG_ASS	SIG_ASF	SIG_SAS	SIG_SAF	EPSL	ALPHA	YMRT	
Type	F	F	F	F	F	F	F	
Default	none	none	none	none	none	0.0	0.0	

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Density
E	Young's modulus
PR	Poisson's ratio
SIG_ASS	Starting value for the forward phase transformation (conversion of austenite into martensite) in the case of a uniaxial tensile state of stress. A load curve for SIG_ASS as a function of temperature is specified by using the negative of the load curve ID number.

VARIABLE	DESCRIPTION
SIG_ASF	Final value for the forward phase transformation (conversion of austenite into martensite) in the case of a uniaxial tensile state of stress. SIG_ASF as a function of temperature is specified by using the negative of the load curve ID number.
SIG_SAS	Starting value for the reverse phase transformation (conversion of martensite into austenite) in the case of a uniaxial tensile state of stress. SIG_SAS as a function of temperature is specified by using the negative of the load curve ID number.
SIG_SAF	Final value for the reverse phase transformation (conversion of martensite into austenite) in the case of a uniaxial tensile state of stress. SIG_SAF as a function of temperature is specified by using the negative of the load curve ID number.
EPSL	Recoverable strain or maximum residual strain. It is a measure of the maximum deformation obtainable all the martensite in one direction.
ALPHA	Parameter measuring the difference between material responses in tension and compression (set alpha = 0 for no difference). Also, see the following Remark.
YMRT	Young's modulus for the martensite if it is different from the modulus for the austenite. Defaults to the austenite modulus if it is set to zero.

Remarks:

The material parameter alpha, α, measures the difference between material responses in tension and compression. In particular, it is possible to relate the parameter α to the initial stress value of the austenite into martensite conversion, indicated respectively as $\sigma_s^{AS,+}$ and $\sigma_s^{AS,-}$, according to the following expression:

$$\alpha = \frac{\sigma_s^{AS,-} - \sigma_s^{AS,+}}{\sigma_s^{AS,-} + \sigma_s^{AS,+}}$$

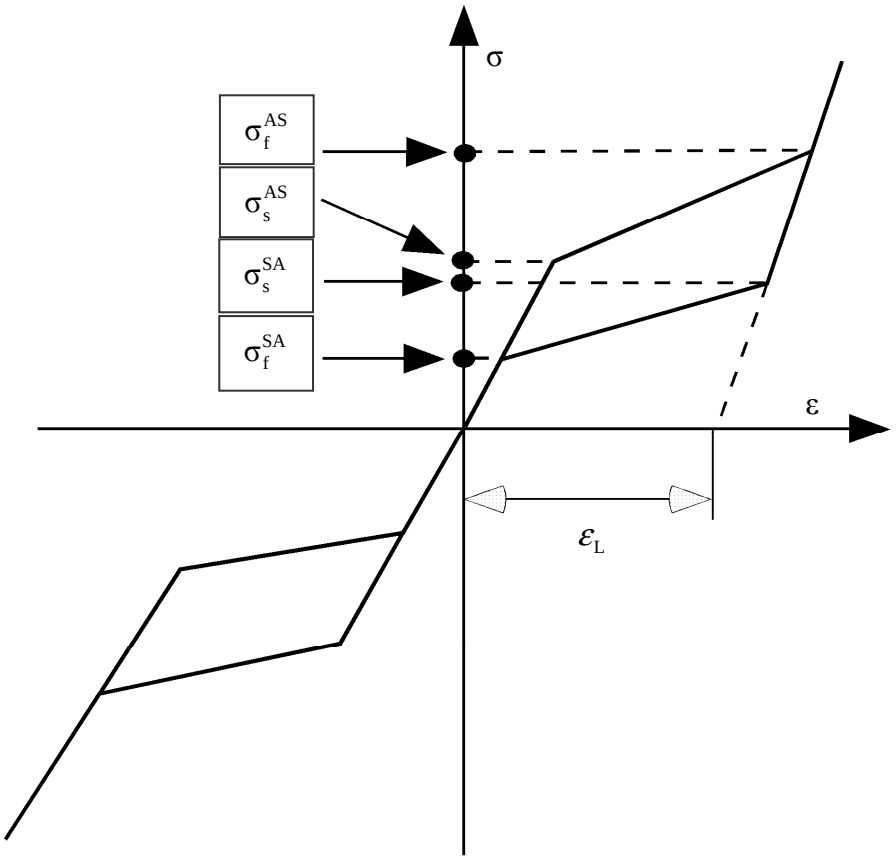


Figure 30.1. Pictorial representation of superelastic behavior for a shape-memory material.

In the following, the results obtained from a simple test problem is reported. The material properties are set as:

E	60000 MPa
Nu	0.3
sig_AS_s	520 MPa
sig_AS_f	600 MPa
sig_SA_s	300 MPa
sig_SA_f	200 MPa
epsL	0.07
alpha	0.12
ymrt	50000 MPa

The investigated problem is the complete loading-unloading test in tension and compression. The uniaxial Cauchy stress versus the logarithmic strain is plotted in Figure 30.2.

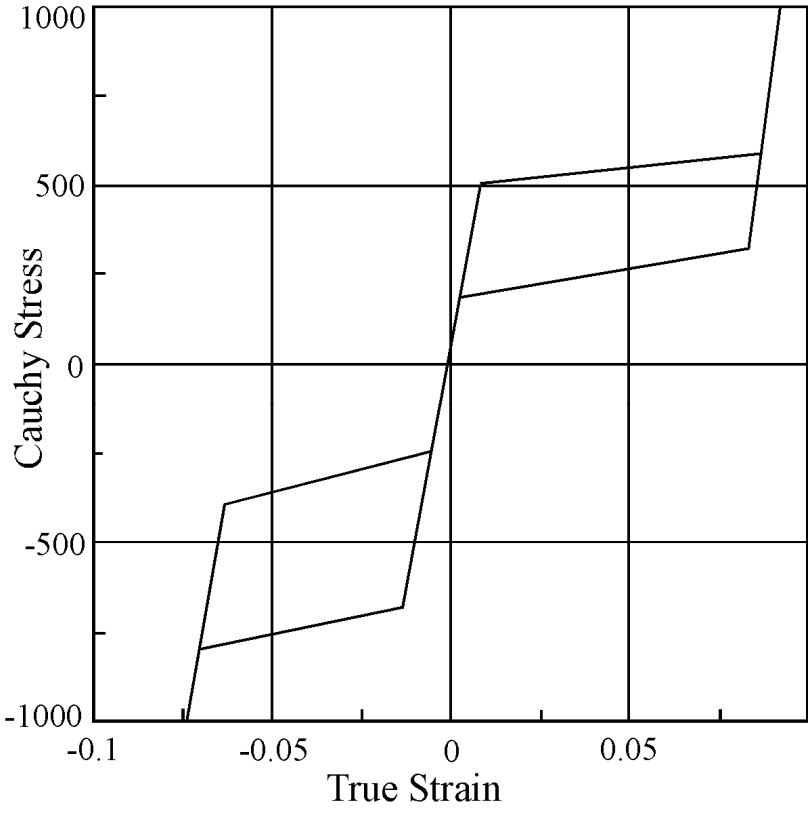


Figure 30.2. Complete loading-unloading test in tension and compression

*MAT_FRAZER_NASH_RUBBER_MODEL

This is Material Type 31. This model defines rubber from uniaxial test data. It is a modified form of the hyperelastic constitutive law first described in Kenchington [1988]. See also the notes below.

Card 1

1	2	3	4	5	6	7	8
Variable	MID	RO	PR	C100	C200	C300	C400	
Type	A8	F	F	F	F	F	F	

Card 2

Variable	C110	C210	C010	C020	EXIT	EMAX	EMIN	REF
Type	F	F	F	F	F	F	F	F

Card 3

Variable	SGL	SW	ST	LCID				
Type	F	F	F	F				

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
PR	Poisson's ratio. Values between .49 and .50 are suggested.
C100	C_{100} (EQ.1.0 if term is in the least squares fit.)
C200	C_{200} (EQ.1.0 if term is in the least squares fit.)
C300	C_{300} (EQ.1.0 if term is in the least squares fit.)
C400	C_{400} (EQ.1.0 if term is in the least squares fit.)

VARIABLE	DESCRIPTION
C110	C_{110} (EQ.1.0 if term is in the least squares fit.)
C210	C_{210} (EQ.1.0 if term is in the least squares fit.)
C010	C_{010} (EQ.1.0 if term is in the least squares fit.)
C020	C_{020} (EQ.1.0 if term is in the least squares fit.)
EXIT	Exit option: EQ. 0.0: stop if strain limits are exceeded (recommended), NE. 0.0: continue if strain limits are exceeded. The curve is then extrapolated.
EMAX	Maximum strain limit, (Green-St, Venant Strain).
EMIN	Minimum strain limit, (Green-St, Venant Strain).
REF	Use reference geometry to initialize the stress tensor. The reference geometry is defined by the keyword: *INITIAL_FOAM_ REFERENCE_GEOMETRY (see there for more details). EQ.0.0: off, EQ.1.0: on.
SGL	Specimen gauge length, see Figure 27.1.
SW	Specimen width, see Figure 27.1.
ST	Specimen thickness, see Figure 27.1.
LCID	Load curve ID, see DEFINE_CURVE, giving the force versus actual change in gauge length. See also Figure 27.2 for an alternative definition.

Remarks:

The constants can be defined directly or a least squares fit can be performed if the uniaxial data (SGL, SW, ST and LCID) is available. If a least squares fit is chosen, then the terms to be included in the energy functional are flagged by setting their corresponding coefficients to unity. If all coefficients are zero the default is to use only the terms involving I_1 and I_2. C_{100} defaults to unity if the least square fit is used.

The strain energy functional, U, is defined in terms of the input constants as:

$$U = C_{100}I_1 + C_{200}I_1^2 + C_{300}I_1^3 + C_{400}I_1^4 + C_{110}I_1I_2 + C_{210}I_1^2I_2 + C_{010}I_2 + C_{020}I_2^2 + f(J)$$

where the invariants can be expressed in terms of the deformation gradient matrix, F_{ij}, and the Green-St. Venant strain tensor, E_{ij} :

$$J = \left|F_{ij}\right|$$
$$I_1 = E_{ii}$$
$$I_2 = \frac{1}{2!}\delta^{ij}_{pq}E_{pi}E_{qj}$$

The derivative of U with respect to a component of strain gives the corresponding component of stress

$$S_{ij} = \frac{\partial U}{\partial E_{ij}}$$

here, S_{ij}, is the second Piola-Kirchhoff stress tensor.

The load curve definition that provides the uniaxial data should give the change in gauge length, ΔL, and the corresponding force. In compression both the force and the change in gauge length must be specified as negative values. In tension the force and change in gauge length should be input as positive values. The principal stretch ratio in the uniaxial direction, λ_1, is then given by

$$\lambda = \frac{L_o + \Delta L}{L_o}$$

Alternatively, the stress versus strain curve can also be input by setting the gauge length, thickness, and width to unity and defining the engineering strain in place of the change in gauge length and the nominal (engineering) stress in place of the force, see figure 27.2.

The least square fit to the experimental data is performed during the initialization phase and is a comparison between the fit and the actual input is provided in the printed file. It is a good idea to visually check the fit to make sure it is acceptable. The coefficients C_{100} - C_{020} are also printed in the output file.

*MAT_LAMINATED_GLASS

This is Material Type 32. With this material model, a layered glass including polymeric layers can be modeled. Failure of the glass part is possible. See notes below.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	EG	PRG	SYG	ETG	EFG	EP
Type	A8	F	F	F	F	F	F	F

Card 2

Variable	PRP	SYP	ETP					
Type	F	F	F					

Define 1-4 cards with a maximum of 32 number. If less than 4 cards are input, reading is stopped by a "*" control card.

Card 3...	1	2	3	4	5	6	7	8
Variable	F1	F2	F3	F4	F5	F6	F7	F8
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
EG	Young's modulus for glass
PRG	Poisson's ratio for glass
SYG	Yield stress for glass
ETG	Plastic hardening modulus for glass

VARIABLE	DESCRIPTION
EFG	Plastic strain at failure for glass
EP	Young's modulus for polymer
PRP	Poisson's ratio for polymer
SYP	Yield stress for polymer
ETP	Plastic hardening modulus for polymer
F1,..FN	Integration point material: f_n = 0.0: glass, f_n = 1.0: polymer. A user-defined integration rule must be specified, see *INTEGRATION_SHELL. See remarks below.

Remarks:

Isotropic hardening for both materials is assumed. The material to which the glass is bonded is assumed to stretch plastically without failure. A user defined integration rule specifies the thickness of the layers making up the glass. F_i defines whether the integration point is glass (0.0) or polymer (1.0). The material definition, F_i, has to be given for the same number of integration points (NIPTS) as specified in the rule. A maximum of 32 layers is allowed.

If the recommended user defined rule is not defined, the default integration rules are used. The location of the integration points in the default rules are defined in the *SECTION_ SHELL keyword description.

*MAT_BARLAT_ANISOTROPIC_PLASTICITY

This is Material Type 33. This model was developed by Barlat, Lege, and Brem [1991] for modeling anisotropic material behavior in forming processes. The finite element implementation of this model is described in detail by Chung and Shah [1992] and is used here. It is based on a six parameter model, which is ideally suited for 3D continuum problems, see notes below. For sheet forming problems, material 36 based on a 3-parameter model is recommended.

Card 1

Variable	MID	RO	E	PR	K	E0	N	M
Type	A8	F	F	F	F	F	F	F

Card 2

Variable	A	B	C	F	G	H	LCID	
Type	F	F	F	F	F	F	F	

Card 3

Variable	AOPT	OFFANG						
Type	F	F						

Card 4

Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card 5	1	2	3	4	5	6	7	8
Variable	V1	V2	V3	D1	D2	D3		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
E	Young's modulus, E.
PR	Poisson's ratio, ν.
K	k, strength coefficient, see notes below.
EO	ε_0, strain corresponding to the initial yield, see notes below.
N	n, hardening exponent for yield strength.
M	m, flow potential exponent in Barlat's Model.
A	a, anisotropy coefficient in Barlat's Model.
B	b, anisotropy coefficient in Barlat's Model.
C	c anisotropy coefficient in Barlat's Model.
F	f, anisotropy coefficient in Barlat's Model.
G	g, anisotropy coefficient in Barlat's Model.
H	h, anisotropy coefficient in Barlat's Model.
LCID	Option load curve ID defining effective stress versus effective plastic strain. If nonzero, this curve will be used to define the yield stress. The load curve is implemented for solid elements only.
AOPT	Material axes option:

VARIABLE	DESCRIPTION
	EQ.0.0: locally orthotropic with material axes determined by element nodes as shown in Figure 2.1. Nodes 1, 2, and 4 of an element are identical to the Nodes used for the definition of a coordinate system as by *DEFINE_COORDINATE_NODES. EQ.1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center, this is the a-direction. EQ.2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_ VECTOR. EQ.3.0: locally orthotropic material axes determined by offsetting the material axes by an angle, OFFANG, from a line determined by taking the cross product of the vector **v** with the normal to the plane of a shell element or midsurface of a brick. LT.0.0: the absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_NODES, *DEFINE_ COORDINATE_SYSTEM or *DEFINE_COORDINATE_ VECTOR). Available in R3 version of 971 and later.
BETA	Offset angle for AOPT = 3.
MACF	Material axes change flag for brick elements: EQ.1: No change, default, EQ.2: switch material axes a and b, EQ.3: switch material axes a and c, EQ.4: switch material axes b and c.
XP YP ZP	Coordinates of point **p** for AOPT = 1.
A1 A2 A3	Components of vector **a** for AOPT = 2.
V1 V2 V3	Components of vector **v** for AOPT = 3.
D1 D2 D3	Components of vector **d** for AOPT = 2.

Remarks:

The yield function Φ is defined as:

$$\Phi = |S_1 - S_2|^m + |S_2 - S_3|^m + |S_3 - S_1|^m = 2\bar{\sigma}^m$$

where $\bar{\sigma}$ is the effective stress and $S_{i=1,2,3}$ are the principal values of the symmetric matrix $S_{\alpha\beta}$,

$$S_{xx} = \left[c\left(\sigma_{xx} - \sigma_{yy}\right) - b\left(\sigma_{zz} - \sigma_{xx}\right) \right] / 3$$
$$S_{yy} = \left[a\left(\sigma_{yy} - \sigma_{zz}\right) - c\left(\sigma_{xx} - \sigma_{yy}\right) \right] / 3$$
$$S_{zz} = \left[b\left(\sigma_{zz} - \sigma_{xx}\right) - a\left(\sigma_{yy} - \sigma_{zz}\right) \right] / 3$$
$$S_{yz} = f\sigma_{yz}$$
$$S_{zx} = g\sigma_{zx}$$
$$S_{xy} = h\sigma_{xy}$$

The material constants *a*, *b*, *c*, *f*, *g* and *h* represent anisotropic properties. When $a = b = c = f = g = h = 1$, the material is isotropic and the yield surface reduces to the Tresca yield surface for *m*=1 and von Mises yield surface for *m*=2 or 4.

For face centered cubic (FCC) materials *m*=8 is recommended and for body centered cubic (BCC) materials *m*=6 *is* used. The yield strength of the material is

$$\sigma_y = k\left(\varepsilon^p + \varepsilon_0\right)^n$$

where ε_0 is the strain corresponding to the initial yield stress and ε^p is the plastic strain.

***MAT_BARLAT_YLD96**

This is Material Type 33. This model was developed by Barlat, Maeda, Chung, Yanagawa, Brem, Hayashida, Lege, Matsui, Murtha, Hattori, Becker, and Makosey [1997] for modeling anisotropic material behavior in forming processes in particular for aluminum alloys. This model is available for shell elements only.

Card 1

Variable	MID	RO	E	PR	K			
Type	A8	F	F	F	F			

Card 2

Variable	E0	N	ESR0	M	HARD	A		
Type	F	F	F	F	F	F		

Card 2

Variable	C1	C2	C3	C4	AX	AY	AZ0	AZ1
Type	F	F	F	F	F	F	F	F

Card 4

Variable	AOPT	OFFANG						
Type	F	F						

Card 5	1	2	3	4	5	6	7	8
Variable				A1	A2	A3		
Type				F	F	F		

Card 6

Variable	V1	V2	V3	D1	D2	D3		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
E	Young's modulus, E.
PR	Poisson's ratio,ν.
K	*k*, strength coefficient or *a* in Voce, see notes below.
EO	ε_0, strain corresponding to the initial yield or *b* in Voce, see notes below.
N	*n*, hardening exponent for yield strength or *c* in Voce.
ESR0	ε_{SR0}, in powerlaw rate sensitivity.
M	*m*, exponent for strain rate effects
HARD	Hardening option: LT. 0.0: absolute value defines the load curve ID. EQ. 1.0: powerlaw EQ. 2.0: Voce
A	Flow potential exponent.
C1	c1, see equations below.
C2	c2, see equations below.

VARIABLE	DESCRIPTION
C3	c3, see equations below.
C4	c4, see equations below.
AX	ax, see equations below.
AY	ay, see equations below.
AZ0	az0, see equations below.
AZ1	az1, see equations below.
AOPT	Material axes option: EQ.0.0: locally orthotropic with material axes determined by element nodes as shown in Figure 2.1. Nodes 1, 2, and 4 of an element are identical to the Nodes used for the definition of a coordinate system as by *DEFINE_COORDINATE_NODES. EQ.2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_ VECTOR. EQ.3.0: locally orthotropic material axes determined by offsetting the material axes by an angle, OFFANG, from a line determined by taking the cross product of the vector **v** with the normal to the plane of the element. LT.0.0: the absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_NODES, *DEFINE_ COORDINATE_SYSTEM or *DEFINE_COORDINATE_ VECTOR). Available in R3 version of 971 and later.
OFFANG	Offset angle for AOPT = 3.
A1 A2 A3	Components of vector **a** for AOPT = 2.
V1 V2 V3	Components of vector **v** for AOPT = 3.
D1 D2 D3	Components of vector **d** for AOPT = 2.

Remarks:

The yield stress σ_y is defined three ways. The first, the Swift equation, is given in terms of the input constants as:

$$\sigma_y = k\left(\varepsilon_0 + \varepsilon^p\right)^n \left(\frac{\dot{\varepsilon}}{\varepsilon_{SR0}}\right)^m$$

The second, the Voce equation, is defined as:

$$\sigma_y = a - be^{-c\varepsilon^p}$$

and the third option is to give a load curve ID that defines the yield stress as a function of effective plastic strain. The yield function Φ is defined as:

$$\Phi = \alpha_1 |s_1 - s_2|^a + \alpha_2 |s_2 - s_3|^a + \alpha_3 |s_3 - s_1|^a = 2\sigma_y^a$$

where s_i is a principle component of the deviatoric stress tensor where in vector notation:

$$\underset{\sim}{s} = \underset{\sim}{L}\underset{\sim}{\sigma}$$

and $\underset{\sim}{L}$ is given as

$$\underset{\sim}{L} = \begin{bmatrix} \frac{c_2 + c_3}{3} & \frac{-c_3}{3} & \frac{-c_2}{3} & 0 \\ \frac{-c_3}{3} & \frac{c_3 + c_1}{3} & \frac{-c_1}{3} & 0 \\ \frac{-c_2}{3} & \frac{-c_1}{3} & \frac{c_1 + c_2}{3} & 0 \\ 0 & 0 & 0 & c_4 \end{bmatrix}$$

A coordinate transformation relates the material frame to the principle directions of $\underset{\sim}{s}$ is used to obtain the α_k coefficients consistent with the rotated principle axes:

$$\alpha_k = \alpha_x p_{1k}^2 + \alpha_y p_{2k}^2 + \alpha_z p_{3k}^2$$

$$\alpha_z = \alpha_{z0} \cos^2 2\beta + \alpha_{z1} \sin^2 2\beta$$

where p_{ij} are components of the transformation matrix. The angle β defines a measure of the rotation between the frame of the principal value of $\underset{\sim}{s}$ and the principal anisotropy axes.

***MAT_FABRIC**

This is Material Type 34. This material is especially developed for airbag materials. The fabric model is a variation on the layered orthotropic composite model of material 22 and is valid for 3 and 4 node membrane elements only. In addition to being a constitutive model, this model also invokes a special membrane element formulation which is more suited to the deformation experienced by fabrics under large deformation. For thin fabrics, buckling can result in an inability to support compressive stresses; thus a flag is included for this option. A linearly elastic liner is also included which can be used to reduce the tendency for these elements to be crushed when the no-compression option is invoked. In LS-DYNA versions after 931 the isotropic elastic option is available.

Card 1

1	2	3	4	5	6	7	8
MID	RO	EA	EB	EC	PRBA	PRCA	PRCB
A8	F	F	F	F	F	F	F

Card 2

GAB	GBC	GCA	CSE	EL	PRL	LRATIO	DAMP
F	F	F	F	F	F	F	F
			1	2	2	2	

Card 3

AOPT	FLC/X2	FAC/X3	ELA	LNRC	FORM	FVOPT	TSRFAC
F	F	F	F	F	F	F	F
	3	3		4	0	0	9

Card 4	1	2	3	4	5	6	7	8
Variable				A1	A2	A3	X0	X1
Type				F	F	F	F	F

Card 5

Variable	V1	V2	V3	D1	D2	D3	BETA	ISREFG
Type	F	F	F	F	F	F	F	I

Define if and only if FORM=4 or 14.

Card 6	1	2	3	4	5	6	7	8
Variable	LCA	LCB	LCAB	LCUA	LCUB	LCUAB	RL	
Type	I	I	I	I	I	I	F	

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
EA	Young's modulus - longitudinal direction. For an isotopic elastic fabric material only EA and PRBA are defined and are used as the isotropic Young's modulus and Poisson's ratio, respectively. The input for the fiber directions and liner should be input as zero for the isotropic elastic fabric.
EB	Young's modulus - transverse direction, set to zero for isotropic elastic material.
(EC)	Young's modulus - normal direction, set to zero for isotropic elastic material. (Not used)

VARIABLE	DESCRIPTION
PRBA	ν_{ba}, Poisson's ratio ba direction.
(PRCA)	ν_{ca}, Poisson's ratio ca direction, set to zero for isotropic elastic material. (Not used)
(PRCB)	ν_{cb}, Poisson's ratio cb direction, set to zero for isotropic elastic material. (Not used)
GAB	G_{ab}, shear modulus ab direction, set to zero for isotropic elastic material.
(GBC)	G_{bc}, shear modulus bc direction, set to zero for isotropic elastic material. (Not used)
(GCA)	G_{ca}, shear modulus ca direction, set to zero for isotropic elastic material. (Not used)
CSE	Compressive stress elimination option (default 0.0): EQ.0.0: don't eliminate compressive stresses, EQ.1.0: eliminate compressive stresses (This option does not apply to the liner).
EL	Young's modulus for elastic liner (optional).
PRL	Poisson's ratio for elastic liner (optional).
LRATIO	Ratio of liner thickness to total fabric thickness.
DAMP	Rayleigh damping coefficient. A 0.05 coefficient is recommended corresponding to 5% of critical damping. Sometimes larger values are necessary.
AOPT	Material axes option (see MAT_*OPTION* TROPIC_ELASTIC for a more complete description): EQ.0.0: locally orthotropic with material axes determined by element nodes 1, 2, and 4, as with *DEFINE_COORDINATE_NODES. EQ.2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR. EQ.3.0: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle, BETA, from a line in the plane of the element defined by the cross product of the vector **v** with the element normal. LT.0.0: the absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_NODES, *DEFINE_COORDINATE_SYSTEM or *DEFINE_COORDINATE_VECTOR). Available in R3 version of 971 and later.

VARIABLE	DESCRIPTION
FLC/X2	If $X0 \neq 0, X0 \neq 1$: This is X2 coefficient of the porosity equation of Anagonye and Wang [1999]. Else, this is an optional constant, FLC, a fabric porous leakage flow coefficient. LT.0.0: There are two possible definitions. If $X0 = 0$, |FLC| is the load curve ID of the curve defining FLC versus time. If $X0 = 1$, |FLC| is the load curve ID defining FLC versus the stretching ratio defined as $r_s = A/A_0$. See notes below.
FAC/X3	If $X0 \neq 0, X0 \neq 1$: This is X3 coefficient of the porosity equation of Anagonye and Wang [1999]. Else, if and only if $X0 = 0$: This is an optional constant, FAC, a fabric characteristic parameter. LT.0.0: There are three possible definitions. If FVOPT < 7: If $X0 = 0$, |FAC| is the load curve ID of the curve defining FAC versus absolute pressure. If $X0 = 1$, |FAC| is the load curve ID defining FAC versus the pressure ratio defined as $r_p = P_{air}/P_{bag}$. See remark 3 below. If FVOPT = 7 or 8: FAC defines leakage volume flux rate versus absolute pressure. The volume flux (per area) rate (per time) has the unit of $\dot{vol}_{flux} \approx m^3/\left[m^2 s\right] \approx m/s$, equivalent to relative porous gas speed.
ELA	Effective leakage area for blocked fabric, ELA. LT.0.0: |ELA| is the load curve ID of the curve defining ELA versus time. The default value of zero assumes that no leakage occurs. A value of .10 would assume that 10% of the blocked fabric is leaking gas.
LNRC	Flag to turn off compression in liner until the reference geometry is reached, i.e., the fabric element becomes tensile. EQ.0.0: off. EQ.1.0: on.
FORM	Flag to modify membrane formulation for fabric material: EQ.0.0: default. Least costly and very reliable. EQ.1.0: invariant local membrane coordinate system EQ.2.0: Green-Lagrange strain formulation EQ.3.0: large strain with nonorthogonal material angles. See Remark 5. EQ.4.0: large strain with nonorthogonal material angles and nonlinear stress strain behavior. Define optional load curve IDs on optional card.

VARIABLE	DESCRIPTION
	EQ.12.0: Updated form 2. See Remark 10. EQ.13.0: Updated form 3. See Remark 10. EQ.14.0: Updated form 4. See Remark 10.
FVOPT	Fabric venting option. EQ.1: Wang-Nefske formulas for venting through an orifice are used. Blockage is not considered. EQ.2: Wang-Nefske formulas for venting through an orifice are used. Blockage of venting area due to contact is considered. EQ.3: Leakage formulas of Graefe, Krummheuer, and Siejak [1990] are used. Blockage is not considered. EQ.4: Leakage formulas of Graefe, Krummheuer, and Siejak [1990] are used. Blockage of venting area due to contact is considered. EQ.5: Leakage formulas based on flow through a porous media are used. Blockage is not considered. EQ.6: Leakage formulas based on flow through a porous media are used. Blockage of venting area due to contact is considered. EQ.7: Leakage is based on gas volume outflow versus pressure load curve [Lian, 2000]. Blockage is not considered. Absolute pressure is used in the porous-velocity-versus-pressure load curve, given as FAC in the *MAT_FABRIC card. EQ.8: Leakage is based on gas volume outflow versus pressure load curve [Lian 2000]. Blockage of venting or porous area due to contact is considered. Absolute pressure is used in the porous-velocity-versus-pressure load curve, given as FAC in the *MAT_FABRIC card.
TSRFAC	Tensile stress cutoff reduction factor LT.0: |TSRFAC| is the curve ID of the curve defining TSRFAC versus time.' GT.0 and LT.1: TSRFAC applied from time 0. GE.1: TSRFAC is a curve ID for the new option.
A1 A2 A3	Components of vector **a** for AOPT = 2.
X0,X1	Coefficients of Anagonye and Wang [1999] porosity equation for the leakage area: $A_{leak} = A_0\left(X_0 + X_1 r_s + X_2 r_p + X_3 r_s r_p\right)$
V1 V2 V3	Components of vector **v** for AOPT = 3.
D1 D2 D3	Components of vector **d** for AOPT = 2.
BETA	Material angle in degrees for AOPT = 3, may be overridden on the element card, see *ELEMENT_SHELL_BETA.
ISREFG	Initial stress by reference geometry for FORM=12

EQ.0.0: default. Not active.
EQ.1.0: active

VARIABLE	DESCRIPTION
LCA	Load curve ID for stress versus strain along the a-axis fiber; available for FORM=4 or 14 only. If zero, EA is used.
LCB	Load curve ID for stress versus strain along the b-axis fiber; available for FORM=4 or 14 only. If zero, EB is used.
LCAB	Load curve ID for shear stress versus shear strain in the ab-plane; available for FORM=4 or 14 only. If zero, GAB is used.
LCUA	Unload/reload curve ID for stress versus strain along the a-axis fiber; available for FORM=4 or 14 only. If zero, LCA is used.
LCUB	Unload/reload curve ID for stress versus strain along the b-axis fiber; available for FORM=4 or 14 only. If zero, LCB is used.
LCUAB	Unload/reload curve ID for shear stress versus shear strain in the ab-plane; available for FORM=4 or 14 only. If zero, LCAB is used.
RL	Optional reloading parameter for FORM=14. Values between 0.0 (reloading on unloading curve-default) and 1.0 (reloading on a minimum linear slope between unloading curve and loading curve) are possible.

Remarks:

1. The no compression option allows the simulation of airbag inflation with far less elements than would be needed for the discretization of the wrinkles which would occur for the case when compressive stresses are not eliminated.

2. When using this material for the analysis of membranes as airbags it is well known from classical theory that only one layer has to be defined. The so-called elastic liner has to be defined for numerical purposes only when the no compression option is invoked.

3. The parameters FLC and FAC are optional for the Wang-Nefske inflation models. It is possible for the airbag to be constructed of multiple fabrics having different values for porosity and permeability. The leakage of gas through the fabric in an airbag then requires an accurate determination of the areas by part ID available for leakage. The leakage area may change over time due to stretching of the airbag fabric or blockage when the bag contacts the structure. LS-DYNA can check the interaction of the bag with the structure and split the areas into regions that are blocked and unblocked depending on whether the regions are in or not in contact, respectively. Typically, FLC and FAC must be determined experimentally and their variations in time or with pressure are optional to allow for maximum flexibility.

4. The elastic backing layer always acts in tension and compression since the tension cutoff option, CSE, does not apply. This can sometimes cause difficulties if the elements are very small in relationship to their actual size as defined by the reference geometry (See

*AIRBAG_REFERENCE_GEOMETRY.). If the flag, LNRC, is set to 1.0 the elastic liner does not begin to act until the area of defined by the reference geometry is reached.

5. For FORM=0, 1, and 2, the a-axis and b-axis fiber directions are assumed to be orthogonal and are completely defined by the material axes option, AOPT=0, 2, or 3. For FORM=3, 4, 13, or 14, the fiber directions are not assumed orthogonal and must be specified using the ICOMP=1 option on *SECTION_SHELL. Offset angles should be input into the B1 and B2 fields used normally for integration points 1 and 2. The a-axis and b-axis directions will then be offset from the a-axis direction as determined by the material axis option, AOPT=0, 2, or 3.

6. For FORM=4 or 14, 2nd Piola-Kirchoff stress vs. Green's strain curves may be defined for a-axis, b-axis, and shear stresses for loading and also for unloading and reloading. All curves should start at the origin and be defined for positive strains only. The a-axis and b-axis stress follows the curves for tension only. For compression, stress is calculated from the constant values, EA or EB. Shear stress/strain behavior is assumed symmetric. If a load curve is omitted, the stress is calculated from the appropriate constant modulus, EA, EB, or GAB.

7. When both loading and unloading curves are defined, the initial yield strain is assumed to be equal to the strain at the first point in the load curve with stress greater than zero. When strain exceeds the yield strain, the stress continues to follow the load curve and the yield strain is updated to the current strain. When unloading occurs, the unload/reload curve is shifted along the x-axis until it intersects the load curve at the current yield strain. If the curve shift is to the right, unloading and reloading will follow the shifted unload/reload curve. If the curve shift is zero or to the left, unloading and reloading will occur along the load curve. When using unloading curves, compressive stress elimination should be active to prevent the fibers from developing compressive stress during unloading when the strain remains tensile.

   If LCUA, LCUB, or LCUAB are input with negative values, then unloading is handled differently. Instead of shifting the unload curve along the x-axis, the curve is stretched in the x-direction such that the first point remains at (0,0) and the unload curve intersects with the load curve at the current yield point. This option guarantees the stress remains tensile while the strain is tensile so compressive stress elimination is not necessary. To use this option the unload curve should have an initial slope less steep than the load curve, and should steepen such that it intersects the load curve at some positive strain value.

8. The FVOPT flag allows an airbag fabric venting equation to be assigned to an material. The anticipated use for this option is to allow a vent to be defined using FVOPT=1 or 2 for one material and fabric leakage to be defined for using FVOPT=3, 4, 5, or 6 for other materials. In order to use FVOPT, a venting option must first be defined for the airbag using the OPT parameter on *AIRBAG_WANG_NEFSKE or *AIRBAG_HYBRID. If OPT=0, then FVOPT is ignored. If OPT is defined and FVOPT is omitted, then FVOPT is set equal to OPT.

9. The TSRFAC factor is used to assure that airbags that have a reference geometry will open to the correct geometry. Airbags that use a reference geometry might have an initial geometry that results in initial strains. To prevent such strains from prematurely opening an airbag, these strains are eliminated by default. A side effect of this behavior is that airbags that use a reference geometry and that are initially stretched will never achieve the correct shape. The TSRFAC factor is used to restore the tensile strains over time such that the correct geometry is achieved. It is recommended that a load curve be used to define TSRFAC as function of time. Initially the load curve ordinate value should be 0.0 which will allow the bag to remain unstressed. At a time when the bag is partially open, the value of TSRFAC should ramp up to a small number of about 0.0001. Each cycle, the stored initial strains are scaled by (1.0-TSRFAC) such that they reduce to a very small number.

   A new option is invoked by setting TSRFAC≥1 in which case TSRFAC is a curve ID. The curve should ramp from 0.0 to 1.0. When the curve ordinate value is 0.0, the stored initial strain is subtracted from the total strain. For values between 0.0 and 1.0, a fraction of the stored initial strain is subtracted from the total strain where the fraction is 1.0-TSRFAC. When the curve value reaches or exceeds 1.0, the total strain is used. This option gives the user better control of the rate of restoring the strains as it is independent of the solution time step.

10. Material forms 12, 13, and 14 are updated versions of forms 2, 3, and 4, respectively. These new forms are intended to be less susceptible to timestep collapse and also guarantee zero stress in the initial geometry when a reference geometry is used. The behavior should otherwise be similar with one exception. The LNRC flag eliminates not only initial compressive strain but total initial strain. Therefore, the TSRFAC option is recommended (see Remark 9) when forms 12, 13, and 14 are used with a reference geometry and LNRC=1.

11 An option to calculate the initial stress by using a reference geometry is available for material FORM 12 only.

*MAT_PLASTIC_GREEN-NAGHDI_RATE

This is Material Type 35. This model is available only for brick elements and is similar to model 3, but uses the Green-Naghdi Rate formulation rather than the Jaumann rate for the stress update. For some cases this might be helpful. This model also has a strain rate dependency following the Cowper-Symonds model.

Card 1

	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR				
Type	A8	F	F	F				

Card 2

Variable	SIGY	ETAN	SRC	SRP	BETA			
Type	F	F	F	F	F			

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Density
E	Young's modulus
PR	Poisson's ratio
SIGY	Yield stress
ETAN	Plastic hardening modulus
SRC	Strain rate parameter, C
SRP	Strain rate parameter, P
BETA	Hardening parameter, $0 < \beta' < 1$

*MAT_3-PARAMETER_BARLAT

This is Material Type 36. This model was developed by Barlat and Lian [1989] for modeling sheets with anisotropic materials under plane stress conditions. This material allows the use of the Lankford parameters for the definition of the anisotropy. This particular development is due to Barlat and Lian [1989]. A version of this material model which has a flow limit diagram failure option is *MAT_FLD_3-PARAMETER_BARLAT.

Card 1

Variable	MID	RO	E	PR	HR	P1	P2	ITER
Type	A8	F	F	F	F	F	F	F

Card 2

Variable	M	R00 / AB	R45 / CB	R90 / HB	LCID	E0	SPI	P3
Type	F	F	F	F	I	F	F	F

Define the following card if and only if M<0

Card opt.

Variable	CRC1	CRA1	CRC2	CRA2	CRC3	CRA3	CRC4	CRA4
Type	F	F	F	F	F	F	F	F

Card 3

Variable	AOPT	C	P	VLCID		PB		
Type	F	F	F	I		F		

Card 4

Variable				A1	A2	A3		
Type				F	F	F		

Card 5	1	2	3	4	5	6	7	8
Variable	V1	V2	V3	D1	D2	D3	BETA	
Type	F	F	F	F	F	F	F	

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
E	Young's modulus, E GT.0.0: Constant value, LT.0.0: Load curve ID = (-E) which defines Young's Modulus as a function of plastic strain. See Remark 1.
PR	Poisson's ratio, ν
HR	Hardening rule: EQ.1.0: linear (default), EQ.2.0: exponential (Swift) EQ.3.0: load curve or table with strain rate effects EQ.4.0: exponential (Voce) EQ.5.0: exponential (Gosh) EQ.6.0: exponential (Hocket-Sherby) EQ.7.0: load curves in three directions EQ.8.0: table with temperature dependence
P1	Material parameter: HR.EQ.1.0: Tangent modulus, HR.EQ.2.0: k, strength coefficient for Swift exponential hardening HR.EQ.4.0: a, coefficient for Voce exponential hardening HR.EQ.5.0: k, strength coefficient for Gosh exponential hardening HR.EQ.6.0: a, coefficient for Hocket-Sherby exponential hardening

HR.EQ.7.0: load curve ID for hardening in 45 degree direction.
See Remark 2.

P2 Material parameter:
HR.EQ.1.0: Yield stress
HR.EQ.2.0: n, exponent for Swift exponential hardening
HR.EQ.4.0: c, coefficient for Voce exponential hardening
HR.EQ.5.0: n, exponent for Gosh exponential hardening
HR.EQ.6.0: c. coefficient for Hocket-Sherby exponential hardening
HR.EQ.7.0: load curve ID for hardening in 90 degree direction.
See Remark 2.

VARIABLE	DESCRIPTION
ITER	Iteration flag for speed: ITER.EQ.0.0: fully iterative ITER.EQ.1.0: fixed at three iterations Generally, ITER=0 is recommended. However, ITER=1 is somewhat faster and may give acceptable results in most problems.
M	m, exponent in Barlat's yield surface, absolute value is used if negative.
CRCN	Chaboche-Roussiler hardening parameter, see remarks.
CRCA	Chaboche-Roussiler hardening parameter, see remarks.
R00	R_{00}, Lankford parameter in 0 degree direction GT.0.0: Constant value, LT.0.0: Load curve ID = (-R00) which defines R value as a function of plastic strain. See Remark 3.
R45	R_{45}, Lankford parameter in 45 degree direction GT.0.0: Constant value, LT.0.0: Load curve ID = (-R45) which defines R value as a function of plastic strain. See Remarks 2 and 3.
R90	R_{90}, Lankford parameter in 90 degree direction GT.0.0: Constant value, LT.0.0: Load curve ID = (-R90) which defines R value as a function of plastic strain. See Remarks 2 and 3.
AB	a, Barlat89 parameter, which is read instead of R00 if PB>0.
CB	c, Barlat89 parameter, which is read instead of R45 if PB>0.
HB	h, Barlat89 parameter, which is read instead of R90 if PB>0.
LCID	Load curve/table ID for hardening in the 0 degree direction. See Remark 1.
E0	Material parameter HR.EQ.2.0: ε_0 for determining initial yield stress for Swift exponential hardening. (Default=0.0) HR.EQ.4.0: b, coefficient for Voce exponential hardening HR.EQ.5.0: ε_0 for determining initial yield stress for Gosh exponential hardening. (Default=0.0) HR.EQ.6.0: b, coefficient for Hocket-Sherby exponential hardening

VARIABLE	DESCRIPTION
SPI	*spi*, if ε_0 is zero above and HR.EQ.2.0. (Default=0.0) EQ.0.0: $\varepsilon_0 = (E/k)^{**}[1/(n-1)]$ LE.0.02: $\varepsilon_0 = spi$ GT.0.02: $\varepsilon_0 = (spi/k)^{**}[1/n]$ If HR.EQ.5.0 the strain at plastic yield is determined by an iterative procedure based on the same principles as for HR.EQ.2.0.
P3	Material parameter: HR.EQ.5.0: p, parameter for Gosh exponential hardening HR.EQ.6.0: n, exponent for Hocket-Sherby exponential hardening
AOPT	Material axes option (see MAT_*OPTION* TROPIC_ELASTIC for a more complete description): EQ.0.0: locally orthotropic with material axes determined by element nodes 1, 2, and 4, as with *DEFINE_COORDINATE_NODES. EQ.2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR. EQ.3.0: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle, BETA, from a line in the plane of the element defined by the cross product of the vector v with the element normal. LT.0.0: the absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_NODES, *DEFINE_COORDINATE_SYSTEM or *DEFINE_COORDINATE_VECTOR). Available with the R3 release of Version 971 and later.
C	C in Cowper-Symonds strain rate model
P	p in Cowper-Symonds strain rate model, p=0.0 for no strain rate effects
VLCID	Volume correction curve ID defining the relative volume change (change in volume relative to the initial volume) as a function of the effective plastic strain. This is only used when nonzero. See Remark 1.
PB	Barlat89 parameter, p. If PB>0, parameters AB, CB, and HB are read instead of R00, R45, and R90. See Remark 4.
XP YP ZP	Coordinates of point **p** for AOPT = 1.
A1 A2 A3	Components of vector **a** for AOPT = 2.
V1 V2 V3	Components of vector **v** for AOPT = 3.

VARIABLE	DESCRIPTION
D1 D2 D3	Components of vector **d** for AOPT = 2.
BETA	Material angle in degrees for AOPT=3, may be overridden on the element card, see *ELEMENT_SHELL_BETA.

Remarks:

1. The effective plastic strain used in this model is defined to be plastic work equivalent. A consequence of this is that for parameters defined as functions of effective plastic strain, the rolling (00) direction should be used as reference direction. For instance, the hardening curve for HR=3 is the stress as function of strain for uniaxial tension in the rolling direction, VLCID curve should give the relative volume change as function of strain for uniaxial tension in the rolling direction and load curve given by -E should give the Young's modulus as function of strain for uniaxial tension in the rolling direction. Optionally the curve can be substituted for a table defining hardening as function of plastic strain rate (HR=3) or temperature (HR=8).

2. Exceptions from the rule above are curves defined as functions of plastic strain in the 45 and 90 directions, i.e., P1 and P2 for HR=7 and negative R45 or R90. The hardening curves are here defined as measured stress as function of measured plastic strain for uniaxial tension in the direction of interest, i.e., as determined from experimental testing using a standard procedure. Moreover, the curves defining the R values are as function of the measured plastic strain for uniaxial tension in the direction of interest. These curves are transformed internally to be used with the effective stress and strain properties in the actual model. The effective plastic strain does not coincide with the plastic strain components in other directions than the rolling direction and may be somewhat confusing to the user. Therefore the von Mises work equivalent plastic strain is output as history variable #2 if HR=7 or if any of the R-values is defined as function of the plastic strain.

3. The R-values in curves are defined as the ratio of instantaneous width change to instantaneous thickness change. That is, assume that the width W and thickness T are measured as function of strain. Then the corresponding R-value is given by:

$$R = \frac{\frac{dW}{d\varepsilon}/W}{\frac{dT}{d\varepsilon}/T}$$

4. The anisotropic yield criterion Φ for plane stress is defined as:

$$\Phi = a\left|K_1 + K_2\right|^m + a\left|K_1 - K_2\right|^m + c\left|2K_2\right|^m = 2\sigma_Y^m$$

where σ_Y is the yield stress and $K_{i=1,2}$ are given by:

$$K_1 = \frac{\sigma_x + h\sigma_y}{2}$$

$$K_2 = \sqrt{\left(\frac{\sigma_x - h\sigma_y}{2}\right)^2 + p^2\tau_{xy}^2}$$

If PB=0, the anisotropic material constants a, c, h, and p are obtained through R_{00}, R_{45}, and R_{90}:

$$a = 2 - 2\sqrt{\frac{R_{00}}{1+R_{00}}\frac{R_{90}}{1+R_{90}}} \qquad c = 2 - a$$

$$h = \sqrt{\frac{R_{00}}{1+R_{00}}\frac{1+R_{90}}{R_{90}}}$$

The anisotropy parameter p is calculated implicitly. According to Barlat and Lian the R value, width to thickness strain ratio, for any angle ϕ can be calculated from:

$$R_\phi = \frac{2m\sigma_Y^m}{\left(\frac{\partial\Phi}{\partial\sigma_x} + \frac{\partial\Phi}{\partial\sigma_y}\right)\sigma_\phi} - 1$$

where σ_ϕ is the uniaxial tension in the ϕ direction. This expression can be used to iteratively calculate the value of p. Let ϕ=45 and define a function g as

$$g(p) = \frac{2m\sigma_Y^m}{\left(\frac{\partial\Phi}{\partial\sigma_x} + \frac{\partial\Phi}{\partial\sigma_y}\right)\sigma_\phi} - 1 - R_{45}$$

An iterative search is used to find the value of p.

If PB>0, material parameters a (AB), c (CB), h (HB), and p (PB) are used directly.

For face centered cubic (FCC) materials m=8 is recommended and for body centered cubic (BCC) materials m=6 may be used. The yield strength of the material can be expressed in terms of k and n:

$$\sigma_y = k\ \varepsilon^n = k\left(\varepsilon_{yp} + \bar{\varepsilon}^p\right)^n$$

where ε_{yp} is the elastic strain to yield and $\bar{\varepsilon}^p$ is the effective plastic strain (logarithmic). If SIGY is set to zero, the strain to yield if found by solving for the intersection of the linearly elastic loading equation with the strain hardening equation:

$$\sigma = E\ \varepsilon$$
$$\sigma = k\ \varepsilon^n$$

which gives the elastic strain at yield as:

$$\varepsilon_{yp} = \left(\frac{E}{k}\right)^{\left[\frac{1}{n-1}\right]}$$

If SIGY yield is nonzero and greater than 0.02 then:

$$\varepsilon_{yp} = \left(\frac{\sigma_y}{k}\right)^{\left[\frac{1}{n}\right]}$$

The other available hardening models include the Voce equation given by

$$\sigma_{\mathrm{Y}}(\varepsilon_p) = a - be^{-c\varepsilon_p},$$

the Gosh equation given by

$$\sigma_{\mathrm{Y}}(\varepsilon_p) = k(\varepsilon_0 + \varepsilon_p)^n - p,$$

and finally the Hocket-Sherby equation given by

$$\sigma_{\mathrm{Y}}(\varepsilon_p) = a - be^{-c\varepsilon_p^n}.$$

For the Gosh hardening law, the interpretation of the variable SPI is the same, i.e., if set to zero the strain at yield is determined implicitly from the intersection of the strain hardening equation with the linear elastic equation.

To include strain rate effects in the model we multiply the yield stress by a factor depending on the effective plastic strain rate. We use the Cowper-Symonds' model, hence the yield stress can be written

$$\sigma_{\mathrm{Y}}(\varepsilon_p, \dot{\varepsilon}_p) = \sigma_{\mathrm{Y}}^s(\varepsilon_p)\left\{1 + \left(\frac{\dot{\varepsilon}_p}{C}\right)^{1/p}\right\}$$

where σ_{Y}^s denotes the static yield stress, C and p are material parameters, $\dot{\varepsilon}_p$ is the effective plastic strain rate.

5. A kinematic hardening model is implemented following the works of Chaboche and Roussilier. A back stress $\boldsymbol{\alpha}$ is introduced such that the effective stress is computed as

$$\sigma_{\mathrm{eff}} = \sigma_{\mathrm{eff}}(\sigma_{11} - 2\alpha_{11} - \alpha_{22}, \sigma_{22} - 2\alpha_{22} - \alpha_{11}, \sigma_{12} - \alpha_{12})$$

The back stress is the sum of up to four terms according to

$$\alpha_{ij} = \sum_{k=1}^{4} \alpha_{ij}^{k}$$

and the evolution of each back stress component is as follows

$$\delta\alpha_{ij}^{k} = C_k \left(a_k \frac{s_{ij}}{\sigma_{\text{eff}}} - \alpha_{ij}^{k} \right) \delta\varepsilon_p$$

where C_k and a_k are material parameters, s_{ij} is the deviatoric stress tensor, σ_{eff} is the effective stress and ε_p is the effective plastic strain.

***MAT_TRANSVERSELY_ANISOTROPIC_ELASTIC_PLASTIC_**{*OPTION*}

Available option allows the change of Young's Modulus during the simulation:

<BLANK>

ECHANGE

This is Material Type 37. This model is for simulating sheet forming processes with anisotropic material. Only transverse anisotropy can be considered. Optionally an arbitrary dependency of stress and effective plastic strain can be defined via a load curve. This plasticity model is fully iterative and is available only for shell elements. Also see the notes below.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	SIGY	ETAN	R	HLCID
Type	A8	F	F	F	F	F	F	F

Define the following card if and only if option ECHANGE is used

Optional	1	2	3	4	5	6	7	8
Variable	IDSCALE	EA	COE					
Type	I	F	F					

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.
SIGY	Yield stress.
ETAN	Plastic hardening modulus.
R	Anisotropic hardening parameter.

VARIABLE	DESCRIPTION
HLCID	Load curve ID defining effective yield stress versus effective plastic strain.
IDSCALE	oad curve ID defining the scale factor for the Young's modulus change with respect to effective strain (if EA and COE are defined), this curve is not necessary).
EA, COE	coefficients defining the Young's modulus with respect to the effective strain, EA is E^A and Coe is ζ (if IDSCALE is defined, these two parameters are not necessary).

Remarks:

Consider Cartesian reference axes which are parallel to the three symmetry planes of anisotropic behavior. Then, the yield function suggested by Hill [1948] can be written

$$F\left(\sigma_{22}-\sigma_{33}\right)^2+G\left(\sigma_{33}-\sigma_{11}\right)^2+H\left(\sigma_{11}-\sigma_{22}\right)^2+2L\sigma_{23}^2+2M\sigma_{31}^2+2N\sigma_{12}^2-1=0$$

where σ_{y1}, σ_{y2}, and σ_{y3}, are the tensile yield stresses and σ_{y12}, σ_{y23}, and σ_{y31} are the shear yield stresses. The constants F, G H, L, M, and N are related to the yield stress by

$$2L=\frac{1}{\sigma_{y23}^2}$$

$$2M=\frac{1}{\sigma_{y31}^2}$$

$$2N=\frac{1}{\sigma_{y12}^2}$$

$$2F=\frac{1}{\sigma_{y2}^2}+\frac{1}{\sigma_{y3}^2}-\frac{1}{\sigma_{y1}^2}$$

$$2G=\frac{1}{\sigma_{y3}^2}+\frac{1}{\sigma_{y1}^2}-\frac{1}{\sigma_{y2}^2}$$

$$2H=\frac{1}{\sigma_{y1}^2}+\frac{1}{\sigma_{y2}^2}-\frac{1}{\sigma_{y3}^2}.$$

The isotropic case of von Mises plasticity can be recovered by setting $F=G=H=\frac{1}{2\sigma_y^2}$

and $$L=M=N=\frac{3}{2\sigma_y^2}$$

For the particular case of transverse anisotropy, where properties do not vary in the x_1-x_2 plane, the following relations hold:

$$2F = 2G = \frac{1}{\sigma_{y3}^2}$$

$$2H = \frac{2}{\sigma_y^2} - \frac{1}{\sigma_{y3}^2}$$

$$N = \frac{2}{\sigma_y^2} - \frac{1}{2}\frac{1}{\sigma_{y3}^2}$$

where it has been assumed that $\sigma_{y1} = \sigma_{y2} = \sigma_y$.

Letting $K = \dfrac{\sigma_y}{\sigma_{y3}}$, the yield criteria can be written

$$F(\sigma) = \sigma_e = \sigma_y,$$

where

$$F(\sigma) \equiv \Big[\sigma_{11}^2 + \sigma_{22}^2 + K^2\sigma_{33}^2 - K^2\sigma_{33}(\sigma_{11} + \sigma_{22}) - (2 - K^2)\sigma_{11}\sigma_{22}$$
$$+2L\sigma_y^2(\sigma_{23}^2 + \sigma_{31}^2) + 2\left(2 - \frac{1}{2}K^2\right)\sigma_{12}^2 \Big]^{1/2}$$

The rate of plastic strain is assumed to be normal to the yield surface so $\dot{\varepsilon}_{ij}^p$ is found from

$$\dot{\varepsilon}_{ij}^p = \lambda \frac{\partial F}{\partial \sigma_{ij}}.$$

Now consider the case of plane stress, where $\sigma_{33} = 0$. Also, define the anisotropy input parameter, R, as the ratio of the in-plane plastic strain rate to the out-of-plane plastic strain rate,

$$R = \frac{\dot{\varepsilon}_{22}^p}{\dot{\varepsilon}_{33}^p}.$$

It then follows that

$$R = \frac{2}{K^2} - 1.$$

Using the plane stress assumption and the definition of R, the yield function may now be written

$$F(\sigma)=\left[\sigma_{11}^2+\sigma_{22}^2-\frac{2R}{R+1}\sigma_{11}\sigma_{22}+2\frac{2R+1}{R+1}\sigma_{12}^2\right]^{1/2}.$$

Note that there are several differences between this model and other plasticity models for shell elements such as the model, MAT_PIECEWISE_LINEAR_PLASTICITY. First, the yield function for plane stress does not include the transverse shear stress components which are updated elastically, and, secondly, this model is always fully iterative. Consequently, in comparing results for the isotropic case where R=1.0 with other isotropic model, differences in the results are expected, even though they are usually insignificant.

The Young's modulus has been assumed to be constant. Recently, some researchers have found that Young's modulus decreases with respect to the increase of effective strain. To accommodate this new observation, a new option of _ECHANGE is added. There are two methods defining the change of Young's modulus change:

The first method is to use a curve to define the scale factor with respect to the effective strain. The value of this scale factor should decrease from 1 to 0 with the increase of effective strain.

The second method is to use a function as proposed by Yoshida [2003]:

$$E=E^0-(E^0-E^A)(1-exp(-\zeta\bar{\varepsilon})).$$

*MAT_BLATZ-KO_FOAM

This is Material Type 38. This model is for the definition of rubber like foams of polyurethane. It is a simple one-parameter model with a fixed Poisson's ratio of .25.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	G	REF				
Type	A8	F	F	F				

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
G	Shear modulus.
REF	Use reference geometry to initialize the stress tensor. The reference geometry is defined by the keyword:*INITIAL_FOAM_REFERENCE_GEOMETRY (see there for more details). EQ.0.0: off, EQ.1.0: on.

Remarks:

The strain energy functional for the compressible foam model is given by

$$W = \frac{G}{2}\left(\frac{\mathrm{II}}{\mathrm{III}} + 2\sqrt{\mathrm{III}} - 5\right)$$

Blatz and Ko [1962] suggested this form for a 47 percent volume polyurethane foam rubber with a Poisson's ratio of 0.25. In terms of the strain invariants, I, II, and III, the second Piola-Kirchhoff stresses are given as

$$S^{ij} = G\left[\left(I\delta_{ij} - C_{ij}\right)\frac{1}{\mathrm{III}} + \left(\sqrt{\mathrm{III}} - \frac{\mathrm{II}}{\mathrm{III}}\right)C_{ij}^{-1}\right]$$

where C_{ij} is the right Cauchy-Green strain tensor. This stress measure is transformed to the Cauchy stress, σ_{ij}, according to the relationship

$$\sigma^{ij} = \mathrm{III}^{-1/2} F_{ik} F_{jl} S_{lk}$$

where F_{ij} is the deformation gradient tensor.

*MAT_FLD_TRANSVERSELY_ANISOTROPIC

This is Material Type 39. This model is for simulating sheet forming processes with anisotropic material. Only transverse anisotropy can be considered. Optionally, an arbitrary dependency of stress and effective plastic strain can be defined via a load curve. A Forming Limit Diagram (FLD) can be defined using a curve and is used to compute the maximum strain ratio which can be plotted in LS-Prepost. This plasticity model is fully iterative and is available only for shell elements. Also see the notes below.

Card 1

Variable	MID	RO	E	PR	SIGY	ETAN	R	HLCID
Type	A8	F	F	F	F	F	F	F

Card 2

Variable	LCIDFLD							
Type	F							

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.
SIGY	Yield stress.
ETAN	Plastic hardening modulus, see notes for model 37.
R	Anisotropic hardening parameter, see notes for model 37.
HLCID	Load curve ID defining effective stress versus effective plastic strain. The yield stress and hardening modulus are ignored with this option.

VARIABLE	DESCRIPTION
LCIDFLD	Load curve ID defining the Forming Limit Diagram. Minor strains in percent are defined as abscissa values and Major strains in percent are defined as ordinate values. The forming limit diagram is shown in Figure 39.1. In defining the curve list pairs of minor and major strains starting with the left most point and ending with the right most point, see *DEFINE_CURVE.

Remarks:

See material model 37 for the theoretical basis. The first history variable is the maximum strain ratio defined by:

$$\frac{\varepsilon_{major_{workpiece}}}{\varepsilon_{major_{fld}}}$$

corresponding to $\varepsilon_{\text{minor}_{workpiece}}$.

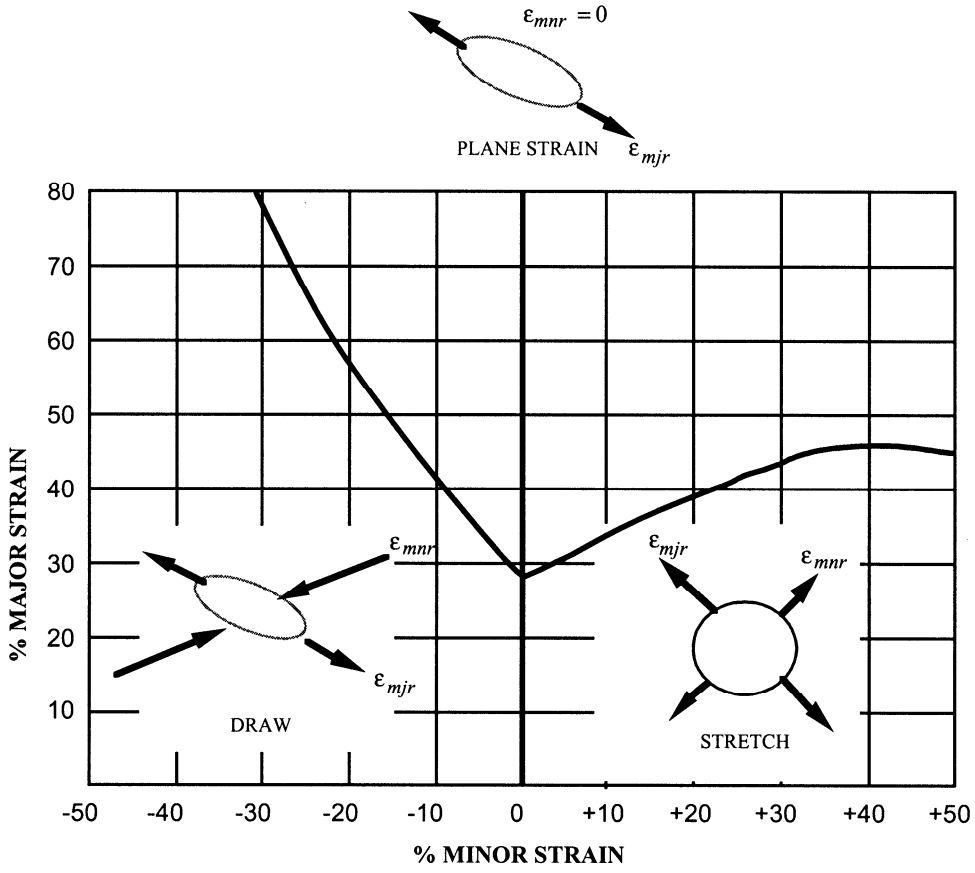


Figure 39.1. Forming Limit Diagram.

*MAT_NONLINEAR_ORTHOTROPIC

This is Material Type 40. This model allows the definition of an orthotropic nonlinear elastic material based on a finite strain formulation with the initial geometry as the reference. Failure is optional with two failure criteria available. Optionally, stiffness proportional damping can be defined. In the stress initialization phase, temperatures can be varied to impose the initial stresses. This model is only available for shell and solid elements. We do not recommend using this model at this time since it can be unstable especially if the stress-strain curves increase in stiffness with increasing strain.

Card 1

Variable	MID	RO	EA	EB	EC	PRBA	PRCA	PRCB
Type	A8	F	F	F	F	F	F	F
Default	None	none	none	none	none	none	none	none

Card 2

Variable	GAB	GBC	GCA	DT	TRAMP	ALPHA		
Type	F	F	F	F	F	F		
Default	None	none	none	0	0	0		

Card 3

Variable	LCIDA	LCIDB	EFAIL	DTFAIL	CDAMP	AOPT	MACF	
Type	F	F	F	F	F	F	I	
Default	0.0	0.0	0.0	0.0	0.0	0.0	0	

Card 4	1	2	3	4	5	6	7	8
Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card 5

Variable	V1	V2	V3	D1	D2	D3	BETA	
Type	F	F	F	F	F	F	F	

Optional Card 6 (Applies to Solid elements only)

Card 6	1	2	3	4	5	6	7	8
Variable	LCIDC	LCIDAB	LCIDBC	LCIDCA				
Type	F	F	F	F				
Default	optional	optional	optional	optional				

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
EA	E_a, Young's modulus in a-direction.
EB	E_b, Young's modulus in b-direction.
EC	E_c, Young's modulus in c-direction.
PRBA	ν_{ba}, Poisson's ratio ba.
PRCA	ν_{ca}, Poisson's ratio ca.
PRCB	ν_{cb}, Poisson's ratio cb.

VARIABLE	DESCRIPTION
GAB	G_{ab}, shear modulus ab.
GBC	G_{bc}, shear modulus bc.
GCA	G_{ca}, shear modulus ca.
DT	Temperature increment for isotropic stress initialization. This option can be used during dynamic relaxation.
TRAMP	Time to ramp up to the final temperature.
ALPHA	Thermal expansion coefficient.
LCIDA	Optional load curve ID defining the nominal stress versus strain along a-axis. Strain is defined as λ_a-1 where λ_a is the stretch ratio along the a axis.
LCIDB	Optional load curve ID defining the nominal stress versus strain along b-axis. Strain is defined as λ_b-1 where λ_b is the stretch ratio along the b axis.
EFAIL	Failure strain, λ-1.
DTFAIL	Time step for automatic element erosion
CDAMP	Damping coefficient.
AOPT	Material axes option (see MAT_*OPTION* TROPIC_ELASTIC for a more complete description): EQ.0.0: locally orthotropic with material axes determined by element nodes as shown in Figure 2.1. Nodes 1, 2, and 4 of an element are identical to the nodes used for the definition of a coordinate system as by *DEFINE_COORDINATE_NODES. EQ.1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center; this is the a-direction. This option is for solid elements only. EQ.2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_ VECTOR. EQ.3.0: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle, BETA, from a line in the plane of the element defined by the cross product of the vector **v** with the element normal. The plane of a solid element is the midsurface between the inner surface and outer surface defined by the first four nodes and the last four nodes of the connectivity of the element, respectively.

VARIABLE	DESCRIPTION
	EQ.4.0: locally orthotropic in cylindrical coordinate system with the material axes determined by a vector **v**, and an originating point, P, which define the centerline axis. This option is for solid elements only. LT.0.0: the absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_NODES, *DEFINE_COORDINATE_SYSTEM or *DEFINE_COORDINATE_VECTOR). Available in R3 version of 971 and later.
MACF	Material axes change flag: EQ.1: No change, default, EQ.2: switch material axes a and b, EQ.3: switch material axes a and c, EQ.4: switch material axes b and c.
XP YP ZP	Define coordinates of point **p** for AOPT = 1 and 4.
A1,A2,A3	a_1 a_2 a_3, define components of vector **a** for AOPT = 2.
D1,D2,D3	d_1 d_2 d_3, define components of vector **d** for AOPT = 2.
V1,V2,V3	v_1 v_2 v_3, define components of vector **v** for AOPT = 3 and 4.
BETA	Material angle in degrees for AOPT = 3, may be overridden on the element card, see *ELEMENT_SHELL_BETA.
The following input is optional and applies to <u>SOLID ELEMENTS</u> only.	
LCIDC	Load curve ID defining the nominal stress versus strain along c-axis. Strain is defined as λ_c-1 where λ_c is the stretch ratio along the c axis.
LCIDAB	Load curve ID defining the nominal ab shear stress versus ab-strain in the ab-plane. Strain is defined as the $\sin(\gamma_{ab})$ where γ_{ab} is the shear angle.
LCIDBC	Load curve ID defining the nominal ab shear stress versus ab-strain in the bc-plane. Strain is defined as the $\sin(\gamma_{bc})$ where γ_{bc} is the shear angle.
LCIDCA	Load curve ID defining the nominal ab shear stress versus ab-strain in the ca-plane. Strain is defined as the $\sin(\gamma_{ca})$ where γ_{bc} is the shear angle.

*MAT_USER_DEFINED_MATERIAL_MODELS

These are Material Types 41-50. The user can supply his own subroutines. See also Appendix A. The keyword input has to be used for the user interface with data. Isotopic and anisotropic material models with failure can be handled.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	MT	LMC	NHV	IORTHO	IBULK	IG
Type	A8	F	I	I	I	I	I	I

Card 2

Variable	IVECT	IFAIL	ITHERM	IHYPER	IEOS			
Type	I	I	I	I	I			

Define the following two cards if and only if IORTHO=1

Card 3	1	2	3	4	5	6	7	8
Variable	AOPT	MACF	XP	YP	ZP	A1	A2	A3
Type	F	I	F	F	F	F	F	F

Card 4

Variable	V1	V2	V3	D1	D2	D3	BETA	
Type	F	F	F	F	F	F	F	

Define LMC material parameters using 8 parameters per card.

Card	1	2	3	4	5	6	7	8
Variable	P1	P2	P3	P4	P5	P6	P7	P8
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
MT	User material type (41-50 inclusive). A number between 41 and 50 has to be chosen.
LMC	Length of material constant array which is equal to the number of material constants to be input. (LMC ≤ 40 if IORTHO=1)
NHV	Number of history variables to be stored, see Appendix A. When the model is to be used with an equation of state, NHV must be increased by 4 to allocate the storage required by the equation of state.
IORTHO	Set to 1 if the material is orthotropic.
IBULK	Address of bulk modulus in material constants array, see Appendix A.
IG	Address of shear modulus in material constants array, see Appendix A.
IVECT	Vectorization flag (on=1). A vectorized user subroutine must be supplied.
IFAIL	Failure flag. EQ.0: No failure, EQ.1: Allows failure of shell and solid elements, LT.0: |IFAIL| is the address of NUMINT in the material constants array. NUMINT is defined as the number of failed integration points that will trigger element deletion. This option applies only to shell and solid elements (release 5 of v.971).
ITHERM	Temperature flag (on=1). Compute element temperature.
IHYPER	Deformation gradient flag (on=1 or –1). Compute deformation gradient, see Appendix A.

VARIABLE	DESCRIPTION
IEOS	Equation of state (on=1).
AOPT	Material axes option (see MAT_*OPTION* TROPIC_ELASTIC for a more complete description): EQ.0.0: locally orthotropic with material axes determined by element nodes 1, 2, and 4, as with *DEFINE_COORDINATE_NODES. EQ.1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center; this is the a-direction. This option is for solid elements only. EQ.2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR. EQ.3.0: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle, BETA, from a line in the plane of the element defined by the cross product of the vector **v** with the element normal. EQ.4.0: locally orthotropic in cylindrical coordinate system with the material axes determined by a vector **v**, and an originating point, P, which define the centerline axis. This option is for solid elements only. LT.0.0: the absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_NODES, *DEFINE_COORDINATE_SYSTEM or *DEFINE_COORDINATE_VECTOR). Available in R3 version of 971 and later.
MACF	Material axes change flag for brick elements for quick changes: EQ.1: No change, default, EQ.2: switch material axes a and b, EQ.3: switch material axes a and c, EQ.4: switch material axes b and c.
XP YP ZP	Coordinates of point **p** for AOPT = 1.
A1 A2 A3	Components of vector **a** for AOPT = 2.
V1 V2 V3	Components of vector **v** for AOPT = 3.
D1 D2 D3	Components of vector **d** for AOPT = 2.
BETA	Material angle in degrees for AOPT = 3, may be overridden on the element card, see *ELEMENT_SHELL_BETA.
P1	First material parameter.
P2	Second material parameter.

P3	Third material parameter.
P4	Fourth material parameter.
.	.
PLMC	LMCth material parameter.

Remarks:

1. The material model for the cohesive element (solid element type 19) uses the first two material parameters to set flags for the element formulation. P1 controls how the density is used to calculate the mass. The cohesive element formulation permits the element to have zero or negative volume. Tractions are calculated on a surface midway between the surfaces defined by nodes 1-2-3-4 and 5-6-7-8. If P1 is set to 1.0, then the density is per unit area of the midsurface instead of per unit volume. The second parameter, P2, specifies the number of integration points (one to four) that are required to fail for the element to fail. If it is zero, the element won't fail regardless of the value of IFAIL. The recommended value of P2 is 1.

2. The cohesive element currently only uses MID, RO, MT, LMC, NHV, IFAIL and IVECT in addition to the material parameters.

3. See Appendix R for the specifics of the umat subroutine requirements for the cohesive element.

***MAT_BAMMAN**

This is Material Type 51. It allows the modeling of temperature and rate dependent plasticity with a fairly complex model that has many input parameters [Bamman 1989].

Card 1

	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	T	HC		
Type	A8	F	F	F	F	F		

Card 2

Variable	C1	C2	C3	C4	C5	C6	C7	C8
Type	F	F	F	F	F	F	F	F

Card 3

Variable	C9	C10	C11	C12	C13	C14	C15	C16
Type	F	F	F	F	F	F	F	F

Card 4

Variable	C17	C18	A1	A2	A4	A5	A6	KAPPA
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
E	Young's modulus (psi)

VARIABLE	DESCRIPTION
PR	Poisson's ratio
T	Initial temperature (^oR)
HC	Heat generation coefficient ($^oR/psi$)
C1	Psi
C2	^oR
C3	Psi
C4	^oR
C5	$1/s$
C6	^oR
C7	$1/psi$
C8	^oR
C9	Psi
C10	^oR
C11	$1/psi\text{-}s$
C12	^oR
C13	$1/psi$
C14	^oR
C15	psi
C16	^oR
C17	$1/psi\text{-}s$
C18	^oR
A1	α_1, initial value of internal state variable 1
A2	α_2, initial value of internal state variable 2. (Note: $\alpha_3 = -(\alpha_1+\alpha_2)$)

VARIABLE	DESCRIPTION
A3	α_4, initial value of internal state variable 3
A4	α_5, initial value of internal state variable 4
A5	α_6, initial value of internal state variable 5
KAPPA	κ, initial value of internal state variable 6

sec-psi-^{o}R	sec-MPa-^{o}R	sec-MPA-^{o}K
C_1	*1⁄145	*1⁄145
C_2	—	*5⁄9
C_3	*1⁄145	*1⁄145
C_4	—	*5⁄9
C_5	—	—
C_6	—	*5⁄9
C_7	*145	*145
C_8	—	*5⁄9
C_9	*1⁄145	*1⁄145
C_{10}	—	*5⁄9
C_{11}	*145	*145
C_{12}	—	*5⁄9
C_{13}	*145	*145
C_{14}	—	*5⁄9
C_{15}	*1⁄145	*1⁄145
C_{16}	—	*5⁄9
C_{17}	*145	*145
C_{18}	—	*5⁄9
C0=HC	*145	*145*5⁄9
E	*1⁄145	*1⁄145
υ	—	—
T	—	*5⁄9

Remarks:

The kinematics associated with the model are discussed in references [Hill 1948, Bammann and Aifantis 1987, Bammann 1989]. The description below is taken nearly verbatim from Bammann [1989].

With the assumption of linear elasticity we can write,

$$\overset{o}{\sigma} = \lambda \, \mathrm{tr}\left(D^e\right)1 + 2\mu D^e$$

where the Cauchy stress σ is convected with the elastic spin W^e as,

$$\underline{\overset{o}{\sigma} = \dot{\sigma} - W^e\sigma + \sigma W^e}$$

This is equivalent to writing the constitutive model with respect to a set of directors whose direction is defined by the plastic deformation [Bammann and Aifantis 1987, Bammann and Johnson 1987]. Decomposing both the skew symmetric and symmetric parts of the velocity gradient into elastic and plastic parts we write for the elastic stretching D^e and the elastic spin W^e,

$$D^e = D - D^p - D^{th}, \;\; W^e = W = W^p .$$

Within this structure it is now necessary to prescribe an equation for the plastic spin W^p in addition to the normally prescribed flow rule for D^p and the stretching due to the thermal expansion D^{th}. As proposed, we assume a flow rule of the form,

$$D^p = f\left(T\right)\sinh\left[\frac{\left|\xi\right| - \kappa - Y\left(T\right)}{V\left(T\right)}\right]\frac{\xi'}{\left|\xi'\right|}.$$

where T is the temperature, κ is the scalar hardening variable, and ξ' is the difference between the deviatoric Cauchy stress σ' and the tensor variable α',

$$\xi' = \sigma' - \alpha'$$

and f(T), Y(T), V(T) are scalar functions whose specific dependence upon the temperature is given below. Assuming isotropic thermal expansion and introducing the expansion coefficient $\dot{A}$, the thermal stretching can be written,

$$D^{th} = \dot{A}\dot{T}1$$

The evolution of the internal variables α and κ are prescribed in a hardening minus recovery format as,

$$\overset{o}{\alpha} = h(T)D^p - \left[r_d(T)\left|D^p\right| + r_s(T) \right] |\alpha| \alpha,$$

$$\dot{\kappa} = H(T)D^p - \left[R_d(T)\left|D^p\right| + R_s(T) \right] \kappa^2$$

where h and H are the hardening moduli, r_s (T) and R_s (T) are scalar functions describing the diffusion controlled 'static' or 'thermal' recovery, and r_d (T) and R_d (T) are the functions describing dynamic recovery.

If we assume that $W^p = 0$, we recover the Jaumann stress rate which results in the prediction of an oscillatory shear stress response in simple shear when coupled with a Prager kinematic hardening assumption [Johnson and Bammann 1984]. Alternatively we can choose,

$$W^p = R^T \dot{U} U^{-1} R,$$

which recovers the Green-Naghdi rate of Cauchy stress and has been shown to be equivalent to Mandel's isoclinic state [Bammann and Aifantis 1987]. The model employing this rate allows a reasonable prediction of directional softening for some materials, but in general under-predicts the softening and does not accurately predict the axial stresses which occur in the torsion of the thin walled tube.

The final equation necessary to complete our description of high strain rate deformation is one which allows us to compute the temperature change during the deformation. In the absence of a coupled thermo-mechanical finite element code we assume adiabatic temperature change and follow the empirical assumption that 90 -95% of the plastic work is dissipated as heat. Hence,

$$\dot{T} = \frac{.9}{\rho C_v}\left(\sigma \cdot D^p\right),$$

where ρ is the density of the material and C_v the specific heat.

In terms of the input parameters the functions defined above become:

V(T) = C1 exp(-C2/T)	h(T) = C9 exp(C10/T)
Y(T) = C3 exp(C4/T)	rs(T) = C11exp(-C12/T)
f(T) = C5 exp(-C6/T)	RD(T) = C13exp(-C14/T)
rd(T) = C7 exp(-C8/T)	H(T) = C15exp(C16/T)
	RS(T) = C17exp(-C18/T)

and the heat generation coefficient is

$$HC = \frac{.9}{\rho C_v}.$$

*MAT_BAMMAN_DAMAGE

This is Material Type 52. This is an extension of model 51 which includes the modeling of damage. See Bamman et al. [1990].

Card 1

	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	T	HC		
Type	A8	F	F	F	F	F		

Card 2

Variable	C1	C2	C3	C4	C5	C6	C7	C8
Type	F	F	F	F	F	F	F	F

Card 3

Variable	C9	C10	C11	C12	C13	C14	C15	C16
Type	F	F	F	F	F	F	F	F

Card 4

Variable	C17	C18	A1	A2	A3	A4	A5	A6
Type	F	F	F	F	F	F	F	F

Card 5

Variable	N	D0	FS					
Type	F	F	F					

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
E	Young's modulus (psi)
PR	Poisson's ratio
T	Initial temperature (^{o}R)
HC	Heat generation coefficient (^{o}R/psi)
C1	Psi
C2	^{o}R
C3	Psi
C4	^{o}R
C5	1/s
C6	^{o}R
C7	1/psi
C8	^{o}R
C9	Psi
C10	^{o}R
C11	1/psi-s
C12	^{o}R
C13	1/psi
C14	^{o}R
C15	psi
C16	^{o}R
C17	1/psi-s

VARIABLE	DESCRIPTION
C18	^{o}R
A1	α_1, initial value of internal state variable 1
A2	α_2, initial value of internal state variable 2
A3	α_3, initial value of internal state variable 3
A4	α_4, initial value of internal state variable 4
A5	α_5, initial value of internal state variable 5
A6	α_6, initial value of internal state variable 6
N	Exponent in damage evolution
D0	Initial damage (porosity)
FS	Failure strain for erosion.

Remarks:

The evolution of the damage parameter, ϕ is defined by Bammann et al. [1990]

$$\dot{\phi} = \beta\left[\frac{1}{(1-\phi)^N} - (1-\phi)\right]^{|D^p|}$$

in which

$$\beta = \sinh\left[\frac{2(2N-1)\,p}{(2N-1)\,\bar{\sigma}}\right]$$

where p is the pressure and $\bar{\sigma}$ is the effective stress.

*MAT_CLOSED_CELL_FOAM

This is Material Type 53. This allows the modeling of low density, closed cell polyurethane foam. It is for simulating impact limiters in automotive applications. The effect of the confined air pressure is included with the air being treated as an ideal gas. The general behavior is isotropic with uncoupled components of the stress tensor.

Card 1

	1	2	3	4	5	6	7	8
Variable	MID	RO	E	A	B	C	P0	PHI
Type	A8	F	F	F	F	F	F	F

Card 2

Variable	GAMA0	LCID						
Type	F	I						

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
E	Young's modulus
A	a, factor for yield stress definition, see notes below.
B	b, factor for yield stress definition, see notes below.
C	c, factor for yield stress definition, see notes below.
P0	Initial foam pressure, P_0
PHI	Ratio of foam to polymer density, ϕ
GAMA0	Initial volumetric strain, $\gamma_{0.}$ The default is zero.

VARIABLE	DESCRIPTION
LCID	Optional load curve defining the von Mises yield stress versus $-\gamma$. If the load curve ID is given, the yield stress is taken from the curve and the constants a, b, and c are not needed. The load curve is defined in the positive quadrant, i.e., positive values of γ are defined as negative values on the abscissa.

Remarks:

A rigid, low density, closed cell, polyurethane foam model developed at Sandia Laboratories [Neilsen, Morgan and Krieg 1987] has been recently implemented for modeling impact limiters in automotive applications. A number of such foams were tested at Sandia and reasonable fits to the experimental data were obtained.

In some respects this model is similar to the crushable honeycomb model type 26 in that the components of the stress tensor are uncoupled until full volumetric compaction is achieved. However, unlike the honeycomb model this material possesses no directionality but includes the effects of confined air pressure in its overall response characteristics.

$$\sigma_{ij} = \sigma_{ij}^{sk} - \delta_{ij}\sigma^{air}$$

where σ_{ij}^{sk} is the skeletal stress and σ^{air} is the air pressure computed from the equation:

$$\sigma^{air} = -\frac{p_0\gamma}{1+\gamma-\phi}$$

where p_0 is the initial foam pressure, usually taken as the atmospheric pressure, and γ defines the volumetric strain

$$\gamma = V - 1 + \gamma_0$$

where V is the relative volume, defined as the ratio of the current volume to the initial volume, and γ_0 is the initial volumetric strain, which is typically zero. The yield condition is applied to the principal skeletal stresses, which are updated independently of the air pressure. We first obtain the skeletal stresses:

$$\sigma_{ij}^{sk} = \sigma_{ij} + \sigma_{ij}\sigma^{air}$$

and compute the trial stress, σ^{skt}

$$\sigma_{ij}^{skt} = \sigma_{ij}^{sk} + E\,\dot{\varepsilon}_{ij}\,\Delta t$$

where E is Young's modulus. Since Poisson's ratio is zero, the update of each stress component is uncoupled and 2G=E where G is the shear modulus. The yield condition is applied to the principal skeletal stresses such that, if the magnitude of a principal trial stress component, σ_i^{skt}, exceeds the yield stress, σ_y, then

$$\sigma_i^{sk} = \min\left(\sigma_y, \left|\sigma_i^{skt}\right|\right) \frac{\sigma_i^{skt}}{\left|\sigma_i^{skt}\right|}$$

The yield stress is defined by

$$\sigma_y = a + b(1 + c\gamma)$$

where a, b, and c are user defined input constants and γ is the volumetric strain as defined above. After scaling the principal stresses they are transformed back into the global system and the final stress state is computed

$$\sigma_{ij} = \sigma_{ij}^{sk} - \delta_{ij}\sigma^{air} .$$

*MAT_ENHANCED_COMPOSITE_DAMAGE

These are Material Types 54-55 which are enhanced versions of the composite model material type 22. Arbitrary orthothropic materials, e.g., unidirectional layers in composite shell structures can be defined. Optionally, various types of failure can be specified following either the suggestions of [Chang and Chang 1987b] or [Tsai and Wu 1971]. In addition special measures are taken for failure under compression. See [Matzenmiller and Schweizerhof 1991]. This model is only valid for thin shell elements. The parameters in parentheses below apply only to solid elements and are therefore always ignored in this material model. They are included for consistency with material types 22 and 59. By using the user defined integration rule, see *INTEGRATION_SHELL, the constitutive constants can vary through the shell thickness. For all shells, except the DKT formulation, laminated shell theory can be activated to properly model the transverse shear deformation. Lamination theory is applied to correct for the assumption of a uniform constant shear strain through the thickness of the shell. For sandwich shells where the outer layers are much stiffer than the inner layers, the response will tend to be too stiff unless lamination theory is used. To turn on lamination theory see *CONTROL_SHELL. A damage model for transverse shear strain is added since version 971 release R4 to model interlaminar shear failure. The definition of minimum stress limits is available since version 971 R5.

Card 1

Variable	MID	RO	EA	EB	(EC)	PRBA	PRCA	PRCB
Type	A8	F	F	F	F	F	F	F

Card 2

Variable	GAB	GBC	GCA	(KF)	AOPT			
Type	F	F	F	F	F			

Card 3

Variable				A1	A2	A3	MANGLE	
Type				F	F	F	F	

Card 4	1	2	3	4	5	6	7	8
Variable	V1	V2	V3	D1	D2	D3	DFAILM	DFAILS
Type	F	F	F	F	F	F	F	F

Card 5

Variable	TFAIL	ALPH	SOFT	FBRT	YCFAC	DFAILT	DFAILC	EFS
Type	F	F	F	F	F	F	F	F

Card 6

Variable	XC	XT	YC	YT	SC	CRIT	BETA	
Type	F	F	F	F	F	F	F	

Optional Card 7 (starting with version 971 release R4)

Variable	PFL	EPSF	EPSR	TSMD	SOFT2			
Type	F	F	F	F	F			

Optional Card 8 (starting with version 971 release R5)

Variable	SLIMT1	SLIMC1	SLIMT2	SLIMC2	SLIMS	NCYRED		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density

VARIABLE	DESCRIPTION
EA	E_a, Young's modulus - longitudinal direction
EB	E_b, Young's modulus - transverse direction
(EC)	E_c, Young's modulus - normal direction (not used)
PRBA	ν_{ba}, Poisson's ratio ba
PRCA	ν_{ca}, Poisson's ratio ca (if 0, defaults to PRBA for shell thicness update)
PRCB	ν_{cb}, Poisson's ratio cb (if 0, defaults to PRBA for shell thickness update)
GAB	G_{ab}, shear modulus ab
GBC	G_{bc}, shear modulus bc
GCA	G_{ca}, shear modulus ca
(KF)	Bulk modulus of failed material (not used)
AOPT	Material axes option (see MAT_*OPTION* TROPIC_ELASTIC for a more complete description): EQ.0.0: locally orthotropic with material axes determined by element nodes 1, 2, and 4, as with *DEFINE_COORDINATE_NODES. EQ.2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR. EQ.3.0: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle (MANGLE) from a line in the plane of the element defined by the cross product of the vector **v** with the element normal. LT.0.0: the absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_NODES, *DEFINE_COORDINATE_SYSTEM or *DEFINE_COORDINATE_VECTOR). Available in R3 version of 971 and later.
A1 A2 A3	Define components of vector **a** for AOPT = 2.
V1 V2 V3	Define components of vector **v** for AOPT = 3.
D1 D2 D3	Define components of vector **d** for AOPT = 2.
MANGLE	Material angle in degrees for AOPT = 3, may be overridden on the element card, see *ELEMENT_SHELL_BETA or *ELEMENT_SOLID_ORTHO.

VARIABLE	DESCRIPTION
DFAILM	Maximum strain for matrix straining in tension or compression (active only for MAT_054 and only if DFAILT > 0). The layer in the element is completely removed after the maximum strain in the matrix direction is reached. The input value is always positive.
DFAILS	Maximum shear strain (active only for MAT_054 and only if DFAILT > 0). The layer in the element is completely removed after the maximum shear strain is reached. The input value is always positive.
TFAIL	Time step size criteria for element deletion: ≤ 0: no element deletion by time step size. The crashfront algorithm only works if t_{fail} is set to a value above zero. $0 < t_{fail} \leq 0.1$: element is deleted when its time step is smaller than the given value, >.1: element is deleted when the quotient of the actual time step and the original time step drops below the given value.
ALPH	Shear stress parameter for the nonlinear term, see Material 22.
SOFT	Softening reduction factor for material strength in crashfront elements (default = 1.0). TFAIL must be greater than zero to activate this option.
FBRT	Softening for fiber tensile strength: EQ.0.0: tensile strength = X_t GT.0.0: tensile strength = X_t, reduced to $X_t * FBRT$ after failure has occurred in compressive matrix mode.
YCFAC	Reduction factor for compressive fiber strength after matrix compressive failure (MAT_054 only). The compressive strength in the fiber direction after compressive matrix failure is reduced to: $X_c = YCFAC * Y_c$ ($default: YCFAC = 2.0$)
DFAILT	Maximum strain for fiber tension (MAT_054 only). (Maximum 1 = 100% strain). The layer in the element is completely removed after the maximum tensile strain in the fiber direction is reached. If a nonzero value is given for DFAILT, a nonzero, negative value must also be provided for DFAILC.
DFAILC	Maximum strain for fiber compression (MAT_054 only). (Maximum -1 = 100% compression). The layer in the element is completely removed after the maximum compressive strain in the fiber direction is reached. The input value should be negative and is required if DFAILT > 0.
EFS	Effective failure strain (MAT_054 only).
XC	Longitudinal compressive strength (positive value).

VARIABLE	DESCRIPTION
XT	Longitudinal tensile strength, see below.
YC	Transverse compressive strength, b-axis (positive value), see below.
YT	Transverse tensile strength, b-axis, see below.
SC	Shear strength, ab plane, see below.
CRIT	Failure criterion (material number): EQ.54.0: Chang matrix failure criterion (as Material 22) (default), EQ.55.0: Tsai-Wu criterion for matrix failure.
BETA	Weighting factor for shear term in tensile fiber mode (MAT_054 only). ($0.0 \le BETA \le 1.0$)
PFL	Percentage of layers which must fail until crashfront is initiated. E.g. |PFL|=80.0, then 80 % of layers must fail until strengths are reduced in neighboring elements. Default: all layers must fail. A single layer fails if 1 in-plane IP fails (PFL>0) or if 4 in-plane IPs fail (PFL<0). (MAT_054 only).
EPSF	Damage initiation transverser shear strain. (MAT_054 only).
EPSR	Final rupture transverse shear strain. (MAT_054 only).
TSMD	Transverse shear maximum damage, default=0.90. (MAT_054 only).
SOFT2	Optional “orthogonal” softening reduction factor for material strength in crashfront elements (default = 1.0). See remarks.
SLIMT1	Factor to determine the minimum stress limit after stress maximum (fiber tension). Similar to *MAT_058.
SLIMC1	Factor to determine the minimum stress limit after stress maximum (fiber compression). Similar to *MAT_058.
SLIMT2	Factor to determine the minimum stress limit after stress maximum (matrix tension). Similar to *MAT_058.
SLIMT2	Factor to determine the minimum stress limit after stress maximum (matrix compression). Similar to *MAT_058.
SLIMS	Factor to determine the minimum stress limit after stress maximum (shear). Similar to *MAT_058.
NCYRED	Number of cycles for stress reduction from maximum to minimum.

Remarks:

The Chang/Chang (mat_54) criteria is given as follows:

for the tensile fiber mode,

$$\sigma_{aa} > 0 \quad then \quad e_f^2 = \left(\frac{\sigma_{aa}}{X_t}\right)^2 + \beta\left(\frac{\sigma_{ab}}{S_c}\right) - 1 \begin{cases} \geq 0 \ failed \\ < 0 \ elastic \end{cases},$$

$$E_a = E_b = G_{ab} = \nu_{ba} = \nu_{ab} = 0,$$

for the compressive fiber mode,

$$\sigma_{aa} < 0 \quad then \quad e_c^2 = \left(\frac{\sigma_{aa}}{X_c}\right)^2 - 1 \begin{cases} \geq 0 \ failed \\ < 0 \ elastic \end{cases},$$

$$E_a = \nu_{ba} = \nu_{ab} = 0.$$

for the tensile matrix mode,

$$\sigma_{bb} > 0 \quad then \quad e_m^2 = \left(\frac{\sigma_{bb}}{Y_t}\right)^2 + \left(\frac{\sigma_{ab}}{S_c}\right)^2 - 1 \begin{cases} \geq 0 \ failed \\ < 0 \ elastic \end{cases},$$

$$E_b = \nu_{ba} = 0. \quad \rightarrow G_{ab} = 0,$$

and for the compressive matrix mode,

$$\sigma_{bb} < 0 \quad then \quad e_d^2 = \left(\frac{\sigma_{bb}}{2S_c}\right)^2 + \left[\left(\frac{Y_c}{2S_c}\right)^2 - 1\right]\frac{\sigma_{bb}}{Y_c} + \left(\frac{\sigma_{ab}}{S_c}\right)^2 - 1 \begin{cases} \geq 0 \ failed \\ < 0 \ elastic \end{cases},$$

$$_b = \nu_{ba} = \nu_{ab} = 0. \quad \rightarrow G_{ab} = 0$$
$$X_c = 2Y_c \quad for \ 50\% \ fiber \ volume.$$

In the Tsai-Wu (MAT_055) criteria the tensile and compressive fiber modes are treated as in the Chang-Chang criteria. The failure criterion for the tensile and compressive matrix mode is given as:

$$e_{md}^2 = \frac{\sigma_{bb}^2}{Y_c Y_t} + \left(\frac{\sigma_{ab}}{S_c}\right)^2 + \frac{(Y_c - Y_t)\sigma_{bb}}{Y_c Y_t} - 1 \begin{cases} \geq 0 \ failed \\ < 0 \ elastic \end{cases}$$

For β=1 we get the original criterion of Hashin [1980] in the tensile fiber mode. For β=0 we get the maximum stress criterion which is found to compare better to experiments.

In MAT_054, failure can occur in any of four different ways:

1. If DFAILT is zero, failure occurs if the Chang-Chang failure criterion is satisfied in the tensile fiber mode.

2. If DFAILT is greater than zero, failure occurs if the tensile fiber strain is greater than DFAILT or less than DFAILC.

3. If EFS is greater than zero, failure occurs if the effective strain is greater than EFS.

4. If TFAIL is greater than zero, failure occurs according to the element timestep as described in the definition of TFAIL above.

When failure has occurred in all the composite layers (through-thickness integration points), the element is deleted. Elements which share nodes with the deleted element become "crashfront" elements and can have their strengths reduced by using the SOFT parameter with TFAIL greater than zero. An earlier initiation of crashfront elements is possible by using parameter PFL.

An optional direction dependent strength reduction can be invoked by setting 0<SOFT2<1. Then, SOFT equals a strength reduction factor for fiber parallel failure and SOFT2 equals a strength reduction factor for fiber orthogonal failure. Linear interpolation is used for angles in between. See Figure 54.2.

Information about the status in each layer (integration point) and element can be plotted using additional integration point variables. The number of additional integration point variables for shells written to the LS-DYNA database is input by the *DATABASE_EXTENT_BINARY definition as variable NEIPS. For Models 54 and 55 these additional variables are tabulated below (i = shell integration point):

History Variable	Description	Value	LS-Prepost history variable
1.$ef(i)$	*tensile fiber mode*	*1 - elastic* *0 - failed*	*1*
2.$ec(i)$	*compressive fiber mode*		*2*
3.$em(i)$	*tensile matrix mode*		*3*
4.$ed(i)$	*compressive matrix mode*		*4*
5.*efail*	*max[ef(ip)]*		*5*
6.*dam*	*damage parameter*	*-1 - element intact* *10^{-8} - element in crashfront* *+1 – element failed*	*6*

These variables can be plotted in LS-Prepost element history variables 1 to 6. The following components, defined by the sum of failure indicators over all through-thickness integration points, are stored as element component 7 instead of the effective plastic strain.

Description	Integration point
$\frac{1}{nip}\sum_{i=1}^{nip} ef(i)$	*1*
$\frac{1}{nip}\sum_{i=1}^{nip} ec(i)$	*2*
$\frac{1}{nip}\sum_{i=1}^{nip} em(i)$	*3*

In an optional damage model for transverse shear strain, out-of-plane stiffness (GBC and GCA) can get linearly decreased to model interlaminar shear failure. Damage starts when effective transverse shear strain

$$\varepsilon_{56}^{eff} = \sqrt{\varepsilon_{yz}^2 + \varepsilon_{zx}^2}$$

reaches EPSF. Final rupture occurs when effective transverse shear strain reaches EPSR. A maximum damage of TSMD (0.0<TSMD<0.99) cannot be exceeded. See Figure 54.1.

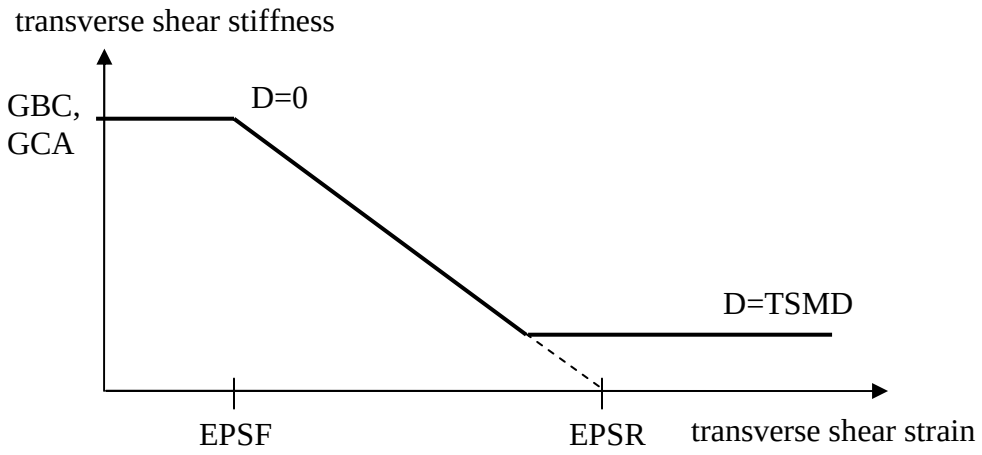


Figure 54.1 Linear damage for transverse shear behavior

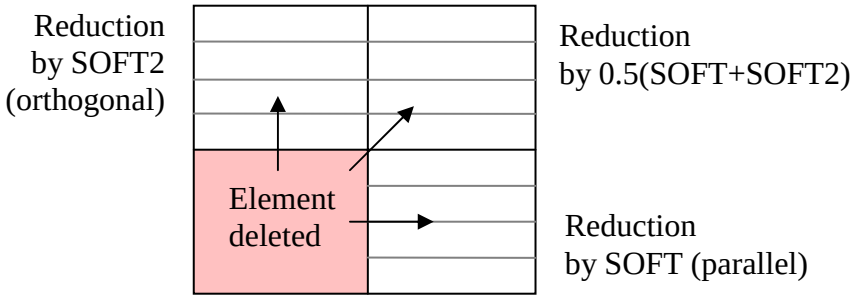


Figure 54.2 Direction dependent softening

*MAT_LOW_DENSITY_FOAM

This is Material Type 57 for modeling highly compressible low density foams. Its main applications are for seat cushions and padding on the Side Impact Dummies (SID). Optionally, a tension cut-off failure can be defined. Also, see the notes below.

Card 1

Variable	MID	RO	E	LCID	TC	HU	BETA	DAMP
Type	A8	F	F	F	F	F	F	F
Default	---	---	---	---	1.E+20	1.		0.05
Remarks	---	---	---	---	---	3	1	---

Card 2

Variable	SHAPE	FAIL	BVFLAG	ED	BETA1	KCON	REF	
Type	F	F	F	F	F	F	F	
Default	1.0	0.0	0.0	0.0	0.0	0.0	0.0	
Remarks	3	---	2	5	5	6		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
E	Young's modulus used in tension. For implicit problems E is set to the initial slope of load curve LCID.
LCID	Load curve ID, see *DEFINE_CURVE, for nominal stress versus strain.
TC	Tension cut-off stress

VARIABLE	DESCRIPTION
HU	Hysteretic unloading factor between 0 and 1 (default=1, i.e., no energy dissipation), see also Figure 57.1.
BETA	β, decay constant to model creep in unloading
DAMP	Viscous coefficient (.05< recommended value <.50) to model damping effects. LT.0.0: |DAMP| is the load curve ID, which defines the damping constant as a function of the maximum strain in compression defined as: $\varepsilon_{\max} = \max(1-\lambda_1, 1-\lambda_2, 1.-\lambda_3)$. In tension, the damping constant is set to the value corresponding to the strain at 0. The abscissa should be defined from 0 to 1.
SHAPE	Shape factor for unloading. Active for nonzero values of the hysteretic unloading factor. Values less than one reduces the energy dissipation and greater than one increases dissipation, see also Figure 57.1.
FAIL	Failure option after cutoff stress is reached: EQ.0.0: tensile stress remains at cut-off value, EQ.1.0: tensile stress is reset to zero.
BVFLAG	Bulk viscosity activation flag, see remark below: EQ.0.0: no bulk viscosity (recommended), EQ.1.0: bulk viscosity active.
ED	Optional Young's relaxation modulus, E_d, for rate effects. See comments below.
BETA1	Optional decay constant, β_1.
KCON	Stiffness coefficient for contact interface stiffness. If undefined the maximum slope in stress vs. strain curve is used. When the maximum slope is taken for the contact, the time step size for this material is reduced for stability. In some cases Δt may be significantly smaller, and defining a reasonable stiffness is recommended.
REF	Use reference geometry to initialize the stress tensor. The reference geometry is defined by the keyword:*INITIAL_FOAM_REFERENCE_GEOMETRY (see there for more details). EQ.0.0: off, EQ.1.0: on.

Remarks:

The compressive behavior is illustrated in Figure 57.1 where hysteresis on unloading is shown. This behavior under uniaxial loading is assumed not to significantly couple in the transverse directions. In tension the material behaves in a linear fashion until tearing occurs. Although our implementation may be somewhat unusual, it was motivated by Storakers [1986].

The model uses tabulated input data for the loading curve where the nominal stresses are defined as a function of the elongations, ε_i, which are defined in terms of the principal stretches, λ_i, as:

$$\varepsilon_i = \lambda_i - 1$$

The stretch ratios are found by solving for the eigenvalues of the left stretch tensor, V_{ij}, which is obtained via a polar decomposition of the deformation gradient matrix, F_{ij}. Recall that,

$$F_{ij} = R_{ik}U_{kj} = V_{ik}R_{kj}$$

The update of V_{ij} follows the numerically stable approach of Taylor and Flanagan [1989]. After solving for the principal stretches, we compute the elongations and, if the elongations are compressive, the corresponding values of the nominal stresses, τ_i are interpolated. If the elongations are tensile, the nominal stresses are given by

$$\tau_i = E\varepsilon_i$$

and the Cauchy stresses in the principal system become

$$\sigma_i = \frac{\tau_i}{\lambda_j \lambda_k}$$

The stresses can now be transformed back into the global system for the nodal force calculations.

Additional Remarks:

1. When hysteretic unloading is used the reloading will follow the unloading curve if the decay constant, β, is set to zero. If β is nonzero the decay to the original loading curve is governed by the expression:
$$1. - e^{-\beta t}$$

2. The bulk viscosity, which generates a rate dependent pressure, may cause an unexpected volumetric response and, consequently, it is optional with this model.

3. The hysteretic unloading factor results in the unloading curve to lie beneath the loading curve as shown in Figure 57.1. This unloading provides energy dissipation which is reasonable in certain kinds of foam.

4. Note that since this material has no effective plastic strain, the internal energy per initial volume is written into the output databases.

5. Rate effects are accounted for through linear viscoelasticity by a convolution integral of the form

$$\sigma_{ij}^{r} = \int_0^t g_{ijkl}\left(t-\tau\right)\frac{\partial \varepsilon_{kl}}{\partial \tau}d\tau$$

where $g_{ijkl}(t-\tau)$ is the relaxation function. The stress tensor, σ_{ij}^{r}, augments the stresses determined from the foam, σ_{ij}^{f}; consequently, the final stress, σ_{ij}, is taken as the summation of the two contributions:

$$\sigma_{ij} = \sigma_{ij}^{f} + \sigma_{ij}^{r}.$$

Since we wish to include only simple rate effects, the relaxation function is represented by one term from the Prony series:

$$g(t) = \alpha_0 + \sum_{m=1}^{N} \alpha_m e^{-\beta t}$$

given by,

$$g(t) = E_d e^{-\beta_1 t}$$

This model is effectively a Maxwell fluid which consists of a damper and spring in series. We characterize this in the input by a Young's modulus, E_d, and decay constant, β_1. The formulation is performed in the local system of principal stretches where only the principal values of stress are computed and triaxial coupling is avoided. Consequently, the one-dimensional nature of this foam material is unaffected by this addition of rate effects. The addition of rate effects necessitates twelve additional history variables per integration point. The cost and memory overhead of this model comes primarily from the need to "remember" the local system of principal stretches.

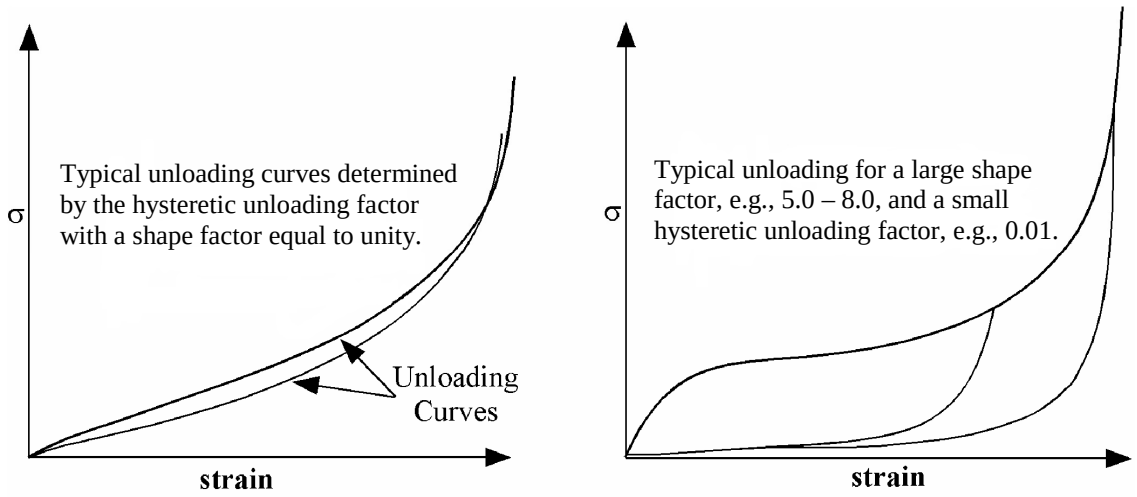


Figure 57.1. Behavior of the low density urethane foam model.

6. The time step size is based on the current density and the maximum of the instantaneous loading slope, E, and KCON. If KCON is undefined the maximum slope in the loading curve is used instead.

*MAT_LAMINATED_COMPOSITE_FABRIC

This is Material Type 58. Depending on the type of failure surface, this model may be used to model composite materials with unidirectional layers, complete laminates, and woven fabrics. This model is implemented for shell and thick shell elements.

Card 1

Variable	MID	RO	EA	EB	(EC)	PRBA	TAU1	GAMMA1
Type	A8	F	F	F	F	F	F	F

Card 2

Variable	GAB	GBC	GCA	SLIMT1	SLIMC1	SLIMT2	SLIMC2	SLIMS
Type	F	F	F	F	F	F	F	F

Card 3

Variable	AOPT	TSIZE	ERODS	SOFT	FS			
Type	F	F	F	F	F			

Card 4

Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card 5	1	2	3	4	5	6	7	8
Variable	V1	V2	V3	D1	D2	D3	BETA	
Type	F	F	F	F	F	F	F	

Card 6

Variable	E11C	E11T	E22C	E22T	GMS			
Type	F	F	F	F	F			

Card 7

Variable	XC	XT	YC	YT	SC			
Type	F	F	F	F	F			

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
EA	E_a, Young's modulus - longitudinal direction
EB	E_b, Young's modulus - transverse direction
(EC)	E_c, Young's modulus - normal direction (not used)
PRBA	ν_{ba}, Poisson's ratio ba
TAU1	$\tau 1$, stress limit of the first slightly nonlinear part of the shear stress versus shear strain curve. The values $\tau 1$ and $\gamma 1$ are used to define a curve of shear stress versus shear strain. These values are input if FS, defined below, is set to a value of -1.
GAMMA1	$\gamma 1$, strain limit of the first slightly nonlinear part of the shear stress versus shear strain curve.

VARIABLE	DESCRIPTION
GAB	G_{ab}, shear modulus ab
GBC	G_{bc}, shear modulus bc
GCA	G_{ca}, shear modulus ca
SLIMT1	Factor to determine the minimum stress limit after stress maximum (fiber tension).
SLIMC1	Factor to determine the minimum stress limit after stress maximum (fiber compression).
SLIMT2	Factor to determine the minimum stress limit after stress maximum (matrix tension).
SLIMC2	Factor to determine the minimum stress limit after stress maximum (matrix compression).
SLIMS	Factor to determine the minimum stress limit after stress maximum (shear).
AOPT	Material axes option (see MAT_*OPTION* TROPIC_ELASTIC for a more complete description): EQ. 0.0: locally orthotropic with material axes determined by element nodes 1, 2, and 4, as with *DEFINE_COORDINATE_NODES. EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR. EQ. 3.0: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle (BETA) from a line in the plane of the element defined by the cross product of the vector **v** with the element normal. LT.0.0: the absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_NODES, *DEFINE_COORDINATE_SYSTEM or *DEFINE_COORDINATE_VECTOR). Available in R3 version of 971 and later.
TSIZE	Time step for automatic element deletion.
ERODS	Maximum effective strain for element layer failure. A value of unity would equal 100% strain.
SOFT	Softening reduction factor for strength in the crashfront.

VARIABLE	DESCRIPTION
FS	Failure surface type: EQ.1.0: smooth failure surface with a quadratic criterion for both the fiber (a) and transverse (b) directions. This option can be used with complete laminates and fabrics. EQ.0.0: smooth failure surface in the transverse (b) direction with a limiting value in the fiber (a) direction. This model is appropriate for unidirectional (UD) layered composites only. EQ.-1.: faceted failure surface. When the strength values are reached then damage evolves in tension and compression for both the fiber and transverse direction. Shear behavior is also considered. This option can be used with complete laminates and fabrics.
XP YP ZP	Define coordinates of point **p** for AOPT = 1.
A1 A2 A3	Define components of vector **a** for AOPT = 2.
V1 V2 V3	Define components of vector **v** for AOPT = 3.
D1 D2 D3	Define components of vector **d** for AOPT = 2.
BETA	Material angle in degrees for AOPT=3, may be overridden on the element card, see *ELEMENT_SHELL_BETA.
E11C	Strain at longitudinal compressive strength, a-axis (positive).
E11T	Strain at longitudinal tensile strength, a-axis.
E22C	Strain at transverse compressive strength, b-axis.
E22T	Strain at transverse tensile strength, b-axis.
GMS	Strain at shear strength, ab plane.
XC	Longitudinal compressive strength (positive value).
XT	Longitudinal tensile strength, see below.
YC	Transverse compressive strength, b-axis (positive value), see below.
YT	Transverse tensile strength, b-axis, see below.
SC	Shear strength, ab plane, see below.

Remarks:

Parameters to control failure of an element layer are: ERODS, the maximum effective strain, i.e., maximum 1 = 100 % straining. The layer in the element is completely removed after the maximum effective strain (compression/tension including shear) is reached.

The stress limits are factors used to limit the stress in the softening part to a given value,

$$\sigma_{\min} = SLIMxx \cdot strength,$$

thus, the damage value is slightly modified such that elastoplastic like behavior is achieved with the threshold stress. As a factor for *SLIMxx* a number between 0.0 and 1.0 is possible. With a factor of 1.0, the stress remains at a maximum value identical to the strength, which is similar to ideal elastoplastic behavior. For tensile failure a small value for *SLIMTx* is often reasonable; however, for compression *SLIMCx = 1.0* is preferred. This is also valid for the corresponding shear value. If *SLIMxx* is smaller than 1.0 then localization can be observed depending on the total behavior of the lay-up. If the user is intentionally using *SLIMxx < 1.0,* it is generally recommended to avoid a drop to zero and set the value to something in between 0.05 and 0.10. Then elastoplastic behavior is achieved in the limit which often leads to less numerical problems. Defaults for SLIMXX = 1.0E-8.

The crashfront-algorithm is started if and only if a value for TSIZE (time step size, with element elimination after the actual time step becomes smaller than TSIZE) is input.

The damage parameters can be written to the postprocessing database for each integration point as the first three additional element variables and can be visualized.

Material models with FS=1 or FS=-1 are favorable for complete laminates and fabrics, as all directions are treated in a similar fashion.

For material model FS=1 an interaction between normal stresses and the shear stresses is assumed for the evolution of damage in the a and b-directions. For the shear damage is always the maximum value of the damage from the criterion in a or b-direction is taken.

For material model FS=-1 it is assumed that the damage evolution is independent of any of the other stresses. A coupling is only present via the elastic material parameters and the complete structure.

In tensile and compression directions and in a as well as in b- direction different failure surfaces can be assumed. The damage values, however, increase only also when the loading direction changes.

Special control of shear behavior of fabrics

For fabric materials a nonlinear stress strain curve for the shear part for failure surface FS=-1 can be assumed as given below. This is not possible for other values of FS.

The curve, shown in Figure 58.1 is defined by three points:

a) the origin (0,0) is assumed,

b) the limit of the first slightly nonlinear part (must be input), stress (TAU1) and strain (GAMMA1), see below.

c) the shear strength at failure and shear strain at failure.

In addition a stress limiter can be used to keep the stress constant via the *SLIMS* parameter. This value must be less or equal 1.0 but positive, and leads to an elastoplastic behavior for the shear part. The default is 1.0E-08, assuming almost brittle failure once the strength limit SC is reached.

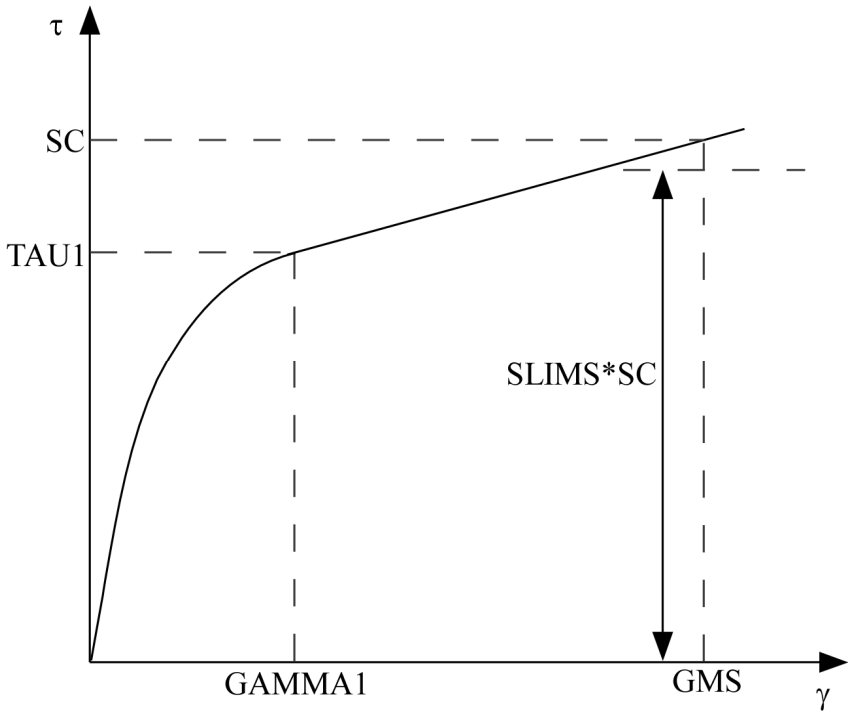


Figure 58.1. Stress-strain diagram for shear.

*MAT_COMPOSITE_FAILURE_*OPTION*_MODEL

This is Material Type 59.

Available options include:

SHELL

SOLID

SPH

depending on the element type the material is to be used with, see *PART.

For both options define cards 1 to 4 below

Card 1

Variable	MID	RO	EA	EB	EC	PRBA	PRCA	PRCB
Type	A8	F	F	F	F	F	F	F

Card 2

Variable	GAB	GBC	GCA	KF	AOPT	MACF		
Type	F	F	F	F	F	I		

Card 3

Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card 4

Variable	V1	V2	V3	D1	D2	D3	BETA	
Type	F	F	F	F	F	F	F	

Cards 5 and 6 for SHELL option

Card 5

	1	2	3	4	5	6	7	8
Variable	TSIZE	ALP	SOFT	FBRT	SR	SF		
Type	F	F	F	F	F	F		

Card 6

Variable	XC	XT	YC	YT	SC			
Type	F	F	F	F	F			

Cards 5 and 6 for SOLID and SPH option

Card 5

	1	2	3	4	5	6	7	8
Variable	SBA	SCA	SCB	XXC	YYC	ZZC		
Type	F	F	F	F	F	F		

Card 6

Variable	XXT	YYT	ZZT					
Type	F	F	F					

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Density
EA	E_a, Young's modulus - longitudinal direction
EB	E_b, Young's modulus - transverse direction
EC	E_c, Young's modulus - normal direction
PRBA	ν_{ba} Poisson's ratio ba
PRCA	ν_{ca} Poisson's ratio ca
PRCB	ν_{cb} Poisson's ratio cb
GAB	G_{ab} Shear Modulus
GBC	G_{bc} Shear Modulus
GCA	G_{ca} Shear Modulus
KF	Bulk modulus of failed material
AOPT	Material axes option (see MAT_*OPTION* TROPIC_ELASTIC for a more complete description): EQ.0.0: locally orthotropic with material axes determined by element nodes 1, 2, and 4, as with *DEFINE_COORDINATE_NODES. EQ.1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center; this is the a-direction. This option is for solid elements only. EQ.2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR. EQ.3.0: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle, BETA, from a line in the plane of the element defined by the cross product of the vector **v** with the element normal. EQ.4.0: locally orthotropic in cylindrical coordinate system with the material axes determined by a vector **v**, and an originating point, P, which define the centerline axis. This option is for solid elements only.

VARIABLE	DESCRIPTION
	LT.0.0: the absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_NODES, *DEFINE_COORDINATE_SYSTEM or *DEFINE_COORDINATE_VECTOR). Available in R3 version of 971 and later.
XP YP ZP	Define coordinates of point **p** for AOPT = 1 and 4.
A1 A2 A3	Define components of vector **a** for AOPT = 2.
V1 V2 V3	Define components of vector **v** for AOPT = 3 and 4.
D1 D2 D3	Define components of vector **d** for AOPT = 2:
BETA	Material angle in degrees for AOPT=3, may be overridden on the element card, see *ELEMENT_SHELL_BETA or *ELEMENT_SOLID_ORTHO.
MACF	Material axes change flag for brick elements. EQ.1: No change, default, EQ.2: switch material axes a and b, EQ.3: switch material axes a and c, EQ.4: switch material axes b and c.
BETA	Material angle in degrees for AOPT = 3, may be overridden on the element card, see *ELEMENT_SHELL_BETA.
TSIZE	Time step for automatic element deletion
ALP	Nonlinear shear stress parameter
SOFT	Softening reduction factor for strength in crush
FBRT	Softening of fiber tensile strength
SR	s_r, reduction factor (default=0.447)
SF	s_f, softening factor (default=0.0)
XC	Longitudinal compressive strength, a-axis (positive value).
XT	Longitudinal tensile strength, a-axis
YC	Transverse compressive strength, b-axis (positive value).
YT	Transverse tensile strength, b-axis

VARIABLE	DESCRIPTION
SC	Shear strength, ab plane: GT.0.0 faceted failure surface theory, LT.0.0 ellipsoidal failure surface theory.
SBA	In plane shear strength.
SCA	Transverse shear strength.
SCB	Transverse shear strength.
XXC	Longitudinal compressive strength a-axis (positive value).
YYC	Transverse compressive strength b-axis (positive value).
ZZC	Normal compressive strength c-axis (positive value).
XXT	Longitudinal tensile strength a-axis.
YYT	Transverse tensile strength b-axis.
ZZT	Normal tensile strength c-axis.

*MAT_ELASTIC_WITH_VISCOSITY

This is Material Type 60 which was developed to simulate forming of glass products (e.g., car windshields) at high temperatures. Deformation is by viscous flow but elastic deformations can also be large. The material model, in which the viscosity may vary with temperature, is suitable for treating a wide range of viscous flow problems and is implemented for brick and shell elements.

Card 1

Variable	MID	RO	V0	A	B	C	LCID	
Type	A8	F	F	F	F	F	F	

Card 2

Variable	PR1	PR2	PR3	PR4	PR5	PR6	PR7	PR8
Type	F	F	F	F	F	F	F	F

Card 3

Variable	T1	T2	T3	T4	T5	T6	T7	T8
Type	F	F	F	F	F	F	F	F

Card 4

Variable	V1	V2	V3	V4	V5	V6	V7	V8
Type	F	F	F	F	F	F	F	F

Card 5	1	2	3	4	5	6	7	8
Variable	E1	E2	E3	E4	E5	E6	E7	E8
Type	F	F	F	F	F	F	F	F

Card 6

Variable	ALPHA1	ALPHA2	ALPHA3	ALPHA4	ALPHA5	ALPHA6	ALPHA7	ALPHA8
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
V0	Temperature independent viscosity coefficient, V_0. If defined, the temperature dependent viscosity defined below is skipped, see type (i) and (ii) definitions for viscosity below.
A	Viscosity coefficient, see type (i) and (ii) definitions for viscosity below.
B	Viscosity coefficient, see type (i) and (ii) definitions for viscosity below.
C	Viscosity coefficient, see type (i) and (ii) definitions for viscosity below.
LCID	Load curve (see *DEFINE_CURVE) defining viscosity versus temperature, see type (iii). (Optional)
T1, T2,...TN	Temperatures, define up to 8 values
PR1, PR2,...PRN	Poisson's ratios for the temperatures T_i
V1, V2,...VN	Corresponding viscosity coefficients (define only one if not varying with temperature)
E1, E2,...EN	Corresponding Young's moduli coefficients (define only one if not varying with temperature)
ALPHA....	Corresponding thermal expansion coefficients

Remarks:

Volumetric behavior is treated as linear elastic. The deviatoric strain rate is considered to be the sum of elastic and viscous strain rates:

$$\underset{\sim\ total}{\dot{\varepsilon}'} = \underset{\sim\ elastic}{\dot{\varepsilon}'} + \underset{\sim\ viscous}{\dot{\varepsilon}'} = \frac{\underset{\sim}{\sigma}'}{2G} + \frac{\underset{\sim}{\sigma}'}{2v}$$

where G is the elastic shear modulus, v is the viscosity coefficient, and ~ indicates a tensor. The stress increment over one timestep dt is

$$d\underset{\sim}{\sigma}' = 2G\underset{\sim total}{\dot{\varepsilon}'}dt - \frac{G}{v}dt\underset{\sim}{\sigma}'$$

The stress before the update is used for $\underset{\sim}{\sigma}'$. For shell elements the through-thickness strain rate is calculated as follows.

$$d\sigma_{33} = 0 = K\left(\dot{\varepsilon}_{11} + \dot{\varepsilon}_{22} + \dot{\varepsilon}_{33}\right)dt + 2G\,\dot{\varepsilon}'_{33}\,dt - \frac{G}{v}dt\sigma'_{33}$$

where the subscript ij = 33 denotes the through-thickness direction and K is the elastic bulk modulus. This leads to:

$$\dot{\varepsilon}_{33} = -a\left(\dot{\varepsilon}_{11} + \dot{\varepsilon}_{22}\right) + bp$$

$$a = \frac{\left(K - \frac{2}{3}G\right)}{\left(K + \frac{4}{3}G\right)}$$

$$b = \frac{Gdt}{v\left(K + \frac{4}{3}G\right)}$$

in which p is the pressure defined as the negative of the hydrostatic stress.

The variation of viscosity with temperature can be defined in any one of the 3 ways.

(i) Constant, V = V_0 Do not define constants, A, B, and C or the piecewise curve.(leave card 4 blank)

(ii) V = $V_0 \times 10^{**}$ (A/(T-B) + C)

(iii) Piecewise curve: define the variation of viscosity with temperature.

Note: Viscosity is inactive during dynamic relaxation.

*MAT_ELASTIC_WITH_VISCOSITY_CURVE

This is Material Type 60 which was developed to simulate forming of glass products (e.g., car windshields) at high temperatures. Deformation is by viscous flow but elastic deformations can also be large. The material model, in which the viscosity may vary with temperature, is suitable for treating a wide range of viscous flow problems and is implemented for brick and shell elements. Load curves are used to represent the temperature dependence of Poisson's ratio, Young's modulus, the coefficient of expansion, and the viscosity.

Card 1

1	2	3	4	5	6	7	8
MID	RO	V0	A	B	C	LCID	
A8	F	F	F	F	F	F	

Card 2

1	2	3	4	5	6	7	8
PR_LC	YM_LC	A_LC	V_LC	V_LOG			
F	F	F	F	F			

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
V0	Temperature independent viscosity coefficient, V_0. If defined, the temperature dependent viscosity defined below is skipped, see type (i) and (ii) definitions for viscosity below.
A	Viscosity coefficient, see type (i) and (ii) definitions for viscosity below.
B	Viscosity coefficient, see type (i) and (ii) definitions for viscosity below.
C	Viscosity coefficient, see type (i) and (ii) definitions for viscosity below.
LCID	Load curve (see *DEFINE_CURVE) defining factor on viscosity versus temperature, see type (iii). (Optional).
PR_LC	Load curve (see *DEFINE_CURVE) defining Poisson's ratio as a function of temperature.

VARIABLE	DESCRIPTION
YM_LC	Load curve (see *DEFINE_CURVE) defining Young's modulus as a function of temperature.
A_LC	Load curve (see *DEFINE_CURVE) defining the coefficient of thermal expansion as a function of temperature.
V_LC	Load curve (see *DEFINE_CURVE) defining the viscosity as a function of temperature.
V_LOG	Flag for the form of V_LC. If V_LOG=1.0, the value specified in V_LC is the natural logarithm of the viscosity, ln(V). The value interpolated from the curve is then exponentiated to obtain the viscosity. If V_LOG=0.0, the value is the viscosity. The logarithmic form is useful if the value of the viscosity changes by orders of magnitude over the temperature range of the data.

Remarks:

Volumetric behavior is treated as linear elastic. The deviatoric strain rate is considered to be the sum of elastic and viscous strain rates:

$$\underset{\sim\ total}{\dot{\varepsilon}'} = \underset{\sim\ elastic}{\dot{\varepsilon}'} + \underset{\sim\ viscous}{\dot{\varepsilon}'} = \frac{\underset{\sim}{\sigma}'}{2G} + \frac{\underset{\sim}{\sigma}'}{2v}$$

where G is the elastic shear modulus, v is the viscosity coefficient, and ~ indicates a tensor. The stress increment over one timestep dt is

$$d\underset{\sim}{\sigma}' = 2G\underset{\sim total}{\dot{\varepsilon}'}dt - \frac{G}{v}dt\underset{\sim}{\sigma}'$$

The stress before the update is used for $\underset{\sim}{\sigma}'$. For shell elements the through-thickness strain rate is calculated as follows.

$$d\sigma_{33} = 0 = K\left(\dot{\varepsilon}_{11} + \dot{\varepsilon}_{22} + \dot{\varepsilon}_{33}\right)dt + 2G\,\dot{\varepsilon}'_{33}\,dt - \frac{G}{v}dt\sigma'_{33}$$

where the subscript ij = 33 denotes the through-thickness direction and K is the elastic bulk modulus. This leads to:

$$\dot{\varepsilon}_{33} = -a\left(\dot{\varepsilon}_{11} + \dot{\varepsilon}_{22}\right) + bp$$

$$a = \frac{\left(K - \frac{2}{3}G\right)}{\left(K + \frac{4}{3}G\right)}$$

$$b = \frac{Gdt}{v\left(K + \frac{4}{3}G\right)}$$

in which p is the pressure defined as the negative of the hydrostatic stress.

The variation of viscosity with temperature can be defined in any one of the 3 ways.

(i) Constant, $V = V_0$ Do not define constants, A, B, and C or the piecewise curve.(leave card 4 blank)

(ii) $V = V_0 \times 10^{**} (A/(T-B) + C)$

(iii) Piecewise curve: define the variation of viscosity with temperature.

Note: Viscosity is inactive during dynamic relaxation.

*MAT_KELVIN-MAXWELL_VISCOELASTIC

This is Material Type 61. This material is a classical Kelvin-Maxwell model for modeling viscoelastic bodies, e.g., foams. This model is valid for solid elements only. See also notes below.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	BULK	G0	GI	DC	FO	SO
Type	A8	F	F	F	F	F	F	F
Default	none	none	none	none	none	0.0	0.0	0.0

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
BULK	Bulk modulus (elastic)
G0	Short-time shear modulus, G_0
GI	Long-time (infinite) shear modulus, G_∞
DC	Maxwell decay constant, β[FO=0.0] or Kelvin relaxation constant, τ [FO=1.0]
FO	Formulation option: EQ.0.0: Maxwell, EQ.1.0: Kelvin.
SO	Strain (logarithmic) output option to control what is written as component 7 to the d3plot database. (LS-Prepost always blindly labels this component as effective plastic strain.) The maximum values are updated for each element each time step: EQ.0.0: maximum principal strain that occurs during the calculation, EQ.1.0: maximum magnitude of the principal strain values that occurs during the calculation, EQ.2.0: maximum effective strain that occurs during the calculation.

Remarks:

The shear relaxation behavior is described for the Maxwell model by:

$$G(t) = G + (G_0 - G_\infty)e^{-\beta t}$$

A Jaumann rate formulation is used

$$\overset{\nabla}{\sigma}'_{ij} = 2\int_0^t G(t-\tau)\, D'_{ij}(\tau)dt$$

where the prime denotes the deviatoric part of the stress rate, $\overset{\nabla}{\sigma}_{ij}$, and the strain rate D_{ij}. For the Kelvin model the stress evolution equation is defined as:

$$\dot{s}_{ij} + \frac{1}{\tau}s_{ij} = \left(1+\delta_{ij}\right)G_0\,\dot{e}_{ij} + \left(1+\delta_{ij}\right)\frac{G_\infty}{\tau}\dot{e}_{ij}$$

The strain data as written to the LS-DYNA database may be used to predict damage, see [Bandak 1991].

***MAT_VISCOUS_FOAM**

This is Material Type 62. It was written to represent the Confor Foam on the ribs of EuroSID side impact dummy. It is only valid for solid elements, mainly under compressive loading.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E1	N1	V2	E2	N2	PR
Type	A8	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
E1	Initial Young's modulus (E_1)
N1	Exponent in power law for Young's modulus (n_1)
V2	Viscous coefficient (V_2)
E2	Elastic modulus for viscosity (E_2), see notes below.
N2	Exponent in power law for viscosity (n_2)
PR	Poisson's ratio, ν

Remarks:

The model consists of a nonlinear elastic stiffness in parallel with a viscous damper. The elastic stiffness is intended to limit total crush while the viscosity absorbs energy. The stiffness E_2 exists to prevent timestep problems. It is used for time step calculations a long as E_1^t is smaller than E_2. It has to be carefully chosen to take into account the stiffening effects of the viscosity. Both E_1 and V_2 are nonlinear with crush as follows:

$$E_1^t = E_1\left(V^{-n_1}\right)$$

$$V_2^t = V_2\left(abs\left(1-V\right)\right)^{n_2}$$

where viscosity generates a shear stress given by

$$\tau = V_2 \dot{\gamma}$$

$\dot{\gamma}$ is the engineering shear strain rate, and V is the relative volume defined by the ratio of the current to initial volume. Typical values are (units of N, mm, s)

E_1=0.0036	n_1=4.0	V_2=0.0015
E_2=100.0	n_2=0.2	ν=0.05

***MAT_CRUSHABLE_FOAM**

This is Material Type 63 which is dedicated to modeling crushable foam with optional damping and tension cutoff. Unloading is fully elastic. Tension is treated as elastic-perfectly-plastic at the tension cut-off value. A modified version of this model, *MAT_MODIFIED_ CRUSHABLE_FOAM includes strain rate effects.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	LCID	TSC	DAMP	
Type	A8	F	F	F	F	F	F	
Default	none	none	none	none	none	0.0	0.10	

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
E	Young's modulus
PR	Poisson's ratio
LCID	Load curve ID defining yield stress versus volumetric strain, γ, see Figure 63.1.
TSC	Tensile stress cutoff. A nonzero, positive value is strongly recommended for realistic behavior.
DAMP	Rate sensitivity via damping coefficient (.05<recommended value<.50).

Remarks:

The volumetric strain is defined in terms of the relative volume, V, as:

$$\gamma = 1.-V$$

The relative volume is defined as the ratio of the current to the initial volume. In place of the effective plastic strain in the D3PLOT database, the integrated volumetric strain is output.

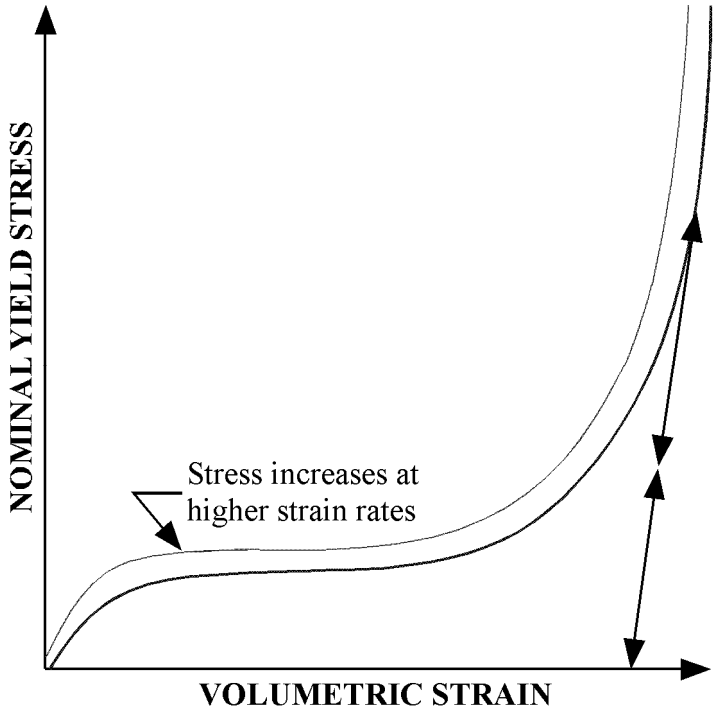


Figure 63.1. Behavior of strain rate sensitive crushable foam. Unloading is elastic to the tension cutoff. Subsequent reloading follows the unloading curve.

*MAT_RATE_SENSITIVE_POWERLAW_PLASTICITY

This is Material Type 64 which will model strain rate sensitive elasto-plastic material with a power law hardening. Optionally, the coefficients can be defined as functions of the effective plastic strain.

Card 1

Variable	1	2	3	4	5	6	7	8
	MID	RO	E	PR	K	M	N	E0
Type	A8	F	F	F	F	F	F	F
Default	---	---	---	---	---	0.0001	---	0.0002

Card 2

Variable	VP	EPS0						
Type	F	F						
Default	0.0	1.0						

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
E	Young's modulus of elasticity
PR	Poisson's ratio
K	Material constant, k. If k<0 the absolute value of k is taken as the load curve number that defines k as a function of effective plastic strain.
M	Strain hardening coefficient, m. If m<0 the absolute value of m is taken as the load curve number that defines m as a function of effective plastic strain.

VARIABLE	DESCRIPTION
N	Strain rate sensitivity coefficient, n. If n<0 the absolute value of n is taken as the load curve number that defines n as a function of effective plastic strain.
E0	Initial strain rate (default = 0.0002)
VP	Formulation for rate effects: EQ.0.0: Scale yield stress (default) EQ.1.0: Viscoplastic formulation
EPS0	Factor to normalize strain rate EQ.1.0: Time units of seconds (default) EQ.1.E-3: Time units of milliseconds EQ.1.E-6: Time units of microseconds

Remarks:

This material model follows a constitutive relationship of the form:

$$\sigma = k\varepsilon^m \dot{\varepsilon}^n$$

where σ is the yield stress, ε is the effective plastic strain, $\dot{\varepsilon}$ is the normalized effective plastic strain rate, and the constants *k*, *m*, and *n* can be expressed as functions of effective plastic strain or can be constant with respect to the plastic strain. The case of no strain hardening can be obtained by setting the exponent of the plastic strain equal to a very small positive value, i.e. 0.0001.

This model can be combined with the superplastic forming input to control the magnitude of the pressure in the pressure boundary conditions in order to limit the effective plastic strain rate so that it does not exceed a maximum value at any integration point within the model.

A fully viscoplastic formulation is optional. An additional cost is incurred but the improvement is results can be dramatic.

*MAT_MODIFIED_ZERILLI_ARMSTRONG

This is Material Type 65 which is a rate and temperature sensitive plasticity model which is sometimes preferred in ordnance design calculations.

Card 1

	1	2	3	4	5	6	7	8
Variable	MID	RO	G	E0	N	TROOM	PC	SPALL
Type	A8	F	F	F	F	F	F	F

Card 2

Variable	C1	C2	C3	C4	C5	C6	EFAIL	VP
Type	F	F	F	F	F	F	F	F

Card 3

Variable	B1	B2	B3	G1	G2	G3	G4	BULK
Type	F	F	F	F	F	F	F	

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
G	Shear modulus
E0	$\dot{\varepsilon}_0$, factor to normalize strain rate
N	n, exponent for bcc metal
TROOM	T_r, room temperature
PC	*pc*, Pressure cutoff

VARIABLE	DESCRIPTION
SPALL	Spall Type: EQ.1.0: minimum pressure limit, EQ.2.0: maximum principal stress, EQ.3.0: minimum pressure cutoff.
C1	C_1, coefficients for flow stress, see notes below.
C2	C_2, coefficients for flow stress, see notes below.
C3	C_3, coefficients for flow stress, see notes below.
C4	C_4, coefficients for flow stress, see notes below.
C5	C_5, coefficients for flow stress, see notes below.
C6	C_6, coefficients for flow stress, see notes below.
EFAIL	Failure strain for erosion
VP	Formulation for rate effects: EQ.0.0: Scale yield stress (default) EQ.1.0: Viscoplastic formulation
B1	B_1, coefficients for polynomial to represent temperature dependency of flow stress yield.
B2	B_2
B3	B_3
G1	G_1, coefficients for defining heat capacity and temperature dependency of heat capacity.
G2	G_2
G3	G_3
G4	G_4
BULK	Bulk modulus defined for shell elements only. Do not input for solid elements.

Remarks:

The Armstrong-Zerilli Material Model expresses the flow stress as follows.

For fcc metals (n=0),

$$\sigma = C_1 + \left\{ C_2 \left(\varepsilon^p\right)^{1/2} \left[e^{\left(-C_3 + C_4 \ln\left(\dot{\varepsilon}^*\right)\right)T} \right] + C_5 \right\} \left(\frac{\mu(T)}{\mu(293)} \right)$$

ε^p = effective plastic strain

$\dot{\varepsilon}^* = \dfrac{\dot{\varepsilon}}{\dot{\varepsilon}_0}$ effective plastic strain rate where $\dot{\varepsilon}_0$ = 1, 1e-3, 1e-6 for time units of seconds, milliseconds, and microseconds, respectively.

For bcc metals (n>0),

$$\sigma = C_1 + C_2 e^{\left(-C_3 + C_4 \ln\left(\dot{\varepsilon}^*\right)\right)T} + \left[C_5 \left(\varepsilon^p\right)^n + C_6 \right] \left(\frac{\mu(T)}{\mu(293)} \right)$$

where

$$\left(\frac{\mu(T)}{\mu(293)} \right) = B_1 + B_2 T + B_3 T^2 .$$

The relationship between heat capacity (specific heat) and temperature may be characterized by a cubic polynomial equation as follows:

$$C_p = G_1 + G_2 T + G_3 T^2 + G_4 T^3$$

A fully viscoplastic formulation is optional. An additional cost is incurred but the improvement is results can be dramatic.

*MAT_LINEAR_ELASTIC_DISCRETE_BEAM

This is Material Type 66. This material model is defined for simulating the effects of a linear elastic beam by using six springs each acting about one of the six local degrees-of-freedom. The two nodes defining a beam may be coincident to give a zero length beam, or offset to give a finite length beam. For finite length discrete beams the absolute value of the variable SCOOR in the SECTION_BEAM input should be set to a value of 2.0, which causes the local r-axis to be aligned along the two nodes of the beam to give physically correct behavior. The distance between the nodes of a beam should not affect the behavior of this model. A triad is used to orient the beam for the directional springs. Translational/rotational stiffness and viscous damping effects are considered for a local cartesian system, see notes below. Applications for this element include the modeling of joint stiffnesses.

Card 1

1	2	3	4	5	6	7	8
Variable	MID	RO	TKR	TKS	TKT	RKR	RKS	RKT
Type	A8	F	F	F	F	F	F	F

Card 2

Variable	TDR	TDS	TDT	RDR	RDS	RDT		
Type	F	F	F	F	F	F		

Card 3

Variable	FOR	FOS	FOT	MOR	MOS	MOT		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density, see also “volume” in the *SECTION_BEAM definition.
TKR	Translational stiffness about local r-axis, see notes below.

VARIABLE	DESCRIPTION
TKS	Translational stiffness about local s-axis.
TKT	Translational stiffness about local t-axis.
RKR	Rotational stiffness about the local r-axis.
RKS	Rotational stiffness about the local s-axis.
RKT	Rotational stiffness about the local t-axis.
TDR	Translational viscous damper about local r-axis. (Optional)
TDS	Translational viscous damper about local s-axis. (Optional)
TDT	Translational viscous damper about local t-axis. (Optional)
RDR	Rotational viscous damper about the local r-axis. (Optional)
RDS	Rotational viscous damper about the local s-axis. (Optional)
RDT	Rotational viscous damper about the local t-axis. (Optional)
FOR	Preload force in r-direction. (Optional)
FOS	Preload force in s-direction. (Optional)
FOT	Preload force in t-direction. (Optional)
MOR	Preload moment about r-axis. (Optional)
MOS	Preload moment about s-axis. (Optional)
MOT	Preload moment about t-axis. (Optional)

Remarks:

The formulation of the discrete beam (type 6) assumes that the beam is of zero length and requires no orientation node. A small distance between the nodes joined by the beam is permitted. The local coordinate system which determines (r,s,t) is given by the coordinate ID, see *DEFINE_COORDINATE_*OPTION*, in the cross sectional input, see *SECTION_BEAM, where the global system is the default. The local coordinate system axes can rotate with either node of the beam or an average rotation of both nodes (see SCOOR variable in *SECTION_BEAM).

For null stiffness coefficients, no forces corresponding to these null values will develop. The viscous damping coefficients are optional.

*MAT_NONLINEAR_ELASTIC_DISCRETE_BEAM

This is Material Type 67. This material model is defined for simulating the effects of nonlinear elastic and nonlinear viscous beams by using six springs each acting about one of the six local degrees-of-freedom. The two nodes defining a beam may be coincident to give a zero length beam, or offset to give a finite length beam. For finite length discrete beams the absolute value of the variable SCOOR in the SECTION_BEAM input should be set to a value of 2.0, which causes the local r-axis to be aligned along the two nodes of the beam to give physically correct behavior. The distance between the nodes of a beam should not affect the behavior of this material model. A triad is used to orient the beam for the directional springs. Arbitrary curves to model transitional/ rotational stiffness and damping effects are allowed. See notes below.

Card 1

1	2	3	4	5	6	7	8
Variable	MID	RO	LCIDTR	LCIDTS	LCIDTT	LCIDRR	LCIDRS	LCIDRT
Type	A8	F	F	F	F	F	F	F

Card 2

Variable	LCIDTDR	LCIDTDS	LCIDTDT	LCIDRDR	LCIDRDS	LCIDRDT		
Type	F	F	F	F	F	F		

Card 3

Variable	FOR	FOS	FOT	MOR	MOS	MOT		
Type	F	F	F	F	F	F		

Cards 4 and 5 must be defined to consider failure; otherwise, they are optional.

Card 4

	1	2	3	4	5	6	7	8
Variable	FFAILR	FFAILS	FFAILT	MFAILR	MFAILS	MFAILT		
Type	F	F	F	F	F	F		
Default	0.0	0.0	0.0	0.0	0.0	0.0		

Card 5

	1	2	3	4	5	6	7	8
Variable	UFAILR	UFAILS	UFAILT	TFAILR	TFAILS	TFAILT		
Type	F	F	F	F	F	F		
Default	0.0	0.0	0.0	0.0	0.0	0.0		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density, see also volume in *SECTION_BEAM definition.
LCIDTR	Load curve ID defining translational force resultant along local r-axis versus relative translational displacement, see Remarks and Figure 67.1.
LCIDTS	Load curve ID defining translational force resultant along local s-axis versus relative translational displacement.
LCIDTT	Load curve ID defining translational force resultant along local t-axis versus relative translational displacement.
LCIDRR	Load curve ID defining rotational moment resultant about local r-axis versus relative rotational displacement.
LCIDRS	Load curve ID defining rotational moment resultant about local s-axis versus relative rotational displacement.
LCIDRT	Load curve ID defining rotational moment resultant about local t-axis versus relative rotational displacement.

VARIABLE	DESCRIPTION
LCIDTDR	Load curve ID defining translational damping force resultant along local r-axis versus relative translational velocity.
LCIDTDS	Load curve ID defining translational damping force resultant along local s-axis versus relative translational velocity.
LCIDTDT	Load curve ID defining translational damping force resultant along local t-axis versus relative translational velocity.
LCIDRDR	Load curve ID defining rotational damping moment resultant about local r-axis versus relative rotational velocity.
LCIDRDS	Load curve ID defining rotational damping moment resultant about local s-axis versus relative rotational velocity.
LCIDRDT	Load curve ID defining rotational damping moment resultant about local t-axis versus relative rotational velocity.
FOR	Preload force in r-direction. (Optional)
FOS	Preload force in s-direction. (Optional)
FOT	Preload force in t-direction. (Optional)
MOR	Preload moment about r-axis. (Optional)
MOS	Preload moment about s-axis. (Optional)
MOT	Preload moment about t-axis. (Optional)
FFAILR	Optional failure parameter. If zero, the corresponding force, F_r, is not considered in the failure calculation.
FFAILS	Optional failure parameter. If zero, the corresponding force, F_s, is not considered in the failure calculation.
FFAILT	Optional failure parameter. If zero, the corresponding force, F_t, is not considered in the failure calculation.
MFAILR	Optional failure parameter. If zero, the corresponding moment, M_r, is not considered in the failure calculation.
MFAILS	Optional failure parameter. If zero, the corresponding moment, M_s, is not considered in the failure calculation.
MFAILT	Optional failure parameter. If zero, the corresponding moment, M_t, is not considered in the failure calculation.

VARIABLE	DESCRIPTION
UFAILR	Optional failure parameter. If zero, the corresponding displacement, u_r, is not considered in the failure calculation.
UFAILS	Optional failure parameter. If zero, the corresponding displacement, u_s, is not considered in the failure calculation.
UFAILT	Optional failure parameter. If zero, the corresponding displacement, u_t, is not considered in the failure calculation.
TFAILR	Optional failure parameter. If zero, the corresponding rotation, θ_r, is not considered in the failure calculation.
TFAILS	Optional failure parameter. If zero, the corresponding rotation, θ_s, is not considered in the failure calculation.
TFAILT	Optional failure parameter. If zero, the corresponding rotation, θ_t, is not considered in the failure calculation.

Remarks:

For null load curve ID's, no forces are computed.

The formulation of the discrete beam (type 6) assumes that the beam is of zero length and requires no orientation node. A small distance between the nodes joined by the beam is permitted. The local coordinate system which determines (r,s,t) is given by the coordinate ID, see *DEFINE_ COORDINATE_*OPTION*, in the cross sectional input, see *SECTION_BEAM, where the global system is the default. The local coordinate system axes can rotate with either node of the beam or an average rotation of both nodes (see SCOOR variable in *SECTION_BEAM).

If different behavior in tension and compression is desired in the calculation of the force resultants, the load curve(s) must be defined in the negative quadrant starting with the most negative displacement then increasing monotonically to the most positive. If the load curve behaves similarly in tension and compression, define only the positive quadrant. Whenever displacement values fall outside of the defined range, the resultant forces will be extrapolated. Figure 67.1 depicts a typical load curve for a force resultant. Load curves used for determining the damping forces and moment resultants always act identically in tension and compression, since only the positive quadrant values are considered, i.e., start the load curve at the origin [0,0].

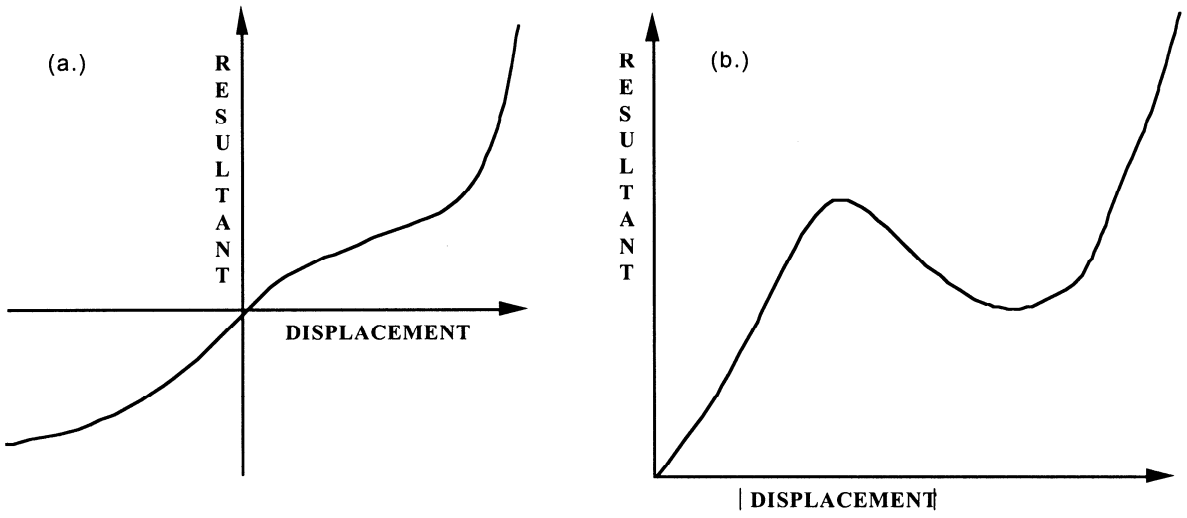


Figure 67.1. The resultant forces and moments are determined by a table lookup. If the origin of the load curve is at [0,0] as in (b.) and tension and compression responses are symmetric.

Catastrophic failure based on force resultants occurs if the following inequality is satisfied.

$$\left(\frac{F_r}{F_r^{fail}}\right)^2+\left(\frac{F_s}{F_s^{fail}}\right)^2+\left(\frac{F_t}{F_t^{fail}}\right)^2+\left(\frac{M_r}{M_r^{fail}}\right)^2+\left(\frac{M_s}{M_s^{fail}}\right)^2+\left(\frac{M_t}{M_t^{fail}}\right)^2-1.\geq 0.$$

After failure the discrete element is deleted. Likewise, catastrophic failure based on displacement resultants occurs if the following inequality is satisfied:

$$\left(\frac{u_r}{u_r^{fail}}\right)^2+\left(\frac{u_s}{u_s^{fail}}\right)^2+\left(\frac{u_t}{u_t^{fail}}\right)^2+\left(\frac{\theta_r}{\theta_r^{fail}}\right)^2+\left(\frac{\theta_s}{\theta_s^{fail}}\right)^2+\left(\frac{\theta_t}{\theta_t^{fail}}\right)^2-1.\geq 0.$$

After failure the discrete element is deleted. If failure is included either one or both of the criteria may be used.

*MAT_NONLINEAR_PLASTIC_DISCRETE_BEAM

This is Material Type 68. This material model is defined for simulating the effects of nonlinear elastoplastic, linear viscous behavior of beams by using six springs each acting about one of the six local degrees-of-freedom. The two nodes defining a beam may be coincident to give a zero length beam, or offset to give a finite length beam. For finite length discrete beams the absolute value of the variable SCOOR in the SECTION_BEAM input should be set to a value of 2.0, which causes the local r-axis to be aligned along the two nodes of the beam to give physically correct behavior. The distance between the nodes of a beam should not affect the behavior of this material model. A triad is used to orient the beam for the directional springs. Translational/rotational stiffness and damping effects can be considered. The plastic behavior is modeled using force/moment curves versus displacements/ rotation. Optionally, failure can be specified based on a force/moment criterion and a displacement/ rotation criterion. See also notes below.

Card 1

1	2	3	4	5	6	7	8
MID	RO	TKR	TKS	TKT	RKR	RKS	RKT
A8	F	F	F	F	F	F	F
none	none	none	none	none	none	none	none

Card 2

TDR	TDS	TDT	RDR	RDS	RDT		
F	F	F	F	F	F		
none	none	none	none	none	none		

Card 3

Variable	1	2	3	4	5	6	7	8
	LCPDR	LCPDS	LCPDT	LCPMR	LCPMS	LCPMT		
Type	F	F	F	F	F	F		
Default	0.0	0.0	0.0	0.0	0.0	0.0		

Card 4

Variable	1	2	3	4	5	6	7	8
	FFAILR	FFAILS	FFAILT	MFAILR	MFAILS	MFAILT		
Type	F	F	F	F	F	F		
Default	0.0	0.0	0.0	0.0	0.0	0.0		

Card 5

Variable	1	2	3	4	5	6	7	8
	UFAILR	UFAILS	UFAILT	TFAILR	TFAILS	TFAILT		
Type	F	F	F	F	F	F		
Default	0.0	0.0	0.0	0.0	0.0	0.0		

Card 6

Variable	1	2	3	4	5	6	7	8
	FOR	FOS	FOT	MOR	MOS	MOT		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density, see also volume on *SECTION_BEAM definition.

VARIABLE	DESCRIPTION
TKR	Translational stiffness about local r-axis
TKS	Translational stiffness about local s-axis
TKT	Translational stiffness about local t-axis
RKR	Rotational stiffness about the local r-axis
RKS	Rotational stiffness about the local s-axis
RKT	Rotational stiffness about the local t-axis
TDR	Translational viscous damper about local r-axis
TDS	Translational viscous damper about local s-axis
TDT	Translational viscous damper about local t-axis
RDR	Rotational viscous damper about the local r-axis
RDS	Rotational viscous damper about the local s-axis
RDT	Rotational viscous damper about the local t-axis
LCPDR	Load curve ID-yield force versus plastic displacement r-axis. If the curve ID is zero, and if TKR is nonzero, then elastic behavior is obtained for this component.
LCPDS	Load curve ID-yield force versus plastic displacement s-axis. If the curve ID is zero, and if TKS is nonzero, then elastic behavior is obtained for this component.
LCPDT	Load curve ID-yield force versus plastic displacement t-axis. If the curve ID is zero, and if TKT is nonzero, then elastic behavior is obtained for this component.
LCPMR	Load curve ID-yield moment versus plastic rotation r-axis. If the curve ID is zero, and if RKR is nonzero, then elastic behavior is obtained for this component.
LCPMS	Load curve ID-yield moment versus plastic rotation s-axis. If the curve ID is zero, and if RKS is nonzero, then elastic behavior is obtained for this component.
LCPMT	Load curve ID-yield moment versus plastic rotation t-axis. If the curve ID is zero, and if RKT is nonzero, then elastic behavior is obtained for this component.

VARIABLE	DESCRIPTION
FFAILR	Optional failure parameter. If zero, the corresponding force, F_r, is not considered in the failure calculation.
FFAILS	Optional failure parameter. If zero, the corresponding force, F_s, is not considered in the failure calculation.
FFAILT	Optional failure parameter. If zero, the corresponding force, F_t, is not considered in the failure calculation.
MFAILR	Optional failure parameter. If zero, the corresponding moment, M_r, is not considered in the failure calculation.
MFAILS	Optional failure parameter. If zero, the corresponding moment, M_s, is not considered in the failure calculation.
MFAILT	Optional failure parameter. If zero, the corresponding moment, M_t, is not considered in the failure calculation.
UFAILR	Optional failure parameter. If zero, the corresponding displacement, u_r, is not considered in the failure calculation.
UFAILS	Optional failure parameter. If zero, the corresponding displacement, u_s, is not considered in the failure calculation.
UFAILT	Optional failure parameter. If zero, the corresponding displacement, u_t, is not considered in the failure calculation.
TFAILR	Optional failure parameter. If zero, the corresponding rotation, θ_r, is not considered in the failure calculation.
TFAILS	Optional failure parameter. If zero, the corresponding rotation, θ_s, is not considered in the failure calculation.
TFAILT	Optional failure parameter. If zero, the corresponding rotation, θ_t, is not considered in the failure calculation.
FOR	Preload force in r-direction. (Optional)
FOS	Preload force in s-direction. (Optional)
FOT	Preload force in t-direction. (Optional)
MOR	Preload moment about r-axis. (Optional)
MOS	Preload moment about s-axis. (Optional)
MOT	Preload moment about t-axis. (Optional)

Remarks:

For the translational and rotational degrees of freedom where elastic behavior is desired, set the load curve ID to zero.

The formulation of the discrete beam (type 6) assumes that the beam is of zero length and requires no orientation node. A small distance between the nodes joined by the beam is permitted. The local coordinate system which determines (r,s,t) is given by the coordinate ID (see *DEFINE_COORDINATE_*OPTION*) in the cross sectional input, see *SECTION_BEAM, where the global system is the default. The local coordinate system axes can rotate with either node of the beam or an average rotation of both nodes (see SCOOR variable in *SECTION_BEAM).

Catastrophic failure based on force resultants occurs if the following inequality is satisfied.

$$\left(\frac{F_r}{F_r^{fail}}\right)^2+\left(\frac{F_s}{F_s^{fail}}\right)^2+\left(\frac{F_t}{F_t^{fail}}\right)^2+\left(\frac{M_r}{M_r^{fail}}\right)^2+\left(\frac{M_s}{M_s^{fail}}\right)^2+\left(\frac{M_t}{M_t^{fail}}\right)^2-1.\geq 0.$$

After failure the discrete element is deleted. Likewise, catastrophic failure based on displacement resultants occurs if the following inequality is satisfied:

$$\left(\frac{u_r}{u_r^{fail}}\right)^2+\left(\frac{u_s}{u_s^{fail}}\right)^2+\left(\frac{u_t}{u_t^{fail}}\right)^2+\left(\frac{\theta_r}{\theta_r^{fail}}\right)^2+\left(\frac{\theta_s}{\theta_s^{fail}}\right)^2+\left(\frac{\theta_t}{\theta_t^{fail}}\right)^2-1.\geq 0.$$

After failure the discrete element is deleted. If failure is included either one or both of the criteria may be used.

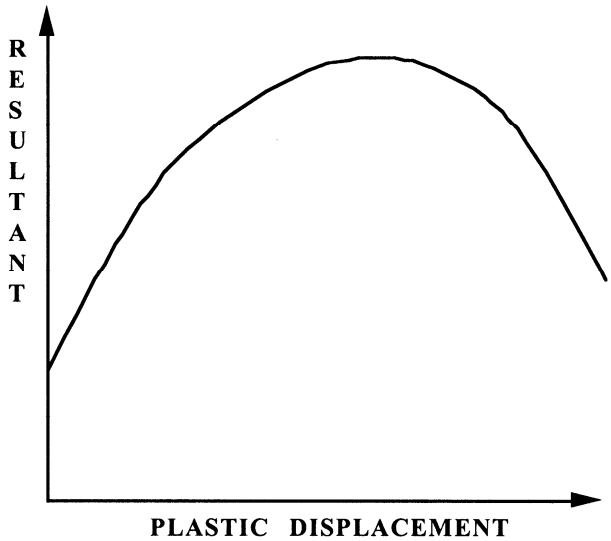


Figure 68.1. The resultant forces and moments are limited by the yield definition. The initial yield point corresponds to a plastic displacement of zero.

*MAT_SID_DAMPER_DISCRETE_BEAM

This is Material Type 69. The side impact dummy uses a damper that is not adequately treated by the nonlinear force versus relative velocity curves since the force characteristics are dependent on the displacement of the piston. See also notes below.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	ST	D	R	H	K	C
Type	A8	F	F	F	F	F	F	F

Card 2

Variable	C3	STF	RHOF	C1	C2	LCIDF	LCIDD	S0
Type	F	F	F	F	F	F	F	F

Read in up to 15 orifice locations with orifice location per card. Input is terminated when a "*" card is found. On the first card below the optional input parameters SF and DF may be specified.

Cards 3...	1	2	3	4	5	6	7	8
Variable	ORFLOC	ORFRAD	SF	DC				
Type	F	F	F	F				

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density, see also volume on *SECTION_BEAM definition.
ST	S_t, piston stroke. S_t must equal or exceed the length of the beam element, see Figure 69.1 below.
D	d, piston diameter

VARIABLE	DESCRIPTION
R	R, default orifice radius
H	h, orifice controller position
K	K, damping constant LT.0.0: |K| is the load curve number ID, see *DEFINE_CURVE, defining the damping coefficient as a function of the absolute value of the relative velocity.
C	C, discharge coefficient
C3	Coefficient for fluid inertia term
STF	k, stiffness coefficient if piston bottoms out
RHOF	ρ_{fluid}, fluid density
C1	C_1, coefficient for linear velocity term
C2	C_2, coefficient for quadratic velocity term
LCIDF	Load curve number ID defining force versus piston displacement, s, i.e., term $f(s+s_0)$. Compressive behavior is defined in the positive quadrant of the force displacement curve. Displacements falling outside of the defined force displacement curve are extrapolated. Care must be taken to ensure that extrapolated values are reasonable.
LCIDD	Load curve number ID defining damping coefficient versus piston displacement, s, i.e., $g(s+s_0)$. Displacements falling outside the defined curve are extrapolated. Care must be taken to ensure that extrapolated values are reasonable.
S0	Initial displacement s_0, typically set to zero. A positive displacement corresponds to compressive behavior.
ORFLOC	d_i, orifice location of ith orifice relative to the fixed end.
ORFRAD	r_i, orifice radius of ith orifice, if zero the default radius is used.
SF	Scale factor on calculated force. The default is set to 1.0
DC	c, linear viscous damping coefficient used after damper bottoms out either in tension or compression.

Remarks:

As the damper moves, the fluid flows through the open orifices to provide the necessary damping resistance. While moving as shown in Figure 69.1 the piston gradually blocks off and effectively closes the orifices. The number of orifices and the size of their opening control the damper resistance and performance. The damping force is computed from,

$$F = SF\left\{KA_pV_p\left\{\frac{C_1}{A_0^t}+C_2\left|V_p\right|\rho_{fluid}\left[\left(\frac{A_p}{CA_0^t}\right)^2-1\right]\right\}-f\left(s+s_0\right)+V_pg\left(s+s_0\right)\right\}$$

where K is a user defined constant or a tabulated function of the absolute value of the relative velocity, V_p is the piston velocity, C is the discharge coefficient, A_p is the piston area, A_0^t is the total open areas of orifices at time t, ρ_{fluid} is the fluid density, C_1 is the coefficient for the linear term, and C_2 is the coefficient for the quadratic term.

In the implementation, the orifices are assumed to be circular with partial covering by the orifice controller. As the piston closes, the closure of the orifice is gradual. This gradual closure is properly taken into account to insure a smooth response. If the piston stroke is exceeded, the stiffness value, k, limits further movement, i.e., if the damper bottoms out in tension or compression the damper forces are calculated by replacing the damper by a bottoming out spring and damper, *k* and *c*, respectively. The piston stroke must exceed the initial length of the beam element. The time step calculation is based in part on the stiffness value of the bottoming out spring. A typical force versus displacement curve at constant relative velocity is shown in Figure 69.2.

The factor, *SF*, which scales the force defaults to 1.0 and is analogous to the adjusting ring on the damper.

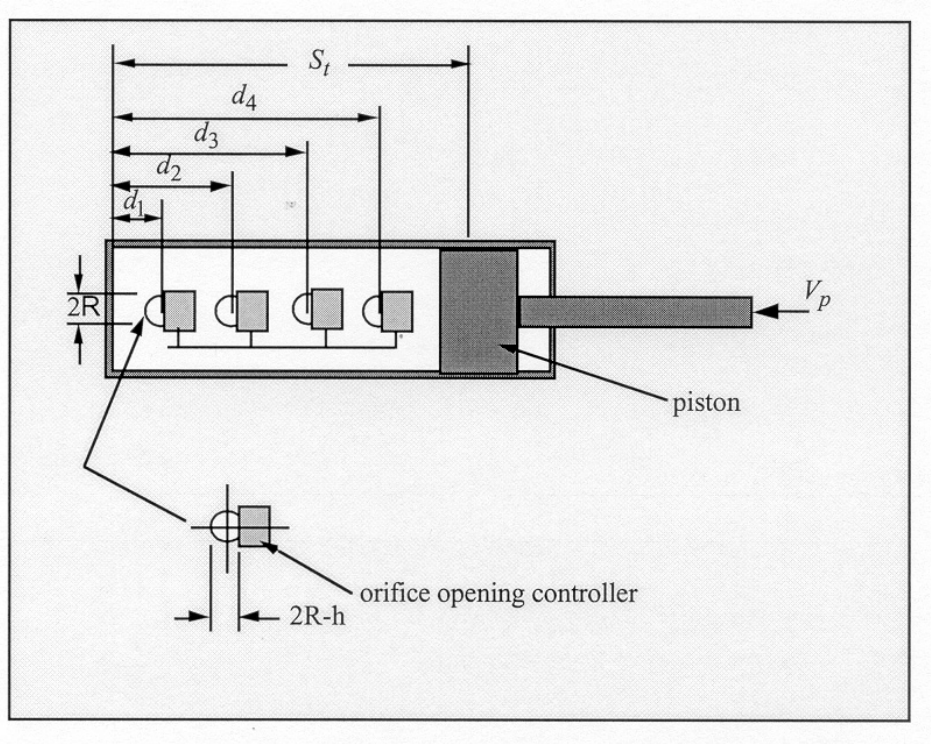


Figure 69.1. Mathematical model for the Side Impact Dummy damper.

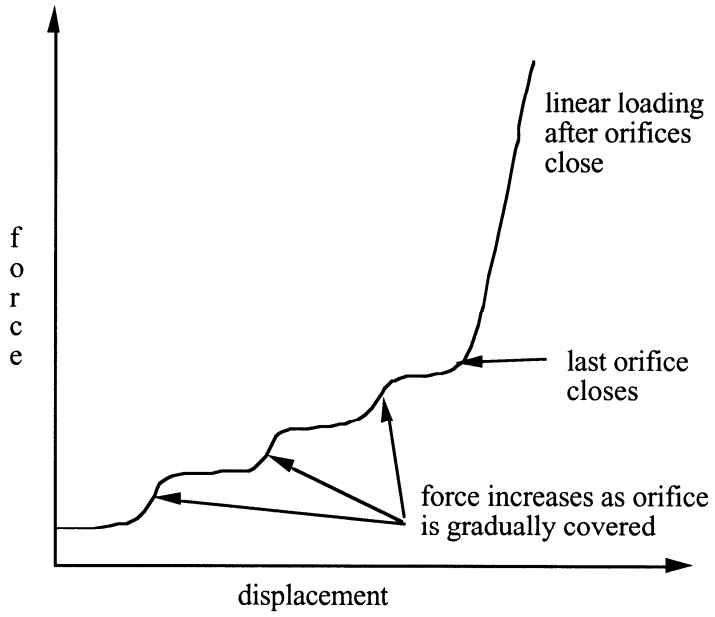


Figure 69.2. Force versus displacement as orifices are covered at a constant relative velocity. Only the linear velocity term is active.

*MAT_HYDRAULIC_GAS_DAMPER_DISCRETE_BEAM

This is Material Type 70. This special purpose element represents a combined hydraulic and gas-filled damper which has a variable orifice coefficient. A schematic of the damper is shown in Figure 70.1. Dampers of this type are sometimes used on buffers at the end of railroad tracks and as aircraft undercarriage shock absorbers. This material can be used only as a discrete beam element. See also notes below.

Card 1

Variable	MID	RO	CO	N	P0	PA	AP	KH
Type	A8	F	F	F	F	F	F	F

Card 2

Variable	LCID	FR	SCLF	CLEAR				
Type	F	F	F	F				

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density, see also volume in *SECTION_BEAM definition.
CO	Length of gas column, C_o
N	Adiabatic constant
P0	Initial gas pressure, P_0
PA	Atmospheric pressure, P_a
AP	Piston cross sectional area, A_p
KH	Hydraulic constant, K
LCID	Load curve ID, see *DEFINE_CURVE, defining the orifice area, a_0, versus element deflection.

VARIABLE	DESCRIPTION
FR	Return factor on orifice force. This acts as a factor on the hydraulic force only and is applied when unloading. It is intended to represent a valve that opens when the piston unloads to relieve hydraulic pressure. Set it to 1.0 for no such relief.
SCLF	Scale factor on force. (Default = 1.0)
CLEAR	Clearance (if nonzero, no tensile force develops for positive displacements and negative forces develop only after the clearance is closed.

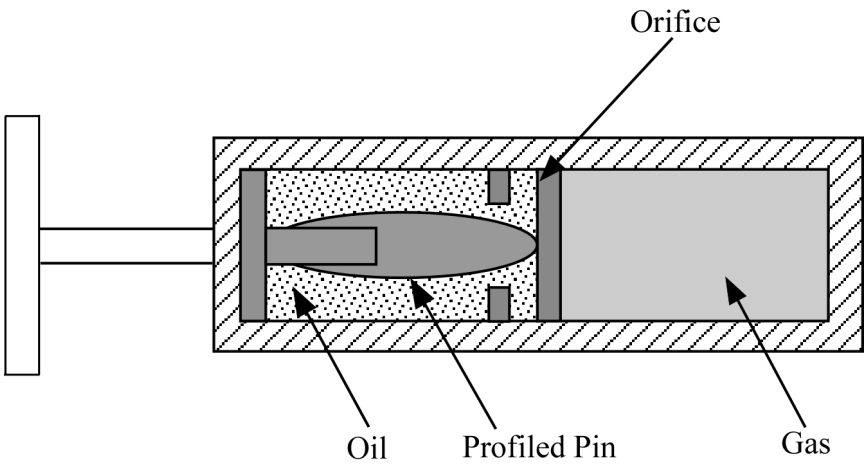


Figure 70.1. Schematic of Hydraulic/Gas damper.

Remarks:

As the damper is compressed two actions contribute to the force which develops. First, the gas is adiabatically compressed into a smaller volume. Secondly, oil is forced through an orifice. A profiled pin may occupy some of the cross-sectional area of the orifice; thus, the orifice area available for the oil varies with the stroke. The force is assumed proportional to the square of the velocity and inversely proportional to the available area.

The equation for this element is:

$$F = SCLF \cdot \left\{ K_h \left(\frac{V}{a_0} \right)^2 + \left[P_0 \left(\frac{C_0}{C_0 - S} \right)^n - P_a \right] \cdot A_p \right\}$$

where S is the element deflection and V is the relative velocity across the element.

*MAT_CABLE_DISCRETE_BEAM

This is Material Type 71. This model permits elastic cables to be realistically modeled; thus, no force will develop in compression.

Note: The following options will be available starting in release 3 of version 971: TMAXF0, TRAMP, IREAD, OUTPUT.

Card 1

1	2	3	4	5	6	7	8
MID	RO	E	LCID	F0	TMAXF0	TRAMP	IREAD
A8	F	F	F	F	F	F	I
none	none	none	none	0	0	0	0

Define Card 2 only if IREAD > 0

Card 2

1	2	3	4	5	6	7	8
OUTPUT							
I							
0							

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density, see also volume in *SECTION_BEAM definition.
E	GT.0.0: Young's modulus LT.0.0: Stiffness
LCID	Load curve ID, see *DEFINE_CURVE, defining the stress versus engineering strain. (Optional).
F0	Initial tensile force. If F0 is defined, an offset is not needed for an initial tensile force.

VARIABLE	DESCRIPTION
TMAXF0	Time for which pre-tension force will be held
TRAMP	Ramp-up time for pre-tension force
OUTPUT	Flag =1 to output axial strain (see note)

Remarks:

The force, *F*, generated by the cable is nonzero if and only if the cable is tension. The force is given by:

$$F = \max\left(F_0 + K\Delta L, 0.\right)$$

where ΔL is the change in length

$$\Delta L = current\ length - \left(initial\ length - offset\right)$$

and the stiffness (E > 0.0 only) is defined as:

$$K = \frac{E \cdot area}{\left(initial\ length - offset\right)}$$

Note that a constant force element can be obtained by setting:

$$F_0 > 0 \ \ and \ \ K = 0$$

although the application of such an element is unknown.

The area and offset are defined on either the cross section or element cards. For a slack cable the offset should be input as a negative length. For an initial tensile force the offset should be positive.

If a load curve is specified the Young's modulus will be ignored and the load curve will be used instead. The points on the load curve are defined as engineering stress versus engineering strain, i.e., the change in length over the initial length. The unloading behavior follows the loading.

By default, cable pretension is applied only at the start of the analysis. If the cable is attached to flexible structure, deformation of the structure will result in relaxation of the cables, which will therefore lose some or all of the intended preload.

This can be overcome by using TMAXF0. In this case, it is expected that the structure will deform under the loading from the cables and that this deformation will take time to occur during the analysis. The unstressed length of the cable will be continuously adjusted until time TMAXF0 such that the force is maintained at the user-defined pre-tension force – this is

analogous to operation of the pre-tensioning screws in real cables. After time TMAXF0, the unstressed length is fixed and the force in the cable is determined in the normal way using the stiffness and change of length.

Sudden application of the cable forces at time zero may result in an excessively dynamic response during pre-tensioning. A ramp-up time TRAMP may optionally be defined. The cable force ramps up from zero at time zero to the full pre-tension F0 at time TRAMP. TRAMP should be less than or equal to TMAXF0.

If the model does not use dynamic relaxation, it is recommended that damping be applied during pre-tensioning so that the structure reaches a steady state by time TMAXF0.

If the model uses dynamic relaxation, TRAMP applies only during dynamic relaxation. The preload is held constant until the end of dynamic relaxation. The full cable pre-tension is retained after convergence, and TMAXF0 applies during the subsequent transient analysis. TMAXF0 can in this case be set to zero if required.

The cable mass will be calculated from length x area x density if VOL is set to zero on *SECTION_BEAM. Otherwise, VOL x density will be used.

If OUTPUT is set in any cable material, extra variables will be written to the d3plot and d3thdt files for all beam elements. Post-processors should interpret the extra data as per Resultant beams. Only the first extra data item, axial strain, is computed for MAT_CABLE elements.

If the stress-strain load curve option, LCID, is combined with preload, two types of behavior are available:

1. If the preload is applied using the TMAXF0/TRAMP method, the initial strain is calculated from the stress-strain curve to achieve the desired preload.
2. If TMAXF0/TRAMP are not used, the preload force is taken as additional to the force calculated from the stress/strain curve. Thus, the total stress in the cable will be higher than indicated by the stress/strain curve.

***MAT_CONCRETE_DAMAGE**

This is Material Type 72. This model has been used to analyze buried steel reinforced concrete structures subjected to impulsive loadings. A newer version of this model is available as *MAT_ CONCRETE_DAMAGE_REL3

Card 1

Variable	MID	RO	PR					
Type	A8	F	F					
Default	none	none	none					

Card 2

Variable	SIGF	A0	A1	A2				
Type	F	F	F	F				
Default	0.0	0.0	0.0	0.0				

Card 3

Variable	A0Y	A1Y	A2Y	A1F	A2F	B1	B2	B3
Type	F	F	F	F	F	F	F	F
Default	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0

Card 4

Variable	PER	ER	PRR	SIGY	ETAN	LCP	LCR	
Type	F	F	F	F	F	F	F	
Default	0.0	0.0	0.0	none	0.0	none	none	

Card 5

Variable	λ	λ2	λ3	λ4	λ5	λ6	λ7	λ8
Type	F	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	none	none

Card 6

Variable	λ9	λ10	λ11	λ12	λ13			
Type	F	F	F	F	F			
Default	none	none	none	none	none			

Card 7

Variable	η1	η2	η3	η4	η5	η6	η7	η8
Type	F	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	none	none

Card 8	1	2	3	4	5	6	7	8
Variable	η9	η10	η11	η12	η13			
Type	F	F	F	F	F			
Default	none	none	none	none	none			

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
PR	Poisson's ratio.
SIGF	Maximum principal stress for failure.
A0	Cohesion.
A1	Pressure hardening coefficient.
A2	Pressure hardening coefficient.
A0Y	Cohesion for yield
A1Y	Pressure hardening coefficient for yield limit
A2Y	Pressure hardening coefficient for yield limit
A1F	Pressure hardening coefficient for failed material.
A2F	Pressure hardening coefficient for failed material.
B1	Damage scaling factor.
B2	Damage scaling factor for uniaxial tensile path.
B3	Damage scaling factor for triaxial tensile path.
PER	Percent reinforcement.
ER	Elastic modulus for reinforcement.

VARIABLE	DESCRIPTION
PRR	Poisson's ratio for reinforcement.
SIGY	Initial yield stress.
ETAN	Tangent modulus/plastic hardening modulus.
LCP	Load curve ID giving rate sensitivity for principal material, see *DEFINE_CURVE.
LCR	Load curve ID giving rate sensitivity for reinforcement, see *DEFINE_CURVE.
$\lambda 1$-$\lambda 13$	Tabulated damage function
$\eta 1$-$\eta 13$	Tabulated scale factor.

Remarks:

Cohesion for failed material $a_{0f} = 0.0$

B3 must be positive or zero.

$\lambda_n < \lambda_n+1$. The first point must be zero.

*MAT_CONCRETE_DAMAGE_REL3

This is Material Type 72. The Karagozian & Case (K&C) Concrete Model - Release III is a three-invariant model, uses three shear failure surfaces, includes damage and strain-rate effects, and has origins based on the Pseudo-TENSOR Model (Material Type 16). The most significant user improvement provided by Release III is a model parameter generation capability, based solely on the unconfined compression strength of the concrete. The implementation of Release III significantly changed the user input, thus previous input files using Material Type 72, i.e. prior to LS-DYNA Version 971, are not compatible with the present input format.

An open source reference, that precedes the parameter generation capability, is provided in Malvar et al. [1997]. A workshop proceedings reference, Malvar et al. [1996], is useful, but may be difficult to obtain. More recent, but *limited distribution* reference materials, e.g. Malvar et al. [2000], may be obtained by contacting Karagozian & Case.

Seven card images are required to define the *complete* set of model parameters for the K&C Concrete Model; an Equation-of-State is also required for the pressure-volume strain response. Brief descriptions of all the input parameters are provided below, however it is expected that this model will be used primarily with the option to generate the model parameters based on the unconfined compression strength of the concrete. For those users wishing to examine, or modify, the generated model parameters, the generated parameters are written to the LS-DYNA `messag` file (`FORTRAN` Unit 59). The pressure-volume strain response for the model is also generated, in the form of a Tabulated Compaction Equation-of-State (EOS 8) whose parameters are also written to the LS-DYNA "messag" file.

Card 1

Variable	MID	RO	PR					
Type	A8	F	F					
Default	none	none	none					

Card 2

Variable	FT	A0	A1	A2	B1	OMEGA	A1F	
Type	F	F	F	F	F	F	F	
Default		0.0	0.0	0.0	0.0		0.0	

Card 3

Variable	Sλ	NOUT	EDROP	RSIZE	UCF	LCRATE	LOCWID	NPTS
Type	F	F	F	F	F	F	F	F
Default								

Card 4

Variable	λ01	λ02	λ03	λ04	λ05	λ06	λ07	λ08
Type	F	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	none	none

Card 5

Variable	λ09	λ10	λ11	λ12	λ13	B3	A0Y	A1Y
Type	F	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	0.0	0.0

Card 6

Variable	η01	η02	η03	η04	η05	η06	η07	η08
Type	F	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	none	none

Card 7	1	2	3	4	5	6	7	8
Variable	η09	η10	η11	η12	η13	B2	A2F	A2Y
Type	F	F	F	F	F	F	F	F
Default	none	none	none	none	none	0.0	0.0	0.0

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
PR	Poisson's ratio, v.
FT	Uniaxial tensile strength, f_t.
A0	Maximum shear failure surface parameter, a_0 **or** $-f_c'$ for **parameter generation** (recommended).
A1	Maximum shear failure surface parameter, a_1.
A2	Maximum shear failure surface parameter, a_2.
B1	Compressive damage scaling parameter, b_1
OMEGA	Fractional dilatancy, ω.
A1F	Residual failure surface coefficient, a_{1f}.
Sλ	λ stretch factor, *s*.
NOUT	Output selector for effective plastic strain (see table).
EDROP	Post peak dilatancy decay, N^{α}.
RSIZE	Unit conversion factor for length (inches/user-unit), e.g. 39.37 if user length unit in meters.
UCF	Unit conversion factor for stress (psi/user-unit), e.g. 145 if f_c' in MPa.

VARIABLE	DESCRIPTION
LCRATE	Define (load) curve number for strain-rate effects; effective strain rate on abscissa and strength enhancement on ordinate.
LOCWID	Three times the maximum aggregate diameter (input in user length units).
NPTS	Number of points in λ versus η damage relation; must be 13 points.
λ01	1st value of damage function, λ_1
λ02	2nd value of damage function,
λ03	3rd value of damage function,
λ04	4th value of damage function,
λ05	5th value of damage function,
λ06	6th value of damage function,
λ07	7th value of damage function,
λ08	8th value of damage function, λ_8
λ09	9th value of damage function,
λ10	10th value of damage function,
λ11	11th value of damage function,
λ12	12th value of damage function,
λ13	13th value of damage function, λ_{13}.
B3	Damage scaling coefficient for triaxial tension, b_3.
A0Y	Initial yield surface cohesion, a_{0y}.
A1Y	Initial yield surface coefficient, a_{1y}.
η01	1st value of scale factor, η_1.
η02	2nd value of scale factor,
η03	3rd value of scale factor,

VARIABLE	DESCRIPTION
η04	4^{th} value of scale factor,
η05	5^{th} value of scale factor,
η06	6^{th} value of scale factor,
η07	7^{th} value of scale factor,
η08	8^{th} value of scale factor, η_8
η09	9^{th} value of scale factor, η_9
η10	10^{th} value of scale factor,
η11	11^{th} value of scale factor,
η12	12^{th} value of scale factor,
η13	13^{th} value of scale factor, η_{13}.
B2	Tensile damage scaling exponent, b_2.
A2F	Residual failure surface coefficient, a_{2f}.
A2Y	Initial yield surface coefficient, a_{2y}.

Output of Selected Variables

LS-PrePost will display the variable described in Table 72.1 when the effective plastic strain is selected, for the corresponding user input value of NOUT; see Card 3 above.

Table 72.1 Output variables for post-processing using NOUT parameter.

NOUT	Function	Description
1		Current shear failure surface radius
2	$\delta = 2\lambda/(\lambda+\lambda_m)$	Scaled damage measure
3	$\dot{\sigma}_{ij}\dot{\varepsilon}_{ij}$	Strain energy (rate)
4	$\dot{\sigma}_{ij}\dot{\varepsilon}^{p}_{ij}$	Plastic strain energy (rate)

Sample Input for Concrete

As an example of the K&C Concrete Model material parameter generation, the following sample input for a 45.4 MPa (6,580 psi) unconfined compression strength concrete is provided. The basic units for the provided parameters are length in millimeters (mm), time in milliseconds (msec), and mass in grams (g). This base unit set yields units of force in Newtons (N) and pressure in Mega-Pascals (MPa).

Card 1

1	2	3	4	5	6	7	8
MID	RO	PR					
72	2.3E-3						

Card 2

FT	A0	A1	A2	B1	OMEGA	A1F	
F	-45.4						

Card 3

	1	2	3	4	5	6	7	8
Variable	Sλ	NOUT	EDROP	RSIZE	UCF	LCRATE	LOCWID	NPTS
Type				3.94E-2	145.0	723.0		

Card 4

Variable	λ01	λ02	λ03	λ04	λ05	λ06	λ07	λ08
Type								

Card 5

Variable	λ09	λ10	λ11	λ12	λ13	B3	A0Y	A1Y
Type								

Card 6

Variable	η01	η02	η03	η04	η05	η06	η07	η08
Type								

Card 7

Variable	η09	η10	η11	η12	η13	B2	A2F	A2Y
Type								

The effective strain rate versus shear strength enhancement is given by a LS-DYNA Define (load) Curve keyword, with ID 723. The sample input values, see Malvar & Ross [1998], are given in Table 72.2.

Table 72.2 Effective strain rate for 45.4 MPa concrete MAT 72 sample input.

Strain-Rate (1/ms)	Enhancement
-3.0E+01	9.70
-3.0E-01	9.70
-1.0E-01	6.72
-3.0E-02	4.50
-1.0E-02	3.12
-3.0E-03	2.09
-1.0E-03	1.45
-1.0E-04	1.36
-1.0E-05	1.28
-1.0E-06	1.20
-1.0E-07	1.13
-1.0E-08	1.06
0.0E+00	1.00
3.0E-08	1.00
1.0E-07	1.03
1.0E-06	1.08
1.0E-05	1.14
1.0E-04	1.20
1.0E-03	1.26
3.0E-03	1.29
1.0E-02	1.33
3.0E-02	1.36
1.0E-01	2.04
3.0E-01	2.94
3.0E+01	2.94

*MAT_LOW_DENSITY_VISCOUS_FOAM

This is Material Type 73 for Modeling Low Density Urethane Foam with high compressibility and with rate sensitivity which can be characterized by a relaxation curve. Its main applications are for seat cushions, padding on the Side Impact Dummies (SID), bumpers, and interior foams. Optionally, a tension cut-off failure can be defined. Also, see the notes below and the description of material 57: *MAT_LOW_DENSITY_FOAM.

Card 1

Variable	MID	RO	E	LCID	TC	HU	BETA	DAMP
Type	A8	F	F	F	F	F	F	F
Default	---	---	---	---	1.E+20	1.		
Remarks	---	---	---	---	---	3	1	---

Card 2

Variable	SHAPE	FAIL	BVFLAG	KCON	LCID2	BSTART	TRAMP	NV
Type	F	F	F	F	F	F	F	I
Default	1.0	0.0	0.0	0.0	0	0.0	0.0	6

If LCID2 = 0 then define the following viscoelastic constants. Up to 6 cards may be input. A keyword card (with a "*" in column 1) terminates this input if less than 6 cards are used. If LCID2 is nonzero skip this input. The variable REF is taken from the first card of this sequence.

Optional Cards	1	2	3	4	5	6	7	8
Variable	GI	BETAI	REF					
Type	F	F	F					

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
E	Young's modulus used in tension. For implicit problems E is set to the initial slope of load curve LCID.
LCID	Load curve ID, see *DEFINE_CURVE, for nominal stress versus strain.
TC	Tension cut-off stress
HU	Hysteretic unloading factor between 0 and 1 (default=1, i.e., no energy dissipation), see also Figure 57.1.
BETA	β, decay constant to model creep in unloading. EQ:0 No relaxation.
DAMP	Viscous coefficient (.05< recommended value <.50) to model damping effects. LT.0.0: |DAMP| is the load curve ID, which defines the damping constant as a function of the maximum strain in compression defined as: $\varepsilon_{\max} = \max\left(1-\lambda_1, 1-\lambda_2, 1.-\lambda_3\right).$ In tension, the damping constant is set to the value corresponding to the strain at 0. The abscissa should be defined from 0 to 1.
SHAPE	Shape factor for unloading. Active for nonzero values of the hysteretic unloading factor. Values less than one reduces the energy dissipation and greater than one increases dissipation, see also Figure 57.1.

VARIABLE	DESCRIPTION
FAIL	Failure option after cutoff stress is reached: EQ.0.0: tensile stress remains at cut-off value, EQ.1.0: tensile stress is reset to zero.
BVFLAG	Bulk viscosity activation flag, see remark below: EQ.0.0: no bulk viscosity (recommended), EQ.1.0: bulk viscosity active.
KCON	Stiffness coefficient for contact interface stiffness. Maximum slope in stress vs. strain curve is used. When the maximum slope is taken for the contact, the time step size for this material is reduced for stability. In some cases Δt may be significantly smaller, and defining a reasonable stiffness is recommended.
LCID2	Load curve ID of relaxation curve. If constants $\beta\iota$ are determined via a least squares fit. This relaxation curve is shown in Figure 76.1. This model ignores the constant stress.
BSTART	Fit parameter. In the fit, β_1 is set to zero, β_2 is set to BSTART, β_3 is 10 times β_2, β_4 is 10 times greater than β_3 , and so on. If zero, BSTART= .01.
TRAMP	Optional ramp time for loading.
NV	Number of terms in fit. If zero, the default is 6. Currently, the maximum number is set to 6. Values of 2 are 3 are recommended, since each term used adds significantly to the cost. Caution should be exercised when taking the results from the fit. Preferably, all generated coefficients should be positive. Negative values may lead to unstable results. Once a satisfactory fit has been achieved it is recommended that the coefficients which are written into the output file be input in future runs.
GI	Optional shear relaxation modulus for the ith term
BETAI	Optional decay constant if ith term
REF	Use reference geometry to initialize the stress tensor. The reference geometry is defined by the keyword:*INITIAL_FOAM_REFERENCE_GEOMETRY (see there for more details). EQ.0.0: off, EQ.1.0: on.

Remarks:

This viscoelastic foam model is available to model highly compressible viscous foams. The hyperelastic formulation of this model follows that of Material 57.

Rate effects are accounted for through linear viscoelasticity by a convolution integral of the form

$$\sigma_{ij}^r = \int_0^t g_{ijkl}\left(t-\tau\right)\frac{\partial \varepsilon_{kl}}{\partial \tau}d\tau$$

where $g_{ijkl}\left(t-\tau\right)$ is the relaxation function. The stress tensor, σ_{ij}^r, augments the stresses determined from the foam, σ_{ij}^f; consequently, the final stress, σ_{ij}, is taken as the summation of the two contributions:

$$\sigma_{ij} = \sigma_{ij}^f + \sigma_{ij}^r .$$

Since we wish to include only simple rate effects, the relaxation function is represented by up to six terms of the Prony series:

$$g\left(t\right) = \alpha_0 + \sum_{m=1}^{N}\alpha_m e^{-\beta t}$$

This model is effectively a Maxwell fluid which consists of a dampers and springs in series. The formulation is performed in the local system of principal stretches where only the principal values of stress are computed and triaxial coupling is avoided. Consequently, the one-dimensional nature of this foam material is unaffected by this addition of rate effects. The addition of rate effects necessitates 42 additional history variables per integration point. The cost and memory overhead of this model comes primarily from the need to "remember" the local system of principal stretches and the evaluation of the viscous stress components.

Additional Remarks:

1. When hysteretic unloading is used the reloading will follow the unloading curve if the decay constant, β, is set to zero. If β is nonzero the decay to the original loading curve is governed by the expression:

$$1-e^{-\beta t}$$

2. The bulk viscosity, which generates a rate dependent pressure, may cause an unexpected volumetric response and, consequently, it is optional with this model.

3. The hysteretic unloading factor results in the unloading curve to lie beneath the loading curve as shown in Figure 57.1. This unloading provides energy dissipation which is reasonable in certain kinds of foam.

*MAT_ELASTIC_SPRING_DISCRETE_BEAM

This is Material Type 74. This model permits elastic springs with damping to be combined and represented with a discrete beam element type 6. Linear stiffness and damping coefficients can be defined, and, for nonlinear behavior, a force versus deflection and force versus rate curves can be used. Displacement based failure and an initial force are optional.

Card 1

	1	2	3	4	5	6	7	8
Variable	MID	RO	K	F0	D	CDF	TDF	
Type	A8	F	F	F	F	F	F	

Card 2

Variable	FLCID	HLCID	C1	C2	DLE	GLCID		
Type	F	F	F	F	F	I		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density, see also volume in *SECTION_BEAM definition.
K	Stiffness coefficient.
F0	Optional initial force. This option is inactive if this material is referenced in a part referenced by material type *MAT_ELASTIC_ 6DOF_SPRING
D	Viscous damping coefficient.
CDF	Compressive displacement at failure. Input as a positive number. After failure, no forces are carried. This option does not apply to zero length springs. EQ.0.0: inactive.
TDF	Tensile displacement at failure. After failure, no forces are carried.
FLCID	Load curve ID, see *DEFINE_CURVE, defining force versus deflection for nonlinear behavior.

VARIABLE	DESCRIPTION
HLCID	Load curve ID, see *DEFINE_CURVE, defining force versus relative velocity for nonlinear behavior (optional). If the origin of the curve is at (0,0) the force magnitude is identical for a given magnitude of the relative velocity, i.e., only the sign changes.
C1	Damping coefficient for nonlinear behavior (optional).
C2	Damping coefficient for nonlinear behavior (optional).
DLE	Factor to scale time units. The default is unity.
GLCID	Optional load curve ID, see *DEFINE_CURVE, defining a scale factor versus deflection for load curve ID, HLCID. If zero, a scale factor of unity is assumed.

Remarks:

If the linear spring stiffness is used, the force, F, is given by:

$$F = F_0 + K\Delta L + D\Delta\dot{L}$$

but if the load curve ID is specified, the force is then given by:

$$F = F_0 + Kf\left(\Delta L\right)\left[1 + C1\cdot\Delta\dot{L} + C2\cdot\mathrm{sgn}\left(\Delta\dot{L}\right)\ln\left(\max\left\{1., \frac{\Delta\dot{L}}{DLE}\right\}\right)\right] + D\Delta\dot{L} + g\left(\Delta L\right)h\left(\Delta\dot{L}\right)$$

In these equations, ΔL is the change in length

$$\Delta L = current\ length - initial\ length$$

The cross sectional area is defined on the section card for the discrete beam elements, See *SECTION_BEAM. The square root of this area is used as the contact thickness offset if these elements are included in the contact treatment.

***MAT_BILKHU/DUBOIS_FOAM**

This is Material Type 75. This model is for the simulation of isotropic crushable forms. Uniaxial and triaxial test data are used to describe the behavior.

Card 1

1	2	3	4	5	6	7	8
Variable	MID	RO	YM	LCPY	LCUYS	VC	PC	VPC
Type	A8	F	F	F	F	F	F	F

Card 2

Variable	TSC	VTSC	LCRATE	PR	KCON	ISFLG		
Type	I	F	F	F	F	F		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
YM	Young's modulus (E)
LCPY	Load curve ID giving pressure for plastic yielding versus volumetric strain, see Figure 75.1.
LCUYS	Load curve ID giving uniaxial yield stress versus volumetric strain, see Figure 75.1, all abscissa should be positive if only the results of a compression test are included, optionally the results of a tensile test can be added (corresponding to negative values of the volumetric strain), in the latter case PC, VPC, TC and VTC will be ignored
VC	Viscous damping coefficient (.05<recommended value<.50).
PC	Pressure cutoff. If zero, the default is set to one-tenth of p_0, the yield pressure corresponding to a volumetric strain of zero.
VPC	Variable pressure cutoff as a fraction of pressure yield value. If non-zero this will override the pressure cutoff value PC.

VARIABLE	DESCRIPTION
TC	Tension cutoff for uniaxial tensile stress. Default is zero. A nonzero value is recommended for better stability.
VTC	Variable tension cutoff as a fraction of the uniaxial compressive yield strength, if non-zero this will override the tension cutoff value TC.
LCRATE	Load curve ID giving a scale factor for the previous yield curves, dependent upon the volumetric plastic strain.
PR	Poisson coefficient, which applies to both elastic and plastic deformations, must be smaller then 0.5
KCON	Stiffness coefficient for contact interface stiffness. If undefined one-third of Young's modulus, YM, is used. KCON is also considered in the element time step calculation; therefore, large values may reduce the element time step size.
ISFLG	Flag for tensile response (active only if negative abscissa are present in load curve LCUYS) EQ.0: load curve abscissa in tensile region correspond to volumetric strain EQ.1: load curve abscissa in tensile region correspond to effective strain

Remarks:

The logarithmic volumetric strain is defined in terms of the relative volume, V , as:

$$\gamma = -\ln(V)$$

If used (ISFLG-1), the effective strain is defined in the usual way:

$$\varepsilon_{eff} = \sqrt{\frac{2}{3}\varepsilon:\varepsilon}$$

In defining the load curve LCPY the stress and strain pairs should be positive values starting with a volumetric strain value of zero.

The load curve LCUYS can optionally contain the results of the tensile test (corresponding to negative values of the volumetric strain), if so, then the load curve information will override PC, VPC, TC and VTC

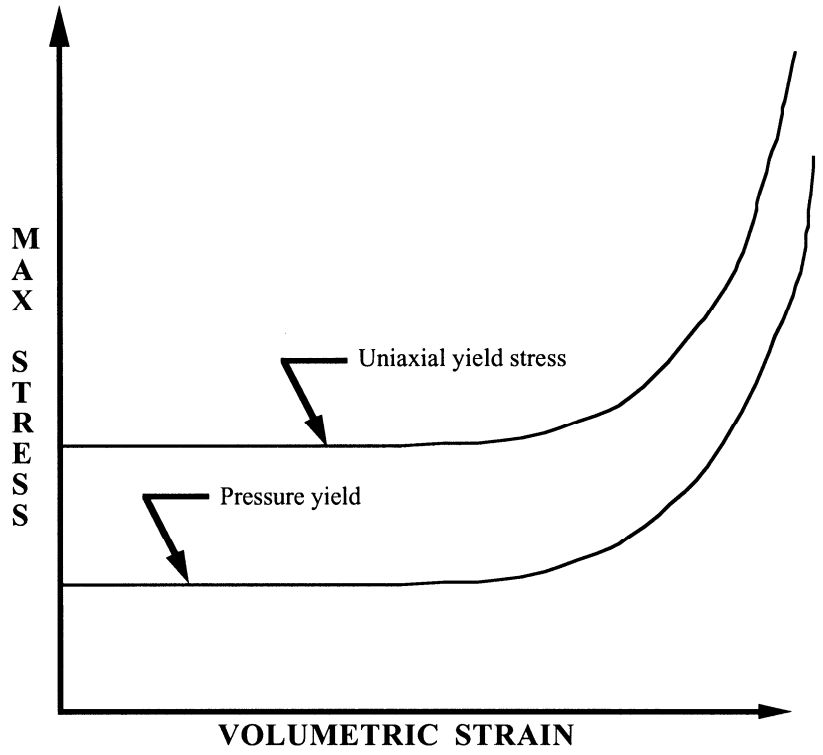


Figure 75.1. Behavior of crushable foam. Unloading is elastic.

The yield surface is defined as an ellipse in the equivalent pressure and von Mises stress plane.

*MAT_GENERAL_VISCOELASTIC

This is Material Type 76. This material model provides a general viscoelastic Maxwell model having up to 6 terms in the prony series expansion and is useful for modeling dense continuum rubbers and solid explosives. Either the coefficients of the prony series expansion or a relaxation curve may be specified to define the viscoelastic deviatoric and bulk behavior.

The material model can also be used with laminated shell. Either an elastic or viscoelastic layer can be defined with the laminated formulation. To activate laminated shell you need the laminated formulation flag on *CONTROL_SHELL. With the laminated option a userdefined integration rule is needed.

Card 1

1	2	3	4	5	6	7	8
MID	RO	BULK	PCF	EF	TREF	A	B
A8	F	F	F	F	F	F	F

Insert a blank card here if constants are defined on cards 3,4,... below.

If an elastic layer is defined in a laminated shell this card must be blank.

Card 2

1	2	3	4	5	6	7	8
LCID	NT	BSTART	TRAMP	LCIDK	NTK	BSTARTK	TRAMPK
F	I	F	F	F	I	F	F

Card Format for viscoelastic constants. Up to 6 cards may be input. A keyword card (with a "*" in column 1) terminates this input if less than 6 cards are used. These cards are not needed if relaxation data is defined. The number of terms for the shear behavior may differ from that for the bulk behavior: simply insert zero if a term is not included.

If an elastic layer is defined you only need to define GI and KI (note in an elastic layer only one card is needed)

Optional Cards	1	2	3	4	5	6	7	8
Variable	GI	BETAI	KI	BETAKI				
Type	F	F	F	F				

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
BULK	Elastic bulk modulus.
PCF	Tensile pressure elimination flag for solid elements only. If set to unity tensile pressures are set to zero.
EF	Elastic flag (if equal 1, the layer is elastic. If 0 the layer is viscoelastic).
TREF	Reference temperature for shift function (must be greater than zero).
A	Coefficient for the Arrhenius and the Williams-Landau-Ferry shift functions.
B	Coefficient for the Williams-Landau-Ferry shift function.
LCID	Load curve ID for deviatoric behavior if constants, G_i, and β_i are determined via a least squares fit. This relaxation curve is shown below.
NT	Number of terms in shear fit. If zero the default is 6. Fewer than NT terms will be used if the fit produces one or more negative shear moduli. Currently, the maximum number is set to 6.
BSTART	In the fit, β_1 is set to zero, β_2 is set to BSTART, β_3 is 10 times β_2, β_4 is 100 times greater than β_3, and so on. If zero, BSTART is determined by an iterative trial and error scheme.
TRAMP	Optional ramp time for loading.
LCIDK	Load curve ID for bulk behavior if constants, K_i, and $\beta\kappa_i$ are determined via a least squares fit. This relaxation curve is shown below.
NTK	Number of terms desired in bulk fit. If zero the default is 6. Currently, the maximum number is set to 6.

VARIABLE	DESCRIPTION
BSTARTK	In the fit, $\beta\kappa_1$ is set to zero, $\beta\kappa_2$ is set to BSTARTK, $\beta\kappa_3$ is 10 times $\beta\kappa_2$, $\beta\kappa_4$ is 100 times greater than $\beta\kappa_3$, and so on. If zero, BSTARTK is determined by an iterative trial and error scheme.
TRAMPK	Optional ramp time for bulk loading.
GI	Optional shear relaxation modulus for the ith term
BETAI	Optional shear decay constant for the ith term
KI	Optional bulk relaxation modulus for the ith term
BETAKI	Optional bulk decay constant for the ith term

Remarks:

Rate effects are taken into accounted through linear viscoelasticity by a convolution integral of the form:

$$\sigma_{ij} = \int_0^t g_{ijkl}\left(t-\tau\right)\frac{\partial \varepsilon_{kl}}{\partial \tau}d\tau$$

where $g_{ijkl(t-\tau)}$ is the relaxation functions for the different stress measures. This stress is added to the stress tensor determined from the strain energy functional.

If we wish to include only simple rate effects, the relaxation function is represented by six terms from the Prony series:

$$g\left(t\right) = \sum_{m=1}^{N} G_m e^{-\beta_m t}$$

We characterize this in the input by shear moduli, G_i, and decay constants, β_i. An arbitrary number of terms, up to 6, may be used when applying the viscoelastic model.

For volumetric relaxation, the relaxation function is also represented by the Prony series in terms of bulk moduli:

$$k\left(t\right) = \sum_{m=1}^{N} K_m e^{-\beta_{k_m} t}$$

The Arrhenius and Williams-Landau-Ferry (WLF) shift functions account for the effects of the temperature on the stress relaxation. A scaled time, *t'*,

$$t' = \int_0^t \Phi(T)dt$$

is used in the relaxation function instead of the physical time. The Arrhenius shift function is

$$\Phi(T) = \exp\left(-A\{\frac{1}{T} - \frac{1}{T_{REF}}\}\right)$$

and the Williams-Landau-Ferry shift function is

$$\Phi(T) = \exp\left(-A\frac{T - T_{REF}}{B + T - T_{REF}}\right)$$

If all three values (TREF, A, and B) are not zero, the WLF function is used; the Arrhenius function is used if B is zero; and no scaling is applied if all three values are zero.

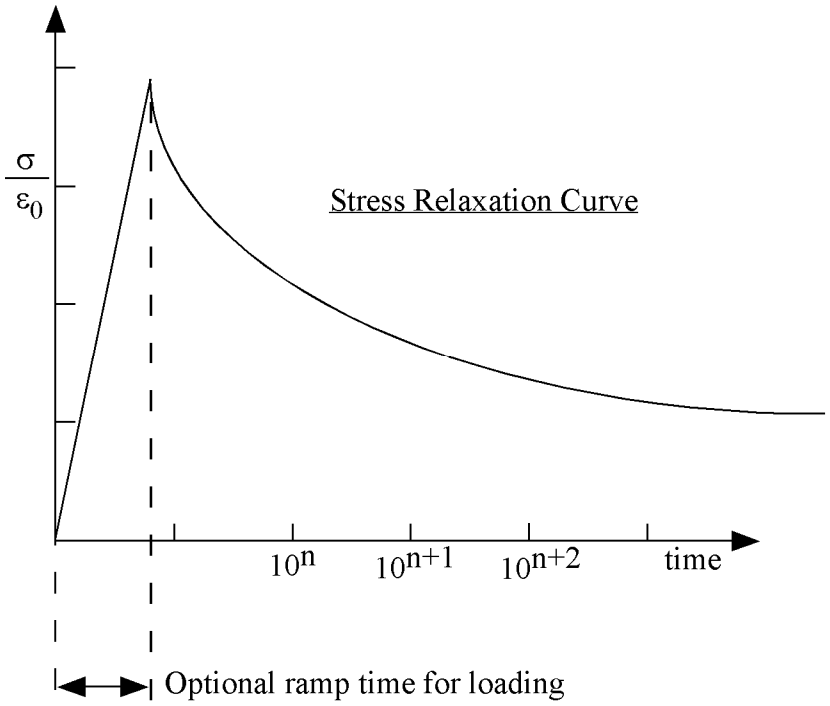


Figure 76.1. Relaxation curve. This curve defines stress versus time where time is defined on a logarithmic scale. For best results, the points defined in the load curve should be equally spaced on the logarithmic scale. Furthermore, the load curve should be smooth and defined in the positive quadrant. If nonphysical values are determined by least squares fit, LS-DYNA will terminate with an error message after the initialization phase is completed. If the ramp time for loading is included, then the relaxation which occurs during the loading phase is taken into account. This effect may or may not be important.

*MAT_HYPERELASTIC_RUBBER

This is Material Type 77. This material model provides a general hyperelastic rubber model combined optionally with linear viscoelasticity as outlined by Christensen [1980].

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	PR	N	NV	G	SIGF	REF
Type	A8	F	F	I	I	F	F	F

Card 2 if N > 0, a least squares fit is computed from uniaxial data

Card 2	1	2	3	4	5	6	7	8
Variable	SGL	SW	ST	LCID1	DATA	LCID2	BSTART	TRAMP
Type	F	F	F	F	F	F	F	F

Card 2 if N = 0 define the following constants

Card 2	1	2	3	4	5	6	7	8
Variable	C10	C01	C11	C20	C02	C30		
Type	F	F	F	F	F	F		

Card Format for Viscoelastic Constants and frictional damping constants. Up to 6 cards may be input. A keyword card (with a "*" in column 1) terminates this input if less than 6 cards are used.

Optional Cards	1	2	3	4	5	6	7	8
Variable	GI	BETAI	GJ	SIGFJ				
Type	F	F	F	F				

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
PR	Poissons ratio (>.49 is recommended, smaller values may not work and should not be used).
N	Number of constants to solve for: EQ.1: Solve for C10 and C01 EQ.2: Solve for C10, C01, C11, C20, and C02 EQ.3: Solve for C10, C01, C11, C20, C02, and C30
NV	Number of Prony series terms in fit. If zero, the default is 6. Currently, the maximum number is set to 6. Values less than 6, possibly 3-5 are recommended, since each term used adds significantly to the cost. Caution should be exercised when taking the results from the fit. Preferably, all generated coefficients should be positive. Negative values may lead to unstable results. Once a satisfactory fit has been achieved it is recommended that the coefficients which are written into the output file be input in future runs.
G	Shear modulus for frequency independent damping. Frequency independent damping is based of a spring and slider in series. The critical stress for the slider mechanism is SIGF defined below. For the best results, the value of G should be 250-1000 times greater than SIGF.
SIGF	Limit stress for frequency independent, frictional, damping.
REF	Use reference geometry to initialize the stress tensor. The reference geometry is defined by the keyword:*INITIAL_FOAM_REFERENCE_GEOMETRY (see there for more details). EQ.0.0: off, EQ.1.0: on.

VARIABLE	DESCRIPTION

If N>0 test information from a uniaxial test are used.

SGL	Specimen gauge length
SW	Specimen width
ST	Specimen thickness
LCID1	Load curve ID giving the force versus actual change in the gauge length
DATA	Type of experimental data. EQ.0.0: uniaxial data (Only option for this model)
LCID2	Load curve ID of relaxation curve If constants β_i are determined via a least squares fit. This relaxation curve is shown in Figure 76.1. This model ignores the constant stress.
BSTART	In the fit, β_1 is set to zero, β_2 is set to BSTART, β_3 is 10 times β_2, β_4 is 100 times greater than β_3 , and so on. If zero, BSTART is determined by an iterative trial and error scheme.
TRAMP	Optional ramp time for loading.

If N=0, the following constants have to be defined:

C10	C_{10}
C01	C_{01}
C11	C_{11}
C20	C_{20}
C02	C_{02}
C30	C_{30}
GI	Optional shear relaxation modulus for the ith term
BETAI	Optional decay constant if ith term
GJ	Optional shear modulus for frequency independent damping represented as the jth spring and slider in series in parallel to the rest of the stress contributions.

SIGFJ	Limit stress for frequency independent, frictional, damping represented as the jth spring and slider in series in parallel to the rest of the stress contributions.

Remarks:

Rubber is generally considered to be fully incompressible since the bulk modulus greatly exceeds the shear modulus in magnitude. To model the rubber as an unconstrained material a hydrostatic work term, $W_H(J)$, is included in the strain energy functional which is function of the relative volume, J, [Ogden 1984]:

$$W(J_1, J_2, J) = \sum_{p,q=0}^{n} C_{pq}(J_1 - 3)^p (J_2 - 3)^q + W_H(J)$$

$$J_1 = I_1 I_3^{-1/3}$$

$$J_2 = I_2 I_3^{-2/3}$$

In order to prevent volumetric work from contributing to the hydrostatic work the first and second invariants are modified as shown. This procedure is described in more detail by Sussman and Bathe [1987].

Rate effects are taken into account through linear viscoelasticity by a convolution integral of the form:

$$\sigma_{ij} = \int_0^t g_{ijkl}(t-\tau)\frac{\partial \varepsilon_{kl}}{\partial \tau} d\tau$$

or in terms of the second Piola-Kirchhoff stress, S_{ij}, and Green's strain tensor, E_{ij},

$$S_{ij} = \int_0^t G_{ijkl}(t-\tau)\frac{\partial E_{kl}}{\partial \tau} d\tau$$

where $g_{ijkl}(t-\tau)$ and $G_{ijkl}(t-\tau)$ are the relaxation functions for the different stress measures. This stress is added to the stress tensor determined from the strain energy functional.

If we wish to include only simple rate effects, the relaxation function is represented by six terms from the Prony series:

$$g(t) = \alpha_0 + \sum_{m=1}^{N} \alpha_m e^{-\beta t}$$

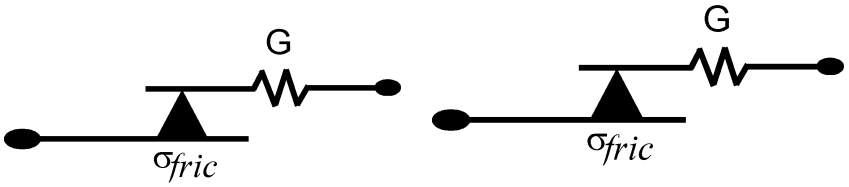
given by,

$$g(t) = \sum_{i=1}^{n} G_i e^{-\beta_i t}$$

This model is effectively a Maxwell fluid which consists of a dampers and springs in series. We characterize this in the input by shear moduli, G_i, and decay constants, β_i. The viscoelastic behavior is optional and an arbitrary number of terms may be used.

The Mooney-Rivlin rubber model (model 27) is obtained by specifying n=1. In spite of the differences in formulations with Model 27, we find that the results obtained with this model are nearly identical with those of Material 27 as long as large values of Poisson's ratio are used.

The frequency independent damping is obtained by the having a spring and slider in series as shown in the following sketch:



Several springs and sliders in series can be defined that are put in parallel to the rest of the stress contributions of this material model.

***MAT_OGDEN_RUBBER**

This is also Material Type 77. This material model provides the Ogden [1984] rubber model combined optionally with linear viscoelasticity as outlined by Christensen [1980].

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	PR	N	NV	G	SIGF	REF
Type	A8	F	F	I	I	F	F	F

Card 2 if N > 0, a least squares fit is computed from uniaxial data

Card 2	1	2	3	4	5	6	7	8
Variable	SGL	SW	ST	LCID1	DATA	LCID2	BSTART	TRAMP
Type	F	F	F	F	F	F		F

Cards 2,3 if N = 0 define the following constants

Card 2	1	2	3	4	5	6	7	8
Variable	MU1	MU2	MU3	MU4	MU5	MU6	MU7	MU8
Type	F	F	F	F	F	F	F	F

Card 3

Variable	ALPHA1	ALPHA2	ALPHA3	ALPHA4	ALPHA5	ALPHA6	ALPHA7	ALPHA8
Type	F	F	F	F	F	F	F	F

Card Format for Viscoelastic Constants. Up to 6 cards may be input. A keyword card (with a "*" in column 1) terminates this input if less than 6 cards are used.

Optional Cards	1	2	3	4	5	6	7	8
Variable	GI	BETAI						
Type	F	F						

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
PR	Poissons ratio ($\geq .49$ is recommended, smaller values may not work and should not be used).
N	Order of fit to the Ogden model, (currently <9, 2 generally works okay). The constants generated during the fit are printed in the output file and can be directly input in future runs, thereby, saving the cost of performing the nonlinear fit. The users need to check the correction of the fit results before proceeding to compute.
NV	Number of Prony series terms in fit. If zero, the default is 6. Currently, the maximum number is set to 6. Values less than 6, possibly 3-5 are recommended, since each term used adds significantly to the cost. Caution should be exercised when taking the results from the fit. Preferably, all generated coefficients should be positive. Negative values may lead to unstable results. Once a satisfactory fit has been achieved it is recommended that the coefficients which are written into the output file be input in future runs.
G	Shear modulus for frequency independent damping. Frequency independent damping is based on a spring and slider in series. The critical stress for the slider mechanism is SIGF defined below. For the best results, the value of G should be 250-1000 times greater than SIGF.
SIGF	Limit stress for frequency independent, frictional, damping.
REF	Use reference geometry to initialize the stress tensor. The reference geometry is defined by the keyword:*INITIAL_FOAM_REFERENCE_GEOMETRY (see there for more details). EQ.0.0: off, EQ.1.0: on.

VARIABLE	DESCRIPTION

If N>0 test information from a uniaxial test are used:

SGL	Specimen gauge length
SW	Specimen width
ST	Specimen thickness
LCID1	Load curve ID giving the force versus actual change in the gauge length
DATA	Type of experimental data. EQ.1.0: uniaxial data (default) EQ.2.0: biaxial data EQ.3.0: pure shear data
LCID2	Load curve ID of relaxation curve. If constants $\beta\iota$ are determined via a least squares fit. This relaxation curve is shown in Figure 76.1. This model ignores the constant stress.
BSTART	In the fit, β_1 is set to zero, β_2 is set to BSTART, β_3 is 10 times β_2 , β_4 is 100 times greater than β_3 , and so on. If zero, BSTART is determined by an iterative trial and error scheme.
TRAMP	Optional ramp time for loading.

If N=0, the constants MUi and ALPHAi have to be defined:

MUi	μ_i, the ith shear modulus, i varies up to 8. See discussion below.
ALPHAi	α_i, the ith exponent, i varies up to 8. See discussion below.
GI	Optional shear relaxation modulus for the ith term
BETAI	Optional decay constant if ith term

Remarks:

Rubber is generally considered to be fully incompressible since the bulk modulus greatly exceeds the shear modulus in magnitude. To model the rubber as an unconstrained material a hydrostatic work term is included in the strain energy functional which is function of the relative volume, J , [Ogden 1984]:

$$W^* = \sum_{i=1}^{3}\sum_{j=1}^{n}\frac{\mu_j}{\alpha_j}\left(\lambda_i^{*\alpha_j}-1\right)+K\left(J-1-\ln J\right)$$

The asterisk (*) indicates that the volumetric effects have been eliminated from the principal stretches, λ_j^*.. The number of terms, n, may vary between 1 to 8 inclusive, and K is the bulk modulus.

Rate effects are taken into account through linear viscoelasticity by a convolution integral of the form:

$$\sigma_{ij} = \int_0^t g_{ijkl}(t-\tau)\frac{\partial \varepsilon_{kl}}{\partial \tau} d\tau$$

or in terms of the second Piola-Kirchhoff stress, $\{S_0\}$, and Green's strain tensor, $\{S_{RT}\}$,

$$S_{ij} = \int_0^t G_{ijkl}(t-\tau)\frac{\partial E_{kl}}{\partial \tau} d\tau$$

where $g_{ijkl}(t-\tau)$ and $G_{ijkl}(t-\tau)$ are the relaxation functions for the different stress measures. This stress is added to the stress tensor determined from the strain energy functional.

If we wish to include only simple rate effects, the relaxation function is represented by six terms from the Prony series:

$$g(t) = \alpha_0 + \sum_{m=1}^{N} \alpha_m e^{-\beta t}$$

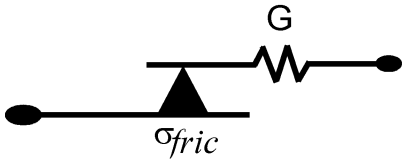
given by,

$$g(t) = \sum_{i=1}^{n} G_i e^{-\beta_i t}$$

This model is effectively a Maxwell fluid which consists of a dampers and springs in series. We characterize this in the input by shear moduli, G_i, and decay constants, β_i. The viscoelastic behavior is optional and an arbitrary number of terms may be used.

The Mooney-Rivlin rubber model (model 27) is obtained by specifying *n*=1. In spite of the differences in formulations with Model 27, we find that the results obtained with this model are nearly identical with those of Material 27 as long as large values of Poisson's ratio are used.

The frequency independent damping is obtained by the having a spring and slider in series as shown in the following sketch:



***MAT_SOIL_CONCRETE**

This is Material Type 78. This model permits concrete and soil to be efficiently modeled. See the explanations below.

Card 1

1	2	3	4	5	6	7	8
Variable	MID	RO	G	K	LCPV	LCYP	LCFP	LCRP
Type	A8	F	F	F	F	F	F	F

Card 2

Variable	PC	OUT	B	FAIL				
Type	F	F	F	F				

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
G	Shear modulus
K	Bulk modulus
LCPV	Load curve ID for pressure versus volumetric strain. The pressure versus volumetric strain curve is defined in compression only. The sign convention requires that both pressure and compressive strain be defined as positive values where the compressive strain is taken as the negative value of the natural logarithm of the relative volume.
LCYP	Load curve ID for yield versus pressure: GT.0: von Mises stress versus pressure, LT.0: Second stress invariant, J_2, versus pressure. This curve must be defined.
LCFP	Load curve ID for plastic strain at which fracture begins versus pressure. This load curve ID must be defined if B>0.0.

VARIABLE	DESCRIPTION
LCRP	Load curve ID for plastic strain at which residual strength is reached versus pressure. This load curve ID must be defined if B>0.0.
PC	Pressure cutoff for tensile fracture
OUT	Output option for plastic strain in database: EQ.0: volumetric plastic strain, EQ.1: deviatoric plastic strain.
B	Residual strength factor after cracking, see Figure 78.1.
FAIL	Flag for failure: EQ.0: no failure, EQ.1: When pressure reaches failure pressure element is eroded, EQ.2: When pressure reaches failure pressure element loses it ability to carry tension.

Remarks:

Pressure is positive in compression. Volumetric strain is defined as the natural log of the relative volume and is *positive* in compression where the relative volume, V, is the ratio of the current volume to the initial volume. The tabulated data should be given in order of increasing compression. If the pressure drops below the cutoff value specified, it is reset to that value and the deviatoric stress state is eliminated.

If the load curve ID (LCYP) is provided as a positive number, the deviatoric, perfectly plastic, pressure dependent, yield function ϕ, is given as

$$\phi=\sqrt{3J_2}-F(p)=\sigma_y-F(p)$$

where , *F(p)* is a tabulated function of yield stress versus pressure, and the second invariant, J_2, is defined in terms of the deviatoric stress tensor as:

$$J_2=\frac{1}{2}S_{ij}S_{ij}$$

assuming that if the ID is given as negative then the yield function becomes:

$$\phi=J_2-F(p)$$

being the deviatoric stress tensor.

If cracking is invoked by setting the residual strength factor, B, on card 2 to a value between 0.0 and 1.0, the yield stress is multiplied by a factor f which reduces with plastic strain according to a trilinear law as shown in Figure 78.1.

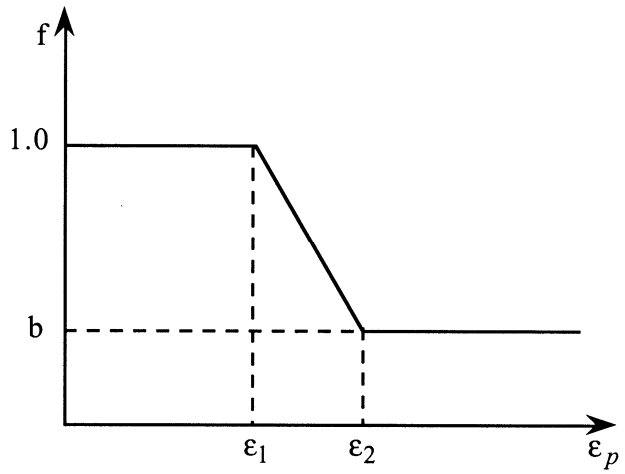


Figure 78.1. Strength reduction factor.

b = residual strength factor

ε_1 = plastic stain at which cracking begins.

ε_2 = plastic stain at which residual strength is reached.

ε_1 and ε_2 are tabulated functions of pressure that are defined by load curves, see Figure 78.2. The values on the curves are pressure versus strain and should be entered in order of increasing pressure. The strain values should always increase monotonically with pressure.

By properly defining the load curves, it is possible to obtain the desired strength and ductility over a range of pressures, see Figure 78.3.

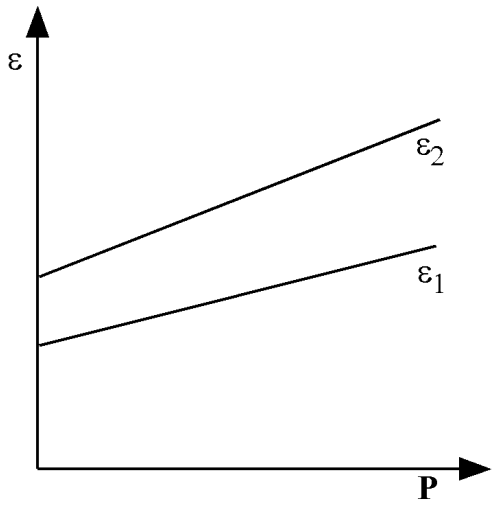


Figure 78.2. Cracking strain versus pressure.

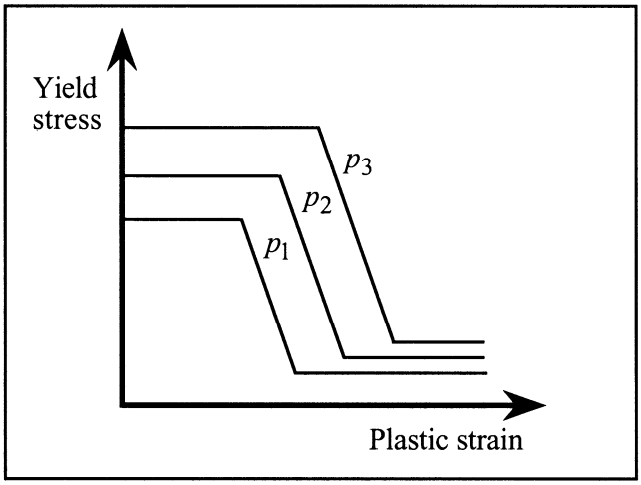


Figure 78.3.

***MAT_HYSTERETIC_SOIL**

This is Material Type 79. This model is a nested surface model with up to ten superposed "layers" of elasto-perfectly plastic material, each with its own elastic moduli and yield values. Nested surface models give hysteric behavior, as the different "layers" yield at different stresses. See Remarks below.

Note: This Material Type will be available starting in release 3 of version 971.

Card 1

Variable	MID	RO	K0	P0	B	A0	A1	A2
Type	A8	F	F	F	F	F	F	F

Card 2

Variable	DF	RP	LCID	SFLC	DIL_A	DIL_B	DIL_C	DIL_D
Type	F	F	F	F	F	F	F	F

Card 3

Variable	GAM1	GAM2	GAM3	GAM4	GAM5			PINIT
Type	F	F	F	F	F			I

Card 4

Variable	TAU1	TAU2	TAU3	TAU4	TAU5			
Type	F	F	F	F	F			

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
K0	Bulk modulus at the reference pressure
P0	Cut-off/datum pressure (must be 0≤ i.e. tensile). Below this pressure, stiffness and strength disappears; this is also the "zero" pressure for pressure-varying properties.
B	Exponent for pressure-sensitive moduli, b: $G = G_0 \left(p - p_o \right)^b$ $K = K_0 \left(p - p_o \right)^b$. b, must lie in the range 0≤b<1. Values close to 1 are not recommended because the pressure becomes indeterminate.
A0	Yield function constant a_0 (Default = 1.0), see Material Type 5.
A1	Yield function constant a_1 (Default = 0.0), see Material Type 5.
A2	Yield function constant a_2 (Default = 0.0), see Material Type 5.
DF	Damping factor. Must be in the range 0≤df≤1: EQ.0: no damping, EQ.1: maximum damping.
RP	Reference pressure for following input data.
LCID	Load curve ID defining shear strain verses shear stress. Up to ten points may be defined in the load curve. See *DEFINE_CURVE.
SFLD	Scale factor to apply to shear stress in LCID.
DIL_A	Dilation parameter A
DIL_B	Dilation parameter B
DIL_C	Dilation parameter C
DIL_D	Dilation parameter D
GAM1	γ_1, shear strain (ignored if LCID is non zero).
GAM2	γ_2, shear strain (ignored if LCID is non zero).

VARIABLE	DESCRIPTION
GAM3	γ_3, shear strain (ignored if LCID is non zero).
GAM4	γ_4, shear strain (ignored if LCID is non zero).
GAM5	γ_5, shear strain (ignored if LCID is non zero).
TAU1	τ_1, shear stress at γ_1 (ignored if LCID is non zero).
TAU2	τ_2, shear stress at γ_2 (ignored if LCID is non zero).
TAU3	τ_3, shear stress at γ_3 (ignored if LCID is non zero).
TAU4	τ_4, shear stress at γ_4 (ignored if LCID is non zero).
TAU5	τ_5, shear stress at γ_5 (ignored if LCID is non zero).
PINIT	Flag for pressure sensitivity (B and A0, A1, A2 equations): EQ.0: Use current pressure (will vary during the analysis) EQ.1: Use pressure from initial stress state EQ.2: Use initial "plane stress" pressure $(\sigma_v + \sigma_h)/2$ EQ.3: Use (compressive) initial vertical stress

Remarks:

The elastic moduli G and K are pressure sensitive:

$$G(p) = \frac{G_0 (p - p_0)^b}{(p_{ref} - p_0)^b}$$

$$K(p) = \frac{K_0 (p - p_0)^b}{(p_{ref} - p_0)^b}$$

where G_0 and K_0 are the input values, p is the current pressure, p_0 the cut-off or datum pressure (must be zero or negative). If p attempts to fall below p_0 (i.e., more tensile) the shear stresses are set to zero and the pressure is set to p_0. Thus, the material has no stiffness or strength in tension. The pressure in compression is calculated as follows:

$$p = \left[-K_0 \ln(V) \right]^{1/(1-b)}$$

where V is the relative volume, i.e., the ratio between the original and current volume.

The constants a_0, a_1, a_2 govern the pressure sensitivity of the yield stress. Only the ratios between these values are important - the absolute stress values are taken from the stress-strain curve.

The stress strain pairs define a shear stress versus shear strain curve. The first point on the curve is assumed by default to be (0,0) and does not need to be entered. The slope of the curve must decrease with increasing γ. This curves applies at the reference pressure; at other pressures the curve is scaled by

$$\frac{\tau(p,\gamma)}{\tau(p_{ref},\gamma)} = \sqrt{\frac{\left[a_0 + a_1(p-p_0) + a_2(p-p_0)^2\right]}{\left[a_0 + a_1(p_{ref}-p_0) + a_2(p_{ref}-p_0)^2\right]}}$$

The shear stress-strain curve (with points (τ_1,γ_1), (τ_2,γ_2)...(τ_N,γ_N)) is converted into a series of N elastic perfectly-plastic curves such that $\sum(\tau_i,(\gamma)) = \tau(\gamma)$, as shown in the figure below.

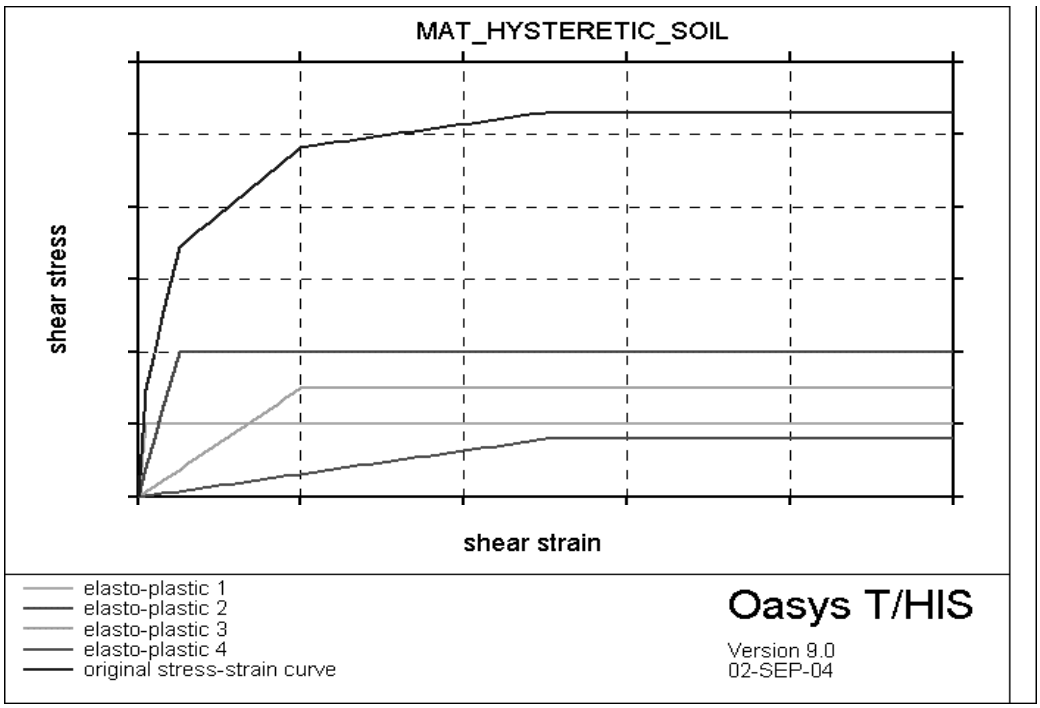


Figure 79.1

Each elastic perfectly-plastic curve represents one "layer" in the material model. Deviatoric stresses are stored and calculated separately for each layer. The yield surface for each layer is defined in terms of stress invariant J_2; this is converted internally from the input values of maximum shear stress, assuming a uniaxial stress state:

$$J_{2i} = \left(\sigma_i' : \frac{\sigma_i'}{2}\right) < \frac{4(\tau_{\max i})^2}{3}$$

where subscript i denotes layer i and τ_{maxi} is the plastic shear stress of the layer.

In cases where the deviatoric stress state is closer to pure shear, the maximum shear stress reached by the material will be up to $\sqrt{\frac{4}{3}}$ times higher than the input curve. Users may wish to allow for this by reducing the input curve by this factor. When performing checks on the output, the following relationships may be useful:

Input shear stress is treated by the material model as $0.5*\textit{Von Mises Stress} = \sqrt{\left(3\sigma_i' : \frac{\sigma_i'}{8}\right)}$

Input shear strain is treated by the material model as $1.5*\textit{Von Mises Strain} = \sqrt{\left(3\varepsilon_i' : \frac{\varepsilon_i'}{2}\right)}$

The total deviatoric stress is the sum of the deviatoric stresses in each layer. By this method, hysteretic (energy-absorbing) stress-strain curves are generated in response to any strain cycle of amplitude greater than the lowest yield strain of any layer. The example below shows response to small and large strain cycles (blue and pink lines) superposed on the input curve (thick red line).

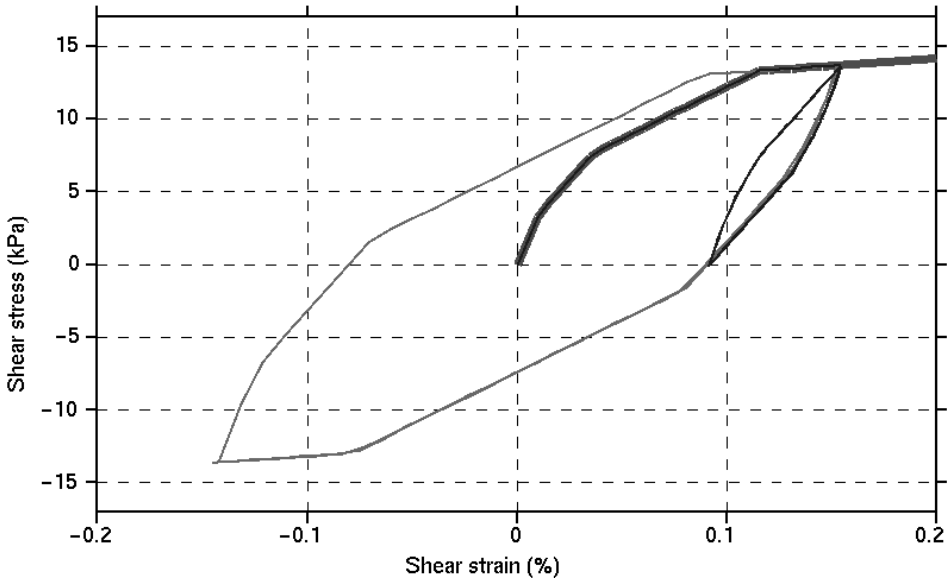


Figure 79.2

Pressure Sensitivity

The yield stresses of the layers, and hence the stress at each point on the shear stress-strain input curve, vary with pressure according to constants A0, A1 and A2. The elastic moduli, and hence also the slope of each section of shear stress-strain curve, vary with pressure according to constant B. These effects combine to modify the shear stress-strain curve according to pressure:

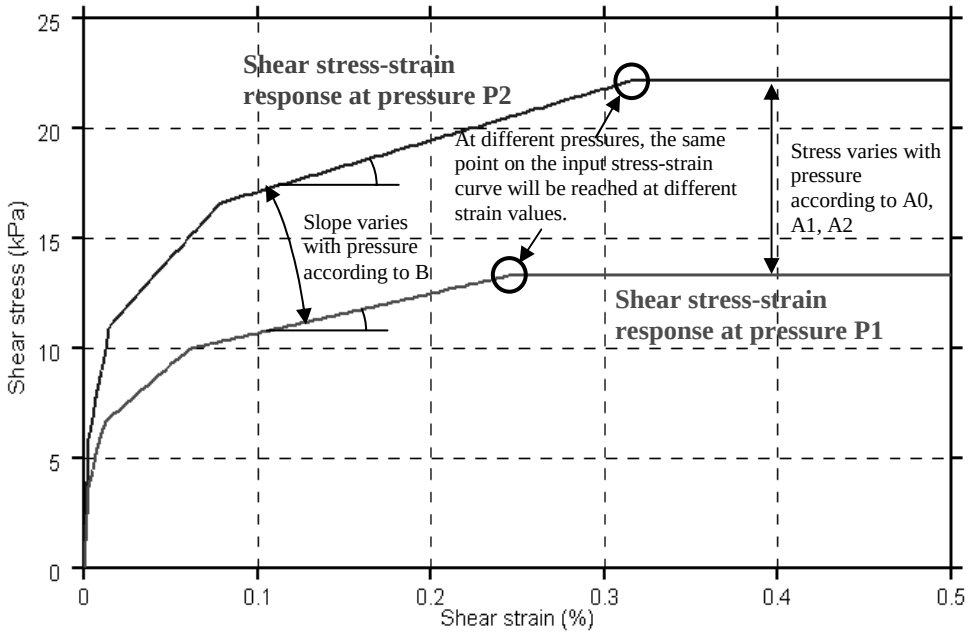


Figure 79.3

Pressure sensitivity can make the solution sensitive to numerical noise. In cases where the expected pressure changes are small compared to the initial stress state, it may be preferable to use pressure from the initial stress state instead of current pressure as the basis for the pressure sensitivity (option PINIT). This causes the bulk modulus and shear stress-strain curve to be calculated once for each element at the start of the analysis and to remain fixed thereafter. PINIT affects both stiffness (calculated using B) and strength (calculated using A0, A1 and A2). If PINIT options 2 ("plane stress" pressure) or 3 (vertical stress) are used, these quantities substitute for pressure p in the equations above. Input values of p_{ref} and p_0 should then also be "plane stress" pressure or vertical stress, respectively.

If PINIT is used, B is allowed to be as high as 1.0 (stiffness proportional to initial pressure); otherwise, values of B higher than about 0.5 are not recommended.

Dilatancy

Parameters DIL_A, DIL_B, DIL_C and DIL_D control the compaction and dilatancy that occur in sandy soils as a result of shearing motion. The dilatancy is expressed as a volume strain γ_v:

$$\varepsilon_v = \varepsilon_r + \varepsilon_g$$

$$\varepsilon_r = DIL\_A(\Gamma)^{DIL\_B}$$

$$\varepsilon_g = \frac{G^*}{DIL\_C + DIL\_D * G^*}$$

$$\Gamma = (\gamma_{xz}^2 + \gamma_{yz}^2)^{1/2}$$

$$G^* = \int (d\gamma_{xz}^2 + d\gamma_{yz}^2)^{1/2}$$

$$\gamma_{xz}, \gamma_{yz} = 2\varepsilon_{xz}, 2\varepsilon_{yz}$$

γ_r describes the dilation of the soil due to the magnitude of the shear strains; this is caused by the soil particles having to climb over each other to develop shear strain.

γ_g describes compaction of the soil due to collapse of weak areas and voids, caused by continuous shear straining.

Recommended inputs for sandy soil:

DIL_A 10
DIL_B 1.6
DIL_C -100
DIL_D -2.5

DIL_A and DIL_B may cause instabilites in some models. If this facility is used with pore water pressure, liquefaction can be modeled.

*MAT_RAMBERG-OSGOOD

This is Material Type 80. This model is intended as a simple model of shear behavior and can be used in seismic analysis.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	GAMY	TAUY	ALPHA	R	BULK	
Type	A8	F	F	F	F	F	F	
Default	none	none	none	none	none	none	none	

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
GAMY	Reference shear strain (γ_y)
TAUY	Reference shear stress (τ_y)
ALPHA	Stress coefficient (α)
R	Stress exponent (r)
BULK	Elastic bulk modulus

Remarks:

The Ramberg-Osgood equation is an empirical constitutive relation to represent the one-dimensional elastic-plastic behavior of many materials, including soils. This model allows a simple rate independent representation of the hysteretic energy dissipation observed in soils subjected to cyclic shear deformation. For monotonic loading, the stress-strain relationship is given by:

$$\frac{\gamma}{\gamma_y} = \frac{\tau}{\tau_y} + \alpha \left| \frac{\tau}{\tau_y} \right|^r \quad if \quad \gamma \geq 0$$

$$\frac{\gamma}{\gamma_y} = \frac{\tau}{\tau_y} - \alpha \left| \frac{\tau}{\tau_y} \right|^r \quad if \quad \gamma < 0$$

where γ is the shear and τ is the stress. The model approaches perfect plasticity as the stress exponent $r \to \infty$. These equations must be augmented to correctly model unloading and reloading material behavior. The first load reversal is detected by $\gamma\dot{\gamma} < 0$. After the first reversal, the stress-strain relationship is modified to

$$\frac{(\gamma - \gamma_0)}{2\gamma_y} = \frac{(\tau - \tau_0)}{2\tau_y} + \alpha \left| \frac{(\tau - \tau_0)}{2\tau_y} \right|^{\prime} \quad if \quad \gamma \geq 0$$

$$\frac{(\gamma - \gamma_0)}{2\gamma_y} = \frac{(\tau - \tau_0)}{2\tau_y} - \alpha \left| \frac{(\tau - \tau_0)}{2\tau_y} \right|^{\prime} \quad if \quad \gamma < 0$$

where γ_0 and τ_0 represent the values of strain and stress at the point of load reversal. Subsequent load reversals are detected by $(\gamma - \gamma_0)\dot{\gamma} < 0$.

The Ramberg-Osgood equations are inherently one-dimensional and are assumed to apply to shear components. To generalize this theory to the multidimensional case, it is assumed that each component of the deviatoric stress and deviatoric tensorial strain is independently related by the one-dimensional stress-strain equations. A projection is used to map the result back into deviatoric stress space if required. The volumetric behavior is elastic, and, therefore, the pressure p is found by

$$p = -K\varepsilon_v$$

where ε_v is the volumetric strain.

***MAT_PLASTICITY_WITH_DAMAGE_**{*OPTION*}

This is Material Types 81 and 82. An elasto-visco-plastic material with an arbitrary stress versus strain curve and arbitrary strain rate dependency can be defined. Damage is considered before rupture occurs. Also, failure based on a plastic strain or a minimum time step size can be defined.

Available options include:

<BLANK>

ORTHO

ORTHO_RCDC

Including ORTHO invokes an orthotropic damage model. It is implemented only for shell elements with multiple through thickness integration points and is an extension to include orthotropic damage as a means of treating failure in aluminum panels. Directional damage begins after a defined failure strain is reached in tension and continues to evolve until a tensile rupture strain is reached in either one of the two orthogonal directions. After rupture is detected at all integration points, the element is deleted. The option ORTHO_RCDC invokes the damage model developed by Wilkins [Wilkins, et al. 1977]. A nonlocal formulation, which requires additional storage, is used if a characteristic length is defined. The RCDC option, which was added at the request of Toyota, works well in predicting failure in cast aluminum see Yamasaki, et al., [2006].

Card 1

Variable	1	2	3	4	5	6	7	8
	MID	RO	E	PR	SIGY	ETAN	EPPF	TDEL
Type	A8	F	F	F	F	F	F	F
Default	none	none	none	none	none	0.0	10.E+20	10.E+20

Card 2

Variable	C	P	LCSS	LCSR	EPPFR	VP	LCDM	NUMINT
Type	F	F	F	F	F	F	F	I
Default	0	0	0	0	0	0	0	0

Card 3	1	2	3	4	5	6	7	8
Variable	EPS1	EPS2	EPS3	EPS4	EPS5	EPS6	EPS7	EPS8
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

Card 4

Variable	ES1	ES2	ES3	ES4	ES5	ES6	ES7	ES8
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

Read the following card if the option ORTHO_RCDC is active.

Card 5	1	2	3	4	5	6	7	8
Variable	ALPHA	BETA	GAMMA	D0	B	LAMBDA	DS	L
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
E	Young’s modulus.
PR	Poisson’s ratio.
SIGY	Yield stress.

VARIABLE	DESCRIPTION
ETAN	Tangent modulus, ignored if (LCSS.GT.0) is defined.
EPPF	Plastic strain, *fs*, at which material softening begins (logarithmic).
TDEL	Minimum time step size for automatic element deletion.
C	Strain rate parameter, C, see formula below.
P	Strain rate parameter, P, see formula below.
LCSS	Load curve ID or Table ID. Load curve ID defining effective stress versus effective plastic strain. If defined EPS1-EPS8 and ES1-ES8 are ignored. The table ID defines for each strain rate value a load curve ID giving the stress versus effectiveplastic strain for that rate, See Figure 24.1. The stress versus effective plastic strain curve for the lowest value of strain rate is used if the strain rate falls below the minimum value. Likewise, the stress versus effective plastic strain curve for the highest value of strain rate is used if the strain rate exceeds the maximum value. The strain rate parameters: C and P;
LCSR	Load curve ID defining strain rate scaling effect on yield stress.
EPPFR	Plastic strain at which material ruptures (logarithmic).
VP	Formulation for rate effects: EQ.0.0: Scale yield stress (default), EQ.1.0: Viscoplastic formulation.
LCDM	Load curve ID defining nonlinear damage curve.
NUMINT	Number of through thickness integration points which must fail before the element is deleted. (If zero, all points must fail.) The default of all integration points is not recommended since elements undergoing large strain are often not deleted due to nodal fiber rotations which limit strains at active integration points after most points have failed. Better results are obtained if NUMINT is set to 1 or a number less than one half of the number of through thickness points. For example, if four through thickness points are used, NUMINT should not exceed 2, even for fully integrated shells which have 16 integration points.
EPS1-EPS8	Effective plastic strain values (optional if SIGY is defined). At least 2 points should be defined.
ES1-ES8	Corresponding yield stress values to EPS1 - EPS8.
ALPHA	Parameter α. for the Rc-Dc model

VARIABLE	DESCRIPTION
BETA	Parameter β. for the Rc-Dc model
GAMMA	Parameter γ. for the Rc-Dc model
D0	Parameter D_0. for the Rc-Dc model
B	Parameter b. for the Rc-Dc model
LAMBDA	Parameter λ. for the Rc-Dc model
DS	Parameter D_s. for the Rc-Dc model
L	Optional characteristic element length for this material. We recommend that the default of 0 always be used, especially in parallel runs. If zero, nodal values of the damage function are used to compute the damage gradient. See discussion below.

Remarks:

The stress strain behavior may be treated by a bilinear stress strain curve by defining the tangent modulus, ETAN. Alternately, a curve similar to that shown in Figure 10.1 is expected to be defined by (EPS1,ES1) - (EPS8,ES8); however, an effective stress versus effective plastic strain curve (LCSS) may be input instead if eight points are insufficient. The cost is roughly the same for either approach. The most general approach is to use the table definition (LCSS) discussed below.

Two options to account for strain rate effects are possible:

I. Strain rate may be accounted for using the Cowper and Symonds model which scales the yield stress with the factor

$$1+\left(\frac{\dot{\varepsilon}}{C}\right)^{1/6}$$

where $\dot{\varepsilon}$ is the strain rate, $\dot{\varepsilon}=\sqrt{\dot{\varepsilon}_{ij}\dot{\varepsilon}_{ij}}$.

If the viscoplastic option is active, VP=1.0, and if SIGY is > 0 then the dynamic yield stress is computed from the sum of the static stress, $\sigma_y^s\left(\varepsilon_{eff}^p\right)$, which is typically given by a load curve ID, and the initial yield stress, SIGY, multiplied by the Cowper-Symonds rate term as follows:

$$\sigma_y\left(\varepsilon_{eff}^p,\dot{\varepsilon}_{eff}^p\right)=\sigma_y^s\left(\varepsilon_{eff}^p\right)+SIGY\cdot\left(\frac{\dot{\varepsilon}_{eff}^p}{C}\right)^{1/p}$$

where the plastic strain rate is used. With this latter approach similar results can be obtained between this model and material model: *MAT_ANISOTROPIC_VISCOPLASTIC. If SIGY=0, the following equation is used instead where the static stress, $\sigma_y^s\left(\varepsilon_{eff}^p\right)$, must be defined by a load curve:

$$\sigma_y\left(\varepsilon_{eff}^p,\dot{\varepsilon}_{eff}^p\right)=\sigma_y^s\left(\varepsilon_{eff}^p\right)\left[1+\left(\frac{\dot{\varepsilon}_{eff}^p}{C}\right)^{1/p}\right]$$

This latter equation is always used if the viscoplastic option is off.

II. For complete generality a load curve (LCSR) to scale the yield stress may be input instead. In this curve the scale factor versus strain rate is defined.

The constitutive properties for the damaged material are obtained from the undamaged material properties. The amount of damage evolved is represented by the constant, ω, which varies from zero if no damage has occurred to unity for complete rupture. For uniaxial loading, the nominal stress in the damaged material is given by

$$\sigma_{\text{nominal}}=\frac{P}{A}$$

where P is the applied load and A is the surface area. The true stress is given by:

$$\sigma_{true}=\frac{P}{A-A_{loss}}$$

where A_{loss} is the void area. The damage variable can then be defined:

$$\omega=\frac{A_{loss}}{A}\qquad\qquad 0\le\omega\le 1$$

In this model damage is defined in terms of plastic strain after the failure strain is exceeded:

$$\omega=\frac{\varepsilon_{eff}^p-\varepsilon_{failure}^p}{\varepsilon_{rupture}^p-\varepsilon_{failure}^p}\quad if\quad \varepsilon_{failure}^p\le\varepsilon_{eff}^p\le\varepsilon_{rupture}^p$$

After exceeding the failure strain softening begins and continues until the rupture strain is reached.

The Rc-Dc model is defined as the following:

The damage D is given by

$$D = \int \omega_1 \omega_2 d\varepsilon^p$$

where ε^p is the equivalent plastic strain,

$$\omega_1 = \left(\frac{1}{1 - \gamma \sigma_m} \right)^{\alpha}$$

is a triaxial stress weighting term and

$$\omega_2 = \left(2 - A_D\right)^{\beta}$$

is a asymmetric strain weighting term. In the above σ_m is the mean stress and

$$A_D = \min\left(\left| \frac{S_2}{S_3} \right|, \left| \frac{S_3}{S_2} \right| \right)$$

Fracture is initiated when the accumulation of damage is

$$\frac{D}{D_c} > 1$$

where D_c is the a critical damage given by

$$D_c = D_0 \left(1 + b \left| \nabla D \right|^{\lambda}\right)$$

A fracture fraction,

$$F = \frac{D - D_c}{D_s}$$

defines the degradations of the material by the Rc-Dc model.

For the Rc-Dc model the gradient of damage needs to be estimated. The damage is connected to the integration points, and, thus, the computation of the gradient requires some manipulation of the LS-DYNA source code. Provided that the damage is connected to nodes, it can be seen as a standard bilinear field and the gradient is easily obtained. To enable this, the damage at the integration points are transferred to the nodes as follows. Let E_n be the set of elements sharing node n, $\left|E_n\right|$ the number of elements in that set, P_e the set of integration points in element e and $\left|P_e\right|$ the number of points in that set. The average damage $\overline{D}_e$ in element e is computed as

$$\overline{D}_e = \frac{\sum_{p \in P_e} D_p}{\left|P_e\right|}$$

where D_p is the damage in integration point p. Finally, the damage value in node n is estimated as

$$D_n = \frac{\sum_{e \in E_n} \bar{D}_e}{|E_n|}.$$

This computation is performed in each time step and requires additional storage. Currently we use three times the total number of nodes in the model for this calculation, but this could be reduced by a considerable factor if necessary. There is an Rc-Dc option for the Gurson dilatational-plastic model. In the implementation of this model, the norm of the gradient is computed differently. Let E_f^l be the set of elements from within a distance l of element, f not including the element itself, and let $|E_f^l|$ be the number of elements in that set. The norm of the gradient of damage is estimated roughly as

$$\|\nabla D\|_f \approx \frac{1}{|E_f^l|} \sum_{e \in E_f^l} \frac{|D_e - D_f|}{d_{ef}}$$

where d_{ef} is the distance between element f and e.

The reason for taking the first approach is that it should be a better approximation of the gradient, it can for one integration point in each element be seen as a weak gradient of an elementwise constant field. The memory consumption as well as computational work should not be much higher than for the other approach.

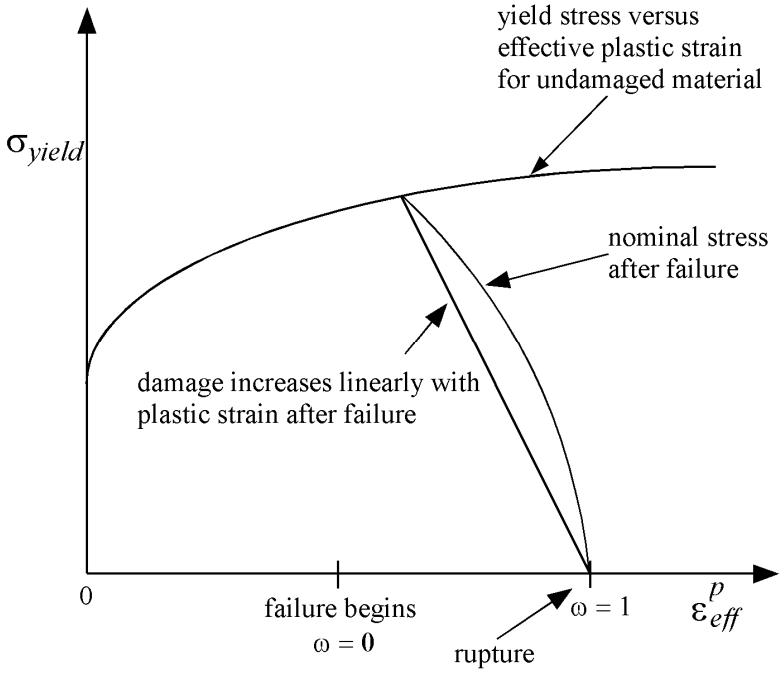


Figure 81-82.1. Stress strain behavior when damage is included.

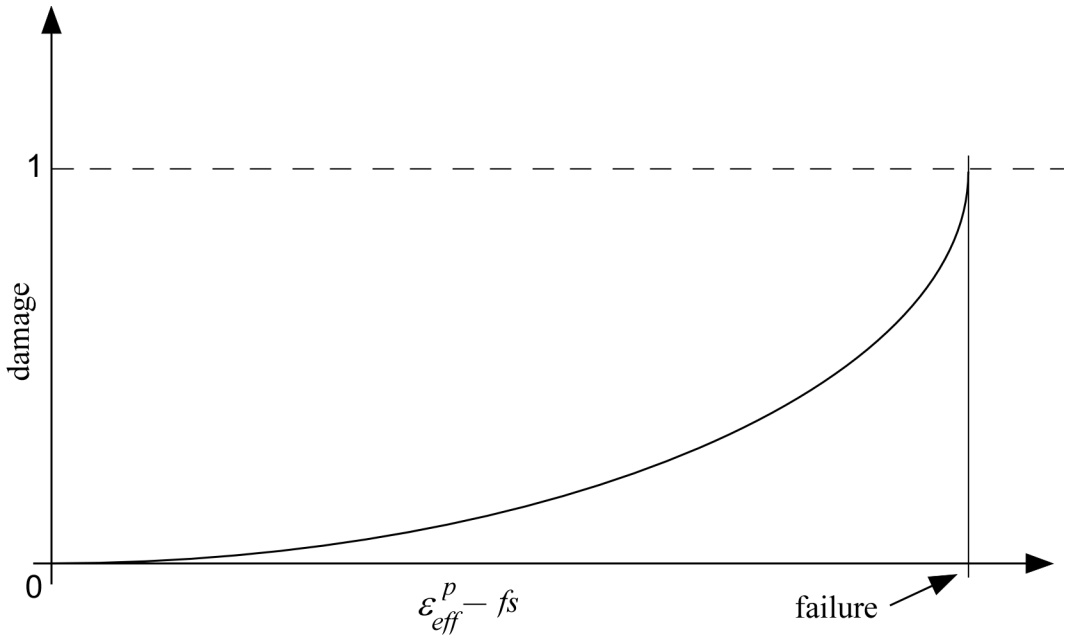


Figure 81-82.2. A nonlinear damage curve is optional. Note that the origin of the curve is at (0,0). It is permissible to input the failure strain, *fs*, as zero for this option. The nonlinear damage curve is useful for controlling the softening behavior after the failure strain is reached.

***MAT_FU_CHANG_FOAM**_*{OPTION}*

This is Material Type 83.

An available option includes:

DAMAGE_DECAY

Rate effects can be modeled in low and medium density foams, see Figure 83.1. Hysteretic unloading behavior in this model is a function of the rate sensitivity with the most rate sensitive foams providing the largest hysteresis and vice versa. The unified constitutive equations for foam materials by Chang [1995] provides the basis for this model. The mathematical description given below is excerpted from the reference. Further improvements have been incorporated based on work by Hirth, Du Bois, and Weimar [1998]. Their improvements permit: load curves generated by drop tower test to be directly input, a choice of principal or volumetric strain rates, load curves to be defined in tension, and the volumetric behavior to be specified by a load curve.

The unloading response was generalized by Kolling, Hirth, Erhart and Du Bois [2006] to allow the Mullin's effect to be modeled, i.e., after the first loading and unloading, further reloading occurs on the unloading curve. If it is desired to reload on the loading curves with the new generalized unloading, the DAMAGE decay option is available which allows the reloading to quickly return to the loading curve as the damage parameter decays back to zero in tension and compression.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E	ED	TC	FAIL	DAMP	TBID
Type	A8	F	F	F	F	F	F	F
Default	none	none	none	none	1.E+20			
Remarks								5

Card 2	1	2	3	4	5	6	7	8
Variable	BVFLAG	SFLAG	RFLAG	TFLAG	PVID	SRAF	REF	HU
Type	F	F	F	F	F	F	F	F
Default	1.0	0.0	0.0	0.0	0.0	0.0	0.0	1.0
Remarks	1	2	3		4			5

Define two additional cards, cards 3 and 4, if and only if the DAMAGE_DECAY option is inactive.

Card 3	1	2	3	4	5	6	7	8
Variable	D0	N0	N1	N2	N3	C0	C1	C2
Type	F	F	F	F	F	F	F	F
Default	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0

Card 4

Variable	C3	C4	C5	AIJ	SIJ	MINR	MAXR	SHAPE
Type	F	F	F	F	F	F	F	F
Default	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0

Define one additional card, card 3, if and only if the DAMAGE_DECAY option is active.

Card 3	MINR	MAXR	SHAPE	BETAT	BETAC	6	7	8
Variable	MINR	MAXR	SHAPE	BETAT	BETAC			
Type	F	F	F	F	F			
Default	0.0	0.0	0.0	0.0	0.0			

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
E	Young's modulus
ED	Optional Young's relaxation modulus, E_d, for rate effects. See comments below. EQ.0.0: Maximum slope in stress vs. strain curve is used. When the maximum slope is taken for the contact, the time step size for this material is reduced for stability. In some cases Δt may be significantly smaller, and defining a reasonable stiffness is recommended.
TC	Tension cut-off stress
FAIL	Failure option after cutoff stress is reached: EQ.0.0: tensile stress remains at cut-off value, EQ.1.0: tensile stress is reset to zero.
DAMP	Viscous coefficient (.05< recommended value <.50) to model damping effects.
TBID	Table ID, see *DEFINE_TABLE, for nominal stress vs. strain data as a function of strain rate. If the table ID is provided, cards 3 and 4 may be left blank and the fit will be done internally. The Table ID can be positive or negative (see remark 5 below).
BVFLAG	Bulk viscosity activation flag, see remark below: EQ.0.0: no bulk viscosity (recommended), EQ.1.0: bulk viscosity active.

VARIABLE	DESCRIPTION
SFLAG	Strain rate flag (see remark 2 below): EQ.0.0: true constant strain rate, EQ.1.0: engineering strain rate.
RFLAG	Strain rate evaluation flag: EQ.0.0: first principal direction, EQ.1.0: principal strain rates for each principal direction, EQ.2.0: volumetric strain rate.
TFLAG	Tensile stress evaluation: EQ.0.0: linear in tension. EQ.1.0: input via load curves with the tensile response corresponds to negative values of stress and strain.
PVID	Optional load curve ID defining pressure versus volumetric strain.
SRAF	Strain rate averaging flag. EQ.0.0: use weighted running average. EQ.1.0: average the last twelve values.
REF	Use reference geometry to initialize the stress tensor. The reference geometry is defined by the keyword: *INITIAL_FOAM_REFERENCE_GEOMETRY (see there for more details). EQ.0.0: off, EQ.1.0: on.
HU	Hysteretic unloading factor between 0 and 1 (default=1, i.e., no energy dissipation), see also Figure 22.57.1.
D0	material constant, see equations below.
N0	material constant, see equations below.
N1	material constant, see equations below.
N2	material constant, see equations below.
N3	material constant, see equations below.
C0	material constant, see equations below.
C1	material constant, see equations below.
C2	material constant, see equations below.
C3	material constant, see equations below.

VARIABLE	DESCRIPTION
C4	material constant, see equations below.
C5	material constant, see equations below.
AIJ,	material constant, see equations below.
SIJ	material constant, see equations below.
MINR	Ratemin, minimum strain rate of interest.
MAXR	Ratemax, maximum strain rate of interest.
SHAPE	Shape factor for unloading. Active for nonzero values of the hysteretic unloading factor HU. Values less than one reduces the energy dissipation and greater than one increases dissipation, see also Figure 57.1.
BETAT	Decay constant for damage in tension. The damage decays after loading in ceases according to $e^{-BETAT \cdot time}$.
BETAC	Decay constant for damage in compression. . The damage decays after loading in ceases according to $e^{-BETAC \cdot time}$.

Remarks:

The strain is divided into two parts: a linear part and a non-linear part of the strain

$$E(t) = E^L(t) + E^N(t)$$

and the strain rate become

$$\dot{E}(t) = \dot{E}^L(t) + \dot{E}^N(t)$$

$\dot{E}^N$ is an expression for the past history of E^N. A postulated constitutive equation may be written as:

$$\sigma(t) = \int_{\tau=0}^{\infty} \left[E_t^N(\tau), S(t) \right] d\tau$$

where $S(t)$ is the state variable and $\int_{.\tau=0}^{\infty}$ is a functional of all values of τ in $T_\tau : 0 \le \tau \le \infty$ and

$$E_t^N(\tau) = E^N(t - \tau)$$

where τ is the history parameter:

$$E_t^N(\tau=\infty) \Leftrightarrow \textit{the virgin material}$$

It is assumed that the material remembers only its immediate past, i.e., a neighborhood about $\tau=0$. Therefore, an expansion of $E_t^N(\tau)$ in a Taylor series about $\tau=0$ yields:

$$E_t^N(\tau) = E^N(0) + \frac{\partial E_t^N}{\partial t}(0)\,dt$$

Hence, the postulated constitutive equation becomes:

$$\sigma(t) = \sigma^*\left(E^N(t), \dot{E}^N(t), S(t)\right)$$

where we have replaced $\frac{\partial E_t^N}{\partial t}$ by $\dot{E}^N$, and σ^* is a function of its arguments.

For a special case,

$$\sigma(t) = \sigma^*\left(E^N(t), S(t)\right)$$

we may write

$$\dot{E}_t^N = f\left(S(t), s(t)\right)$$

which states that the nonlinear strain rate is the function of stress and a state variable which represents the history of loading. Therefore, the proposed kinetic equation for foam materials is:

$$\dot{E}_t^N = \frac{\sigma}{\|\sigma\|} D_0 \exp\left[-c_0\left(\frac{tr(\sigma S)}{(\|\sigma\|)^2}\right)^{2n_0}\right]$$

where D_0, c_0, and n_0 are material constants, and S is the overall state variable. If either $D_0 = 0$ or $c_0 \to \infty$ then the nonlinear strain rate vanishes.

$$\dot{S}_{ij} = \left[c_1\left(a_{ij}R - c_2 S_{ij}\right)P + c_3 W^{n_1}\left(\left\|\dot{E}^N\right\|\right)^{n_2} I_{ij}\right]R$$

$$R = 1 + c_4\left(\frac{\left\|\dot{E}^N\right\|}{c_5} - 1\right)^{n_3}$$

$$P = tr\left(\sigma \dot{E}^N\right)$$

$$W = \int tr\left(\sigma\,(dE)\right)$$

where c_1, c_2, c_3, c_4, c_5, n_1, n_2, n_3, and a_{ij} are material constants and:

$$\|\sigma\| = \left(\sigma_{ij}\sigma_{ij}\right)^{\frac{1}{2}}$$

$$\left\|\dot{E}\right\| = \left(\dot{E}_{ij}\dot{E}_{ij}\right)^{\frac{1}{2}}$$

$$\left\|\dot{E}^N\right\| = \left(\dot{E}^N_{ij}\dot{E}^N_{ij}\right)^{\frac{1}{2}}$$

In the implementation by Fu Chang the model was simplified such that the input constants a_{ij} and the state variables S_{ij} are scalars.

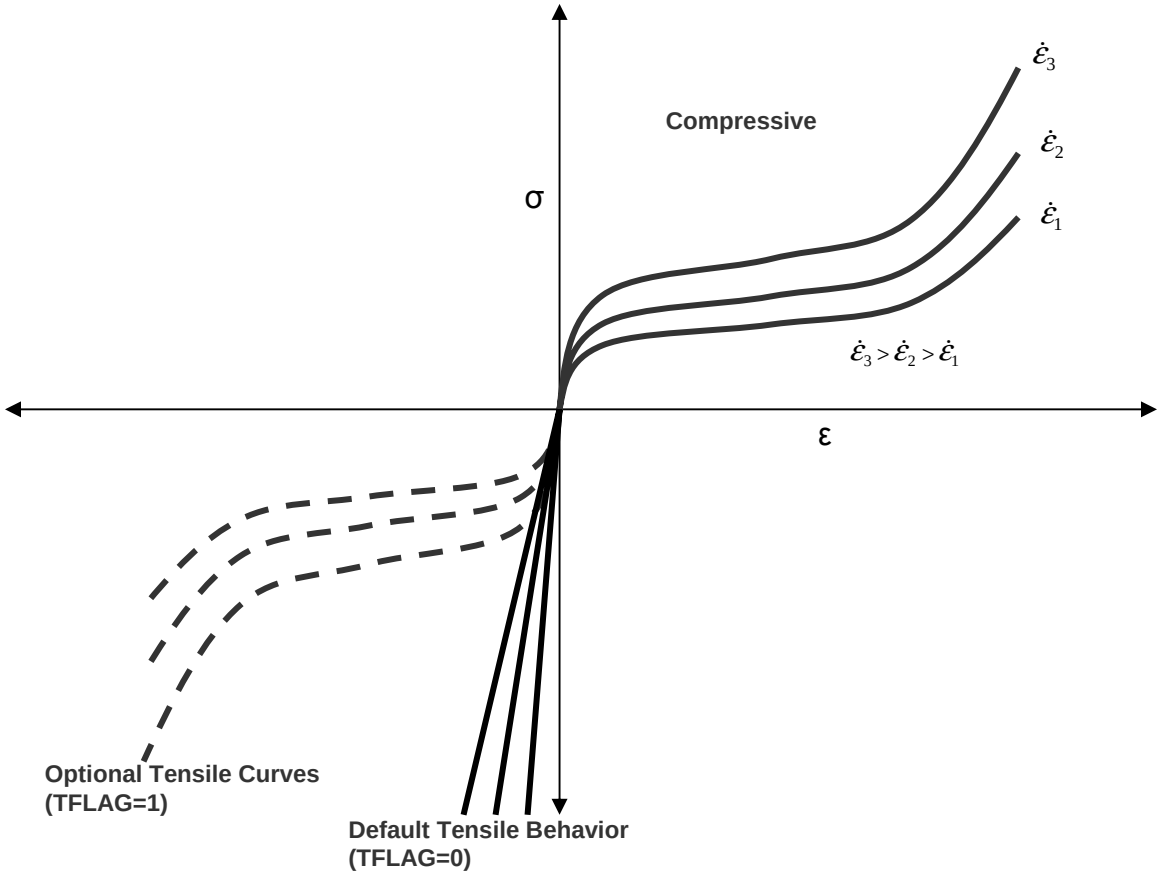


Figure 83.1. Nominal stress versus engineering strain curves, which are used to model rate effects in Fu Chang's foam model.

Additional Remarks:

1. The bulk viscosity, which generates a rate dependent pressure, may cause an unexpected volumetric response and consequently, it is optional with this model.

2. Dynamic compression tests at the strain rates of interest in vehicle crash are usually performed with a drop tower. In this test the loading velocity is nearly constant but the true strain rate, which depends on the instantaneous specimen thickness, is not. Therefore, the engineering strain rate input is optional so that the stress strain curves obtained at constant velocity loading can be used directly.

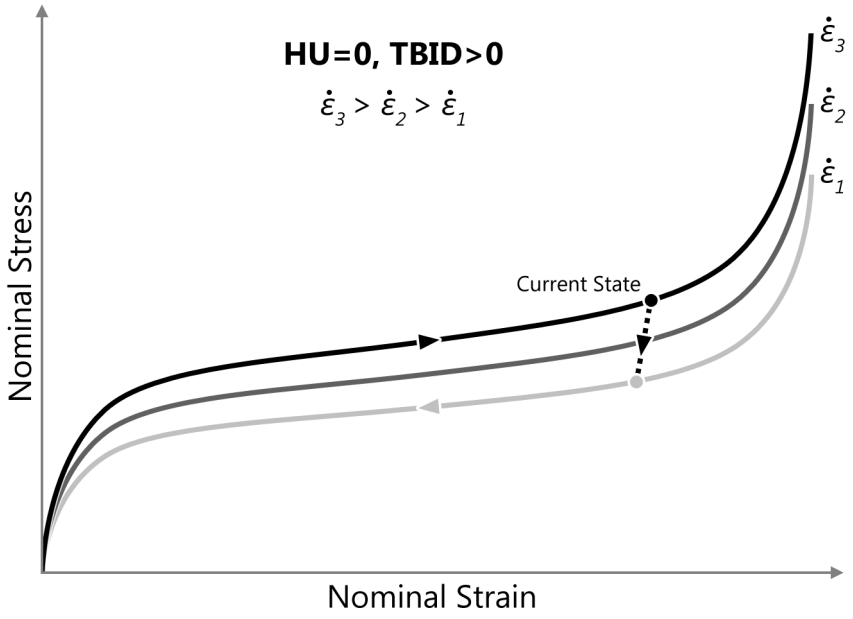
3. To further improve the response under multiaxial loading, the strain rate parameter can either be based on the principal strain rates or the volumetric strain rate.

4. Correlation under triaxial loading is achieved by directly inputting the results of hydrostatic testing in addition to the uniaxial data. Without this additional information which is fully optional, triaxial response tends to be underestimated.

5. Several options are available to control unloading response in MAT_083:

   1) HU=0 and TBID>0

   This is the old way. In this case the unloading response will follow the curve with the lowest strain rate and is rate-independent. The curve with lowest strain rate value (typically zero) in TBID should correspond to the unloading path of the material as measured in a quasistatic test. The quasistatic loading path then corresponds to a realistic (small) value of the strain rate.

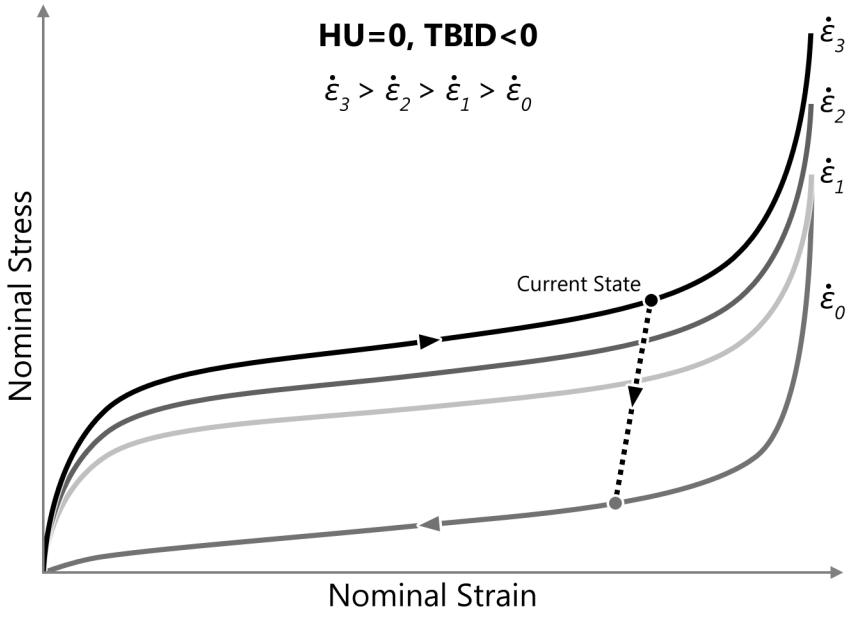


2) HU=0 and TBID<0

In this case the curve with lowest strain rate value (typically zero) in TBID must correspond to the unloading path of the material as measured in a quasistatic test. The quasistatic loading path then corresponds to a realistic (small) value of the strain rate. The quasistatic loading and unloading path (thus the first two curves of the table) should form a closed loop. The unloading response is given by a damage formulation for the principal stresses as follows:

$$\sigma_i = (1-d)\sigma_i$$

The damage parameter d is computed internally in such a way that the unloading path under uniaxial tension and compression is fitted exactly in the simulation. The unloading response is rate dependent in this case.



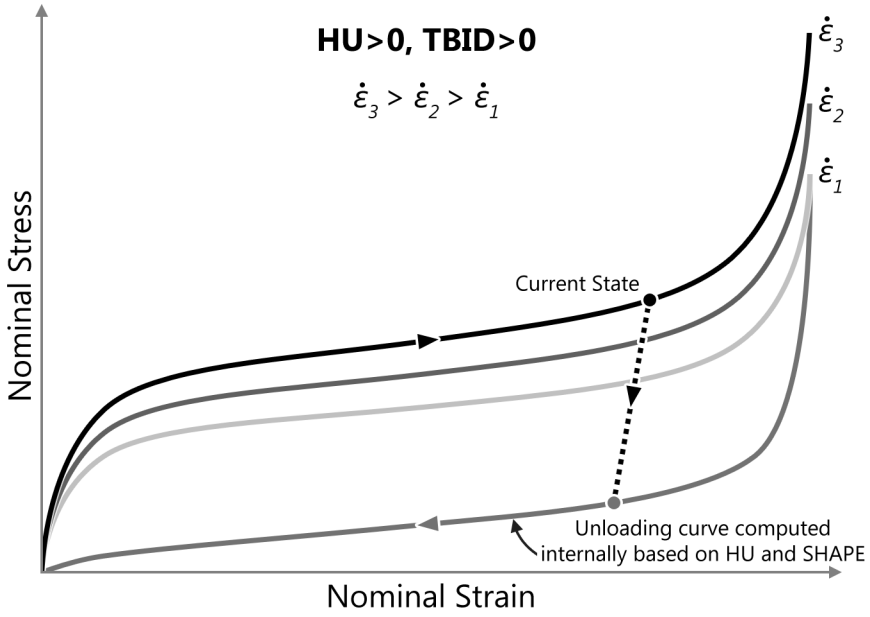
3) HU>0 and TBID>0

No unloading curve should be provided in the table and the curve with the lowest strain rate value in TBID should correspond to the loading path of the material as measured in a quasistatic test. In this case the unloading response is given by a damage formulation for the principal stresses as follows:

$$\sigma_i = (1-d)\sigma_i$$

$$d = (1-HU)\left(1-\left(\frac{W_{cur}}{W_{\max}}\right)^{SHAPE}\right),$$

where W corresponds to the current value of the hyperelastic energy per unit undeformed volume. The unloading response is rate dependent in this case.



*MAT_WINFRITH_CONCRETE

This is Material Type 84 and Material Type 85, only the former of which includes rate effects. The Winfrith concrete model is a smeared crack (sometimes known as pseudo crack), smeared rebar model, implemented in the 8-node single integration point continuum element. This model was developed by Broadhouse and Neilson [1987], and Broadhouse [1995] over many years and has been validated against experiments. The input documentation given here is taken directly form the report by Broadhouse. The Fortran subroutines and quality assurance test problems were also provided to LSTC by the Winfrith Technology Center. The rebar is defined in the section: *MAT_WINFRITH_CONCRETE_REINFORCEMENT which follows.

Card 1

1	2	3	4	5	6	7	8
MID	RO	TM	PR	UCS	UTS	FE	ASIZE
A8	F	F	F	F	F	F	F

Card 2

E	YS	EH	UELONG	RATE	CONM	CONL	CONT
F	F	F	F	F	F	F	F

Card 3

EPS1	EPS2	EPS3	EPS4	EPS5	EPS6	EPS7	EPS8
F	F	F	F	F	F	F	F

Card 4

P1	P2	P3	P4	P5	P6	P7	P8
F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
TM	Initial tangent modulus of concrete.
PR	Poisson's ratio.
UCS	Uniaxial compressive strength.
UTS	Uniaxial tensile strength.
FE	Depends on value of RATE below. RATE.EQ.0: Fracture energy (energy per unit area dissipated in opening crack). RATE.EQ.1: Crack width at which crack-normal tensile stress goes to zero.
ASIZE	Aggregate size (radius).
E	Young's modulus of rebar.
YS	Yield stress of rebar.
EH	Hardening modulus of rebar
UELONG	Ultimate elongation before rebar fails.
RATE	Rate effects: EQ.0.0: strain rate effects are included (mat 84 - recommended). EQ.1.0: strain rate effects are turned off (mat 85).
CONM	GT.0: Factor to convert model mass units to kg. EQ.-1.: Mass, length, time units in model are lbf*sec^2/in, inch, sec. EQ -2.: Mass, length, time units in model are g, cm, microsec. EQ.-3.: Mass, length, time units in model are g, mm, msec. EQ.-4.: Mass, length, time units in model are metric ton, mm, sec. EQ.-5.: Mass, length, time units in model are kg, mm, msec.
CONL	If CONM.GT.0, factor to convert model length units to meters; otherwise CONL is ignored.
CONT	If CONM.GT.0, factor to convert model time units to seconds; otherwise CONT is ignored.

VARIABLE	DESCRIPTION
EPS1,.....	Volumetric strain values (natural logarithmic values), see comments below. A maximum of 8 values are allowed. The tabulated values must completely cover the expected values in the analysis. If the first value is not for a volumetric strain value of zero then the point (0.0,0.0) will be automatically generated and up to a further nine additional values may be defined.
P1, P2,..PN	Pressures corresponding to volumetric strain values.

Remarks:

Pressure is positive in compression; volumetric strain is given by the natural log of the relative volume and is negative in compression. The tabulated data are given in order of increasing compression, with no initial zero point.

If the volume compaction curve is omitted, the following scaled curve is automatically used where p_1 is the pressure at uniaxial compressive failure from:

$$p_1 = \frac{\sigma_c}{3}$$

and K is the bulk unloading modulus computed from

$$K = \frac{E_s}{3(1-2\nu)}$$

where E_s is the input tangent modulus for concrete and ν is Poisson's ratio.

Volumetric Strain	Pressure (MPa)
$-p_1/K$	$1.00 \times p_1$
-0.002	$1.50 \times p_1$
-0.004	$3.00 \times p_1$
-0.010	$4.80 \times p_1$
-0.020	$6.00 \times p_1$
-0.030	$7.50 \times p_1$
-0.041	$9.45 \times p_1$
-0.051	$11.55 \times p_1$
-0.062	$14.25 \times p_1$
-0.094	$25.05 \times p_1$

Table 84-85.1 Default pressure versus volumetric strain curve for concrete if the curve is not defined.

The Winfrith concrete model generates an additional binary output database containing information on crack locations, directions, and widths. In order to generate the crack database, the LS-DYNA execution line is modified by adding:

q=*crf* where crf is the name of the crack database (e.g., q=DYNCRCK).

LS-Prepost can display the cracks on the deformed mesh plots. To do so, read the d3plot database into LS-Prepost and then select File > Open > Crack from the top menu bar. Or, open the crack database by adding the following to the LS-Prepost execution line:

q=*crf* where crf is the name of the crack database (e.g., q=DYNCRCK).

By default, all the cracks in visible elements are shown. You can eliminate narrow cracks from the display by setting a minimum crack width for displayed cracks. Do this by choosing Setting > Concrete Crack Width. From the top menu bar of LS-Prepost, choosing Misc > Model Info will reveal the number of cracked elements and the maximum crack width in a given plot state.

*MAT_WINFRITH_CONCRETE_REINFORCEMENT

This is Material Type 84 rebar reinforcement. Reinforcement may be defined in specific groups of elements, but it is usually more convenient to define a two-dimensional mat in a specified layer of a specified material. Reinforcement quantity is defined as the ratio of the cross-sectional area of steel relative to the cross-sectional area of concrete in the element (or layer). These cards may follow either one of two formats below and may also be defined in any order.

Option 1 (Reinforcement quantities in element groups).

	1	2	3	4	5	6	7	8
Variable	EID1	EID2	INC	XR	YR	ZR		
Type	I	I	I	F	F	F		

Option 2 (Two dimensional layers by part ID).

	1	2	3	4	5	6	7	8
Variable		PID	AXIS	COOR	RQA	RQB		
Type	blank	I	I	F	F	F		

VARIABLE	DESCRIPTION
EID1	First element ID in group.
EID2	Last element ID in group
INC	Element increment for generation.
XR	X-reinforcement quantity (for bars running parallel to global x-axis).
YR	Y-reinforcement quantity (for bars running parallel to global y-axis).
ZR	Z-reinforcement quantity (for bars running parallel to global z-axis).
PID	Part ID of reinforced elements.
AXIS	Axis normal to layer. EQ.1: A and B are parallel to global Y and Z, respectively. EQ.2: A and B are parallel to global X and Z, respectively. EQ.3: A and B are parallel to global X and Y, respectively.

VARIABLE	DESCRIPTION
COOR	Coordinate location of layer (X-coordinate if AXIS.EQ.1; Y-coordinate if AXIS.EQ.2; Z-coordinate if AXIS.EQ.3.
RQA	Reinforcement quantity (A).
RQB	Reinforcement quantity (B).

Remarks:

1. Reinforcement quantity is the ratio of area of reinforcement in an element to the element's total cross-sectional area in a given direction. This definition is true for both Options 1 and 2. Where the options differ is in the manner in which it is decided which elements are reinforced. In Option 1, the reinforced element IDs are spelled out. In Option 2, elements of part ID PID which are cut by a plane (layer) defined by AXIS and COOR are reinforced.

*MAT_ORTHOTROPIC_VISCOELASTIC

This is Material Type 86. It allows the definition of an orthotropic material with a viscoelastic part. This model applies to shell elements.

Card 1

	1	2	3	4	5	6	7	8
Variable	MID	RO	EA	EB	EC	VF	K	
Type	A8	F	F	F	F	F	F	

Card 2

Variable	G0	GINF	BETA	PRBA	PRCA	PRCB		
Type	F	F	F	F	F	F		

Card 3

Variable	GAB	GBC	GCA	AOPT	MANGLE			
Type	F	F	F	F	F			

Card 4

Variable				A1	A2	A3		
Type				F	F	F		

Card 5

Variable	V1	V2	V3	D1	D2	D3		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
EA	Young's Modulus E_a
EB	Young's Modulus E_b
EC	Young's Modulus E_c
VF	Volume fraction of viscoelastic material
K	Elastic bulk modulus
G0	G_0, short-time shear modulus
GINF	G_∞, long-time shear modulus
BETA	β, decay constant
PRBA	Poisson's ratio, ν_{ba}
PRCA	Poisson's ratio, ν_{ca}
PRCB	Poisson's ratio, ν_{cb}
GAB	Shear modulus, G_{ab}
GBC	Shear modulus, G_{bc}
GCA	Shear modulus, G_{ca}
AOPT	Material axes option (see MAT_*OPTION* TROPIC_ELASTIC for a more complete description): EQ.0.0: locally orthotropic with material axes determined by element nodes 1, 2, and 4, as with *DEFINE_COORDINATE_NODES. EQ.2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR. EQ.3.0: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle, MANGLE, from a line in the plane of the element defined by the cross product of the vector **v** with the element normal.

VARIABLE	DESCRIPTION
	LT.0.0: the absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_NODES, *DEFINE_COORDINATE_SYSTEM or *DEFINE_COORDINATE_VECTOR). Available in R3 version of 971 and later.
MANGLE	Material angle in degrees for AOPT = 3, may be overridden on the element card, see *ELEMENT_SHELL_BETA.
A1 A2 A3	Define components of vector **a** for AOPT = 2.
V1 V2 V3	Define components of vector **v** for AOPT = 3.
D1 D2 D3	Define components of vector **d** for AOPT = 2.

Remarks:

For the orthotropic definition it is referred to Material Type 2 and 21.

***MAT_CELLULAR_RUBBER**

This is Material Type 87. This material model provides a cellular rubber model with confined air pressure combined with linear viscoelasticity as outlined by Christensen [1980]. See Figure 87.1.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	PR	N				
Type	A8	F	F	I				

Card 2 if N > 0, a least squares fit is computed from uniaxial data

Card 2	1	2	3	4	5	6	7	8
Variable	SGL	SW	ST	LCID				
Type	F	F	F	F				

Card 2 if N = 0, define the following constants

Card 2	1	2	3	4	5	6	7	8
Variable	C10	C01	C11	C20	C02			
Type	F	F	F	F	F			

Card 3

Variable	P0	PHI	IVS	G	BETA			
Type	F	F	F	F	F			

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
PR	Poisson's ratio, typical values are between .0 to .2. Due to the large compressibility of air, large values of Poisson's ratio generates physically meaningless results.
N	Order of fit (currently < 3). If n>0 then a least square fit is computed with uniaxial data. The parameters given on card 2 should be specified. Also see *MAT_MOONEY_RIVLIN_RUBBER (material model 27). A Poisson's ratio of .5 is assumed for the void free rubber during the fit. The Poisson's ratio defined on Card 1 is for the cellular rubber. A void fraction formulation is used.
Define, if N > 0:	
SGL	Specimen gauge length l_0
SW	Specimen width
ST	Specimen thickness
LCID	Load curve ID giving the force versus actual change ΔL in the gauge length.
Define, if N = 0:	
C10	Coefficient, C_{10}
C01	Coefficient, C_{01}
C11	Coefficient, C_{11}
C20	Coefficient, C_{20}
C02	Coefficient, C_{02}
P0	Initial air pressure, P_0
PHI	Ratio of cellular rubber to rubber density, Φ
IVS	Initial volumetric strain, γ_0
G	Optional shear relaxation modulus, G, for rate effects (viscosity)
BETA	Optional decay constant, β_1

Remarks:

Rubber is generally considered to be fully incompressible since the bulk modulus greatly exceeds the shear modulus in magnitude. To model the rubber as an unconstrained material a hydrostatic work term, $W_H(J)$, is included in the strain energy functional which is function of the relative volume, J, [Ogden 1984]:

$$W(J_1, J_2, J) = \sum_{p,q=0}^{n} C_{pq}(J_1 - 3)^p (J_2 - 3)^q + W_H(J)$$

$$J_1 + I_1 I_3^{-1/3}$$

$$J_2 + I_2 I_3^{-2/3}$$

In order to prevent volumetric work from contributing to the hydrostatic work the first and second invariants are modified as shown. This procedure is described in more detail by Sussman and Bathe [1987].

The effects of confined air pressure in its overall response characteristics is included by augmenting the stress state within the element by the air pressure.

$$\sigma_{ij} = \sigma_{ij}^{sk} - \delta_{ij}\sigma^{air}$$

where σ_{ij}^{sk} is the bulk skeletal stress and σ^{air} is the air pressure computed from the equation:

$$\sigma^{air} = -\frac{p_0 \gamma}{1 + \gamma - \phi}$$

where p_0 is the initial foam pressure usually taken as the atmospheric pressure and γ defines the volumetric strain

$$\gamma = V - 1 + \gamma_0$$

where V is the relative volume of the voids and γ_0 is the initial volumetric strain which is typically zero. The rubber skeletal material is assumed to be incompressible.

Rate effects are taken into account through linear viscoelasticity by a convolution integral of the form:

$$\sigma_{ij} = \int_0^t g_{ijkl}(t - \tau)\frac{\partial \varepsilon_{kl}}{\partial \tau} d\tau$$

or in terms of the second Piola-Kirchhoff stress, S_{ij}, and Green's strain tensor, E_{ij},

$$S_{ij} = \int_0^t G_{ijkl}(t - \tau)\frac{\partial \varepsilon_{kl}}{\partial \tau} d\tau$$

where $g_{ijkl}(t-\tau)$ and $G_{ijkl}(t-\tau)$ are the relaxation functions for the different stress measures. This stress is added to the stress tensor determined from the strain energy functional.

Since we wish to include only simple rate effects, the relaxation function is represented by one term from the Prony series:

$$g(t) = \alpha_0 + \sum_{m=1}^{N} \alpha_m e^{-\beta t}$$

given by,

$$g(t) = E_d e^{-\beta_1 t}$$

This model is effectively a Maxwell fluid which consists of a damper and spring in series. We characterize this in the input by a shear modulus, G, and decay constant, β_1.

The Mooney-Rivlin rubber model (model 27) is obtained by specifying *n*=1 without air pressure and viscosity. In spite of the differences in formulations with Model 27, we find that the results obtained with this model are nearly identical with those of material type 27 as long as large values of Poisson's ratio are used.

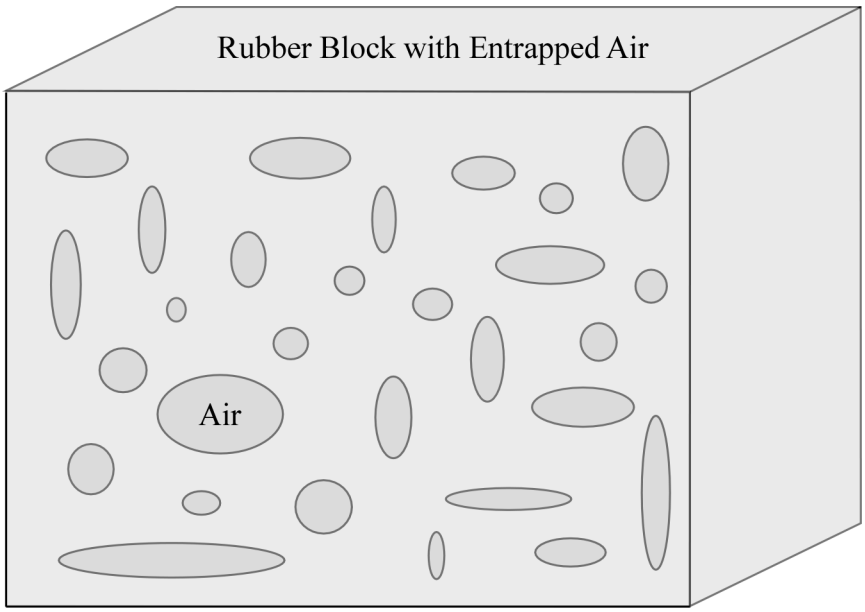


Figure 87.1. Cellular rubber with entrapped air. By setting the initial air pressure to zero, an open cell, cellular rubber can be simulated.

***MAT_MTS**

This is Material Type 88. The MTS model is due to Mauldin, Davidson, and Henninger [1990] and is available for applications involving large strains, high pressures and strain rates. As described in the foregoing reference, this model is based on dislocation mechanics and provides a better understanding of the plastic deformation process for ductile materials by using an internal state variable call the mechanical threshold stress. This kinematic quantity tracks the evolution of the material's microstructure along some arbitrary strain, strain rate, and temperature-dependent path using a differential form that balances dislocation generation and recovery processes. Given a value for the mechanical threshold stress, the flow stress is determined using either a thermal-activation-controlled or a drag-controlled kinetics relationship. An equation-of-state is required for solid elements and a bulk modulus must be defined below for shell elements.

Card 1

1	2	3	4	5	6	7	8
MID	RO	SIGA	SIGI	SIGS	SIG0	BULK	
A8	F	F	F	F	F	F	

Card 2

HF0	HF1	HF2	SIGS0	EDOTS0	BURG	CAPA	BOLTZ
F	F	F	F	F	F	F	F

Card 3

SM0	SM1	SM2	EDOT0	GO	PINV	QINV	EDOTI
F	F	F	F	F	F	F	F

Card 4	1	2	3	4	5	6	7	8
Variable	G0I	PINVI	QINVI	EDOTS	G0S	PINVS	QINVS	
Type	F	F	F	F	F	F	F	

Card 5

Variable	RHOCPR	TEMPRF	ALPHA	EPS0				
Type	F	F						

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
SIGA	$\hat{\sigma}_a$, dislocation interactions with long-range barriers (force/area).
SIGI	$\hat{\sigma}_i$, dislocation interactions with interstitial atoms (force/area).
SIGS	$\hat{\sigma}_s$, dislocation interactions with solute atoms (force/area).
SIG0	$\hat{\sigma}_0$, initial value of $\hat{\sigma}$ at zero plastic strain (force/area) NOT USED.
HF0	a_0, dislocation generation material constant (force/area).
HF1	a_1, dislocation generation material constant (force/area).
HF2	a_2, dislocation generation material constant (force/area).
SIGS0	$\hat{\sigma}_{\varepsilon so}$, saturation threshold stress at 0^o K (force/area).
BULK	Bulk modulus defined for shell elements only. Do not input for solid elements.
EDOTS0	$\dot{\varepsilon}_{\varepsilon so}$, reference strain-rate (time^{-1}).

VARIABLE	DESCRIPTION
BURG	Magnitude of Burgers vector (interatomic slip distance), (distance)
CAPA	Material constant, A.
BOLTZ	Boltzmann's constant, k (energy/degree).
SM0	G_0, shear modulus at zero degrees Kelvin (force/area).
SM1	b_1, shear modulus constant (force/area).
SM2	b_2, shear modulus constant (degree).
EDOT0	$\dot{\varepsilon}_o$, reference strain-rate (time^{-1}).
G0	g_0, normalized activation energy for a dislocation/dislocation interaction.
PINV	$\frac{1}{p}$, material constant.
QINV	$\frac{1}{q}$, material constant.
EDOTI	$\dot{\varepsilon}_{o,i}$, reference strain-rate (time^{-1}).
G0I	$g_{0,i}$, normalized activation energy for a dislocation/interstitial interaction.
PINVI	$\frac{1}{p_i}$, material constant.
QINVI	$\frac{1}{q_i}$, material constant.
EDOTS	$\dot{\varepsilon}_{o,s}$, reference strain-rate (time^{-1}).
G0S	$g_{0,s}$ normalized activation energy for a dislocation/solute interaction.
PINVS	$\frac{1}{p_s}$, material constant.

VARIABLE	DESCRIPTION
QINVS	$\frac{1}{q_s}$, material constant.
RHOCPR	ρc_p, product of density and specific heat.
TEMPRF	T_{ref}, initial element temperature in degrees K.
ALPHA	α, material constant (typical value is between 0 and 2).
EPS0	ε_o, factor to normalize strain rate in the calculation of Θ_o. (Use 1., 10^{-3}, or 10^{-6} for time units of seconds, milliseconds, or microseconds, respectively.)

Remarks:

The flow stress σ is given by:

$$\sigma = \hat{\sigma}_a + \frac{G}{G_0}\left[s_{th}\hat{\sigma} + s_{th,i}\hat{\sigma}_i + s_{th,s}\hat{\sigma}_s \right]$$

The first product in the equation for τ contains a micro-structure evolution variable, i.e., $\hat{\sigma}$, called the *Mechanical Threshold Stress* (MTS), that is multiplied by a constant-structure deformation variable s_{th}: s_{th} is a function of absolute temperature T and the plastic strain-rates $\dot{\varepsilon}^p$. The evolution equation for $\hat{\sigma}$ is a differential hardening law representing dislocation-dislocation interactions:

$$\frac{\partial}{\partial \varepsilon^p} \equiv \Theta_o \left[1 - \frac{\tanh\left(\alpha \frac{\hat{\sigma}}{\hat{\sigma}_{\varepsilon s}} \right)}{\tanh(\alpha)} \right]$$

The term, $\frac{\partial \hat{\sigma}}{\partial \varepsilon^p}$, represents the hardening due to dislocation generation and the stress ratio, $\frac{\hat{\sigma}}{\hat{\sigma}_{\varepsilon s}}$, represents softening due to dislocation recovery. The threshold stress at zero strain-hardening $\hat{\sigma}_{\varepsilon s}$ is called the saturation threshold stress. Relationships for Θ_o, $\hat{\sigma}_{\varepsilon s}$ are:

$$\Theta_o = a_o + a_1 \ln\left(\frac{\dot{\varepsilon}^p}{\varepsilon_0} \right) + a_2 \sqrt{\frac{\dot{\varepsilon}^p}{\varepsilon_0}}$$

which contains the material constants, a_o, a_1, and a_2. The constant, $\hat{\sigma}_{\varepsilon s}$, is given as:

$$\hat{\sigma}_{\varepsilon s} = \hat{\sigma}_{\varepsilon so}\left(\frac{\dot{\varepsilon}^p}{\dot{\varepsilon}_{\varepsilon so}}\right)^{kT/Gb^3A}$$

which contains the input constants: $\hat{\sigma}_{\varepsilon so}$, $\dot{\varepsilon}_{\varepsilon so}$, b, A, and k. The shear modulus G appearing in these equations is assumed to be a function of temperature and is given by the correlation.

$$G = G_0 - b_1/\left(e^{b_2/T} - 1\right)$$

which contains the constants: G_0, b_1, and b_2. For thermal-activation controlled deformation s_{th} is evaluated via an Arrhenius rate equation of the form:

$$s_{th} = \left[1 - \left(\frac{kT1n\left(\frac{\dot{\varepsilon}_0}{\dot{\varepsilon}^p}\right)}{Gb^3 g_0}\right)^{\frac{1}{q}}\right]^{\frac{1}{p}}$$

The absolute temperature is given as:

$$T = T_{ref} + \frac{E}{\rho c_p}$$

where E is the internal energy density per unit initial volume.

*MAT_PLASTICITY_POLYMER

This is Material Type 89. An elasto-plastic material with an arbitrary stress versus strain curve and arbitrary strain rate dependency can be defined. It is intended for applications where the elastic and plastic sections of the response are not as clearly distinguishable as they are for metals. Rate dependency of failure strain is included. Many polymers show a more brittle response at high rates of strain. The material model is currently available only for solid and shell elements.

Card 1

Variable	MID	RO	E	PR				
Type	A8	F	F	F				
Default	none	none	none	none				

Card 2

Variable	C	P	LCSS	LCSR				
Type	F	F	F	F				
Default	0	0	0	0				

Card 3

Variable	EFTX	DAMP	RATEFAC	LCFAIL				
Type	F	F	F	F				
Default	0	0	0	0				

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.
C	Strain rate parameter, C, (Cowper Symonds).
P	Strain rate parameter, P, (Cowper Symonds).
LCSS	Load curve ID defining effective stress versus total effective strain.
LCSR	Load curve ID defining strain rate scaling effect on yield stress.
EFTX	Failure flag. EQ.0.0: failure determined by maximum tensile strain (default), EQ.1.0: failure determined only by tensile strain in local x direction, EQ.2.0: failure determined only by tensile strain in local y direction.
DAMP	Stiffness-proportional damping ratio. Typical values are 1e-3 or 1e-4. If set too high instabilities can result.
RATEFAC	Filtering factor for strain rate effects. Must be between 0 (no filtering) and 1 (infinite filtering). The filter is a simple low pass filter to remove high frequency oscillation from the strain rates before they are used in rate effect calculations. The cut off frequency of the filter is [(1 - RATEFAC) / timestep] rad/sec.
LCFAIL	Load curve ID giving variation of failure strain with strain rate. The points on the x-axis should be natural log of strain rate, the y-axis should be the true strain to failure. Typically this is measured by uniaxial tensile test, and the strain values converted to true strain.

Remarks:

1. Unlike other LS-DYNA material models, both the input stress-strain curve and the strain to failure are defined as total true strain, not plastic strain. The input can be defined from uniaxial tensile tests; nominal stress and nominal strain from the tests must be converted to true stress and true strain. The elastic component of strain must not be subtracted out.

2. The stress-strain curve is permitted to have sections steeper (i.e. stiffer) than the elastic modulus. When these are encountered the elastic modulus is increased to prevent spurious energy generation.

3. Sixty-four bit precision is recommended when using this material model, especially if the strains become high.

4. Invariant shell numbering is recommended when using this material model. See *CONTROL_ACCURACY.

5. Damage in the material begins when the “failure strain” is reached, i.e., when extra history variable 8 reaches a value of 1.0. The element is then progressively softened via a damage model until history variable 8 reaches a value of 1.1 at which point the element is deleted. In other words, the element is deleted at 1.1 times the failure strain.

*MAT_ACOUSTIC

This is Material Type 90. This model is appropriate for tracking low pressure stress waves in an acoustic media such as air or water and can be used only with the acoustic pressure element formulation. The acoustic pressure element requires only one unknown per node. This element is very cost effective. Optionally, cavitation can be allowed.

Card 1

Variable	MID	RO	C	BETA	CF	ATMOS	GRAV	
Type	A8	F	F	F	F	F	F	

Card 2

Variable	XP	YP	ZP	XN	YN	ZN		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
C	Sound speed
BETA	Damping factor. Recommend values are between 0.1 and 1.0.
CF	Cavitation flag: EQ.0.0: off, EQ.1.0: on.
ATMOS	Atmospheric pressure (optional)
GRAV	Gravitational acceleration constant (optional)
XP	x-coordinate of free surface point
YP	y-coordinate of free surface point
ZP	z-coordinate of free surface point

VARIABLE	DESCRIPTION
XN	x-direction cosine of free surface normal vector
YN	y-direction cosine of free surface normal vector
ZN	z-direction cosine of free surface normal vector

***MAT_SOFT_TISSUE_**{*OPTION*}

Available options include:

<BLANK>

VISCO

This is Material Type 91 (*OPTION*=<BLANK>) or Material Type 92 (*OPTION*=VISCO). This material is a transversely isotropic hyperelastic model for representing biological soft tissues such as ligaments, tendons, and fascia. The representation provides an isotropic Mooney-Rivlin matrix reinforced by fibers having a strain energy contribution with the qualitative material behavior of collagen. The model has a viscoelasticity option which activates a six-term Prony series kernel for the relaxation function. In this case, the hyperelastic strain energy represents the elastic (long-time) response. See Weiss et al. [1996] and Puso and Weiss [1998] for additional details. The material is available for use with brick and shell elements. When used with shell elements, the Belytschko-Tsay formulation (#2) must be selected.

Card 1

Variable	MID	RO	C1	C2	C3	C4	C5	
Type	A8	F	F	F	F	F	F	

Card 2

Variable	XK	XLAM	FANG	XLAM0	FAILSF	FAILSM	FAILSHR	
Type	F	F	F	F	F	F	F	

Card 3

Variable	AOPT	AX	AY	AZ	BX	BY	BZ	
Type	F	F	F	F	F	F	F	

Card 4	1	2	3	4	5	6	7	8
Variable	LA1	LA2	LA3	MACF				
Type	F	F	F	I				

Define the following two cards only for the VISCO option:

Card 5	1	2	3	4	5	6	7	8
Variable	S1	S2	S3	S4	S5	S6		
Type	F	F	F	F	F	F		

Card 6

Variable	T1	T2	T3	T4	T5	T6		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
C1 - C5	Hyperelastic coefficients (see equations below)
XK	Bulk Modulus
XLAM	Stretch ratio at which fibers are straightened
FANG	Fiber angle in local shell coordinate system (shells only)
XLAM0	Initial fiber stretch (optional)
FAILSF	Stretch ratio for ligament fibers at failure (applies to shell elements only). If zero, failure is not considered.

VARIABLE	DESCRIPTION
FAILSM	Stretch ratio for surrounding matrix material at failure (applies to shell elements only). If zero, failure is not considered.
FAILSHR	Shear strain at failure at a material point (applies to shell elements only). If zero, failure is not considered. This failure value is independent of FAILSF and FAILSM.
AOPT	Material axes option, see Figure 2.1 (bricks only): EQ.0.0: locally orthotropic with material axes determined by element nodes as shown in Figure 2.1. Nodes 1, 2, and 4 of an element are identical to the nodes used for the definition of a coordinate system as by *DEFINE_COORDINATE_NODES. EQ.1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center; this is the a-direction. This option is for solid elements only. EQ.2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR. EQ.3.0: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle, BETA, from a line in the plane of the element defined by the cross product of the vector **v** with the element normal. The plane of a solid element is the midsurface between the inner surface and outer surface defined by the first four nodes and the last four nodes of the connectivity of the element, respectively. EQ.4.0: locally orthotropic in cylindrical coordinate system with the material axes determined by a vector **v**, and an originating point, P, which define the centerline axis. This option is for solid elements only. LT.0.0: the absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_NODES, *DEFINE_COORDINATE_SYSTEM or *DEFINE_COORDINATE_VECTOR). Available in R3 version of 971 and later.
AX, AY, AZ	Equal to XP,YP,ZP for AOPT=1, Equal to A1,A2,A3 for AOPT=2, Equal to V1,V2,V3 for AOPT=3 or 4.
BX, BY, BZ	Equal to D1,D2,D3 for AOPT=2 Equal to XP,YP,ZP for AOPT=4
LAX, LAY, LAZ	Local fiber orientation vector (bricks only)

VARIABLE	DESCRIPTION
MACF	Material axes change flag for brick elements: EQ.1: No change, default, EQ.2: switch material axes a and b, EQ.3: switch material axes a and c, EQ.4: switch material axes b and c.
T1 - T6	Characteristic times for Prony series relaxation kernel (*OPTION*=**VISCO**)

Remarks:

The overall strain energy W is "uncoupled" and includes two isotropic deviatoric matrix terms, a fiber term F, and a bulk term:

$$W = C_1\left(\tilde{I}_1 - 3\right) + C_2\left(\tilde{I}_2 - 3\right) + F\left(\lambda\right) + \tfrac{1}{2}K\left[\ln\left(J\right)\right]^2$$

Here, $\tilde{I}_1$ and $\tilde{I}_2$ are the deviatoric invariants of the right Cauchy deformation tensor, λ is the deviatoric part of the stretch along the current fiber direction, and $J = \det \mathbf{F}$ is the volume ratio. The material coefficients C_1 and C_2 are the Mooney-Rivlin coefficients, while K is the effective bulk modulus of the material (input parameter XK).

The derivatives of the fiber term F are defined to capture the behavior of crimped collagen. The fibers are assumed to be unable to resist compressive loading - thus the model is isotropic when $\lambda < 1$. An exponential function describes the straightening of the fibers, while a linear function describes the behavior of the fibers once they are straightened past a critical fiber stretch level $\lambda \geq \lambda^*$ (input parameter XLAM):

$$\frac{\partial F}{\partial \lambda} = \begin{Bmatrix} 0 & \lambda < 1 \\ \frac{C_3}{\lambda}\left[\exp\left(C_4\left(\lambda - 1\right)\right) - 1\right] & \lambda < \lambda^* \\ \frac{1}{\lambda}\left(C_5\lambda + C_6\right) & \lambda \geq \lambda^* \end{Bmatrix}$$

Coefficients C_3, C_4, and C_5 must be defined by the user. C_6 is determined by LS-DYNA to ensure stress continuity at $\lambda = \lambda^*$. Sample values for the material coefficients $C_1 - C_5$ and λ^* for ligament tissue can be found in Quapp and Weiss [1998]. The bulk modulus K should be at least 3 orders of magnitude larger than C_1 to ensure near-incompressible material behavior.

Viscoelasticity is included via a convolution integral representation for the time-dependent second Piola-Kirchoff stress $\mathbf{S}(\mathbf{C},t)$:

$$\mathbf{S}\left(\mathbf{C},t\right) = \mathbf{S}^e\left(\mathbf{C}\right) + \int_0^t 2G\left(t-s\right)\frac{\partial W}{\partial \mathbf{C}\left(s\right)}ds$$

Here, $\mathbf{S}^e$ is the elastic part of the second PK stress as derived from the strain energy, and $G(t-s)$ is the reduced relaxation function, represented by a Prony series:

$$G(t) = \sum_{i=1}^{6} S_i \exp\left(\frac{t}{T_i}\right)$$

Puso and Weiss [1998] describe a graphical method to fit the Prony series coefficients to relaxation data that approximates the behavior of the continuous relaxation function proposed by Y-C. Fung, as quasilinear viscoelasticity.

Remarks on Input Parameters:

Cards 1 through 4 must be included for both shell and brick elements, although for shells cards 3 and 4 are ignored and may be blank lines.

For shell elements, the fiber direction lies in the plane of the element. The local axis is defined by a vector between nodes n1 and n2, and the fiber direction may be offset from this axis by an angle FANG.

For brick elements, the local coordinate system is defined using the convention described previously for *MAT_ORTHOTROPIC_ELASTIC. The fiber direction is oriented in the local system using input parameters LAX, LAY, and LAZ. By default, (LAX,LAY,LAZ) = (1,0,0) and the fiber is aligned with the local x-direction.

An optional initial fiber stretch can be specified using XLAM0. The initial stretch is applied during the first time step. This creates preload in the model as soft tissue contacts and equilibrium is established. For example, a ligament tissue "uncrimping strain" of 3% can be represented with initial stretch value of 1.03.

If the **VISCO** option is selected, at least one Prony series term (S1,T1) must be defined.

*MAT_ELASTIC_6DOF_SPRING_DISCRETE_BEAM

This is Material Type 93. This material model is defined for simulating the effects of nonlinear elastic and nonlinear viscous beams by using six springs each acting about one of the six local degrees-of-freedom. The input consists of part ID's that reference material type, *MAT_ELASTIC_SPRING_DISCRETE_BEAM above (type 74 above). Generally, these referenced parts are used only for the definition of this material model and are not referenced by any elements. The two nodes defining a beam may be coincident to give a zero length beam, or offset to give a finite length beam. For finite length discrete beams the absolute value of the variable SCOOR in the SECTION_BEAM input should be set to a value of 2.0, which causes the local r-axis to be aligned along the two nodes of the beam to give physically correct behavior. The distance between the nodes of a beam should not affect the behavior of this material model. A triad is used to orient the beam for the directional springs.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	TPIDR	TPIDS	TPIDT	RPIDR	RPIDS	RPIDT
Type	A8	F	I	I	I	I	I	I

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density, see also volume in *SECTION_BEAM definition.
TPIDR	Translational motion in the local r-direction is governed by part ID TPIDR. If zero, no force is computed in this direction.
TPIDS	Translational motion in the local s-direction is governed by part ID TPIDS. If zero, no force is computed in this direction.
TPIDT	Translational motion in the local t-direction is governed by part ID TPIDT. If zero, no force is computed in this direction.
RPIDR	Rotational motion about the local r-axis is governed by part ID RPIDR. If zero, no moment is computed about this axis.
RPIDS	Rotational motion about the local s-axis is governed by part ID RPIDS. If zero, no moment is computed about this axis.
RPIDT	Rotational motion about the local t-axis is governed by part ID RPIDT. If zero, no moment is computed about this axis.

*MAT_INELASTIC_SPRING_DISCRETE_BEAM

This is Material Type 94. This model permits elastoplastic springs with damping to be represented with a discrete beam element type 6. A yield force versus deflection curve is used which can vary in tension and compression.

Card 1

Variable	MID	RO	K	F0	D	CDF	TDF	
Type	A8	F	F	F	F	F	F	

Card 2

Variable	FLCID	HLCID	C1	C2	DLE	GLCID		
Type	F	F	F	F	F	I		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density, see also volume in *SECTION_BEAM definition.
K	Elastic loading/unloading stiffness. This is required input.
F0	Optional initial force. This option is inactive if this material is referenced in a part referenced by material type *MAT_INELASTIC_6DOF_SPRING
D	Optional viscous damping coefficient.
CDF	Compressive displacement at failure. Input as a positive number. After failure, no forces are carried. This option does not apply to zero length springs. EQ.0.0: inactive.
TDF	Tensile displacement at failure. After failure, no forces are carried. EQ.0.0: inactive.

VARIABLE	DESCRIPTION
FLCID	Load curve ID, see *DEFINE_CURVE, defining the yield force versus plastic deflection. If the origin of the curve is at (0,0) the force magnitude is identical in tension and compression, i.e., only the sign changes. If not, the yield stress in the compression is used when the spring force is negative. The plastic displacement increases monotonically in this implementation. The load curve is required input.
HLCID	Load curve ID, see *DEFINE_CURVE, defining force versus relative velocity (Optional). If the origin of the curve is at (0,0) the force magnitude is identical for a given magnitude of the relative velocity, i.e., only the sign changes.
C1	Damping coefficient.
C2	Damping coefficient
DLE	Factor to scale time units.
GLCID	Optional load curve ID, see *DEFINE_CURVE, defining a scale factor versus deflection for load curve ID, HLCID. If zero, a scale factor of unity is assumed.

Remarks:

The yield force is taken from the load curve:

$$F^Y = F_y\left(\Delta L^{plastic}\right)$$

where $L^{plastic}$ is the plastic deflection. A trial force is computed as:

$$F^T = F^n + K\Delta\dot{L}\left(\Delta t\right)$$

and is checked against the yield force to determine F:

$$F = \begin{cases} F^Y \; if \; F^T > F^Y \\ F^T \; if \; F^T \le F^Y \end{cases}$$

The final force, which includes rate effects and damping, is given by:

$$F^{n+1} = F \cdot \left[1 + C1 \cdot \Delta\dot{L} + C2 \cdot \mathrm{sgn}\left(\Delta\dot{L}\right)\ln\left(\max\left\{1., \frac{\left|\Delta\dot{L}\right|}{DLE}\right\}\right)\right] + D\Delta\dot{L} + g\left(\Delta L\right)h\left(\Delta\dot{L}\right)$$

Unless the origin of the curve starts at (0,0), the negative part of the curve is used when the spring force is negative where the negative of the plastic displacement is used to interpolate, F_y.

The positive part of the curve is used whenever the force is positive. In these equations, ΔL is the change in length

$$\Delta L = current\ length - initial\ length$$

The cross sectional area is defined on the section card for the discrete beam elements, See *SECTION_BEAM. The square root of this area is used as the contact thickness offset if these elements are included in the contact treatment.

*MAT_INELASTIC_6DOF_SPRING_DISCRETE_BEAM

This is Material Type 95. This material model is defined for simulating the effects of nonlinear inelastic and nonlinear viscous beams by using six springs each acting about one of the six local degrees-of-freedom. The input consists of part ID's that reference material type, *MAT_ INELASTIC_SPRING_DISCRETE_BEAM above (type 94). Generally, these referenced parts are used only for the definition of this material model and are not referenced by any elements. The two nodes defining a beam may be coincident to give a zero length beam, or offset to give a finite length beam. For finite length discrete beams the absolute value of the variable SCOOR in the SECTION_BEAM input should be set to a value of 2.0, which causes the local r-axis to be aligned along the two nodes of the beam to give physically correct behavior. The distance between the nodes of a beam should not affect the behavior of this material model. A triad must be used to orient the beam for zero length beams.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	TPIDR	TPIDS	TPIDT	RPIDR	RPIDS	RPIDT
Type	A8	F	I	I	I	I	I	I

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density, see also volume in *SECTION_BEAM definition.
TPIDR	Translational motion in the local r-direction is governed by part ID TPIDR. If zero, no force is computed in this direction.
TPIDS	Translational motion in the local s-direction is governed by part ID TPIDS. If zero, no force is computed in this direction.
TPIDT	Translational motion in the local t-direction is governed by part ID TPIDT. If zero, no force is computed in this direction.
RPIDR	Rotational motion about the local r-axis is governed by part ID RPIDR. If zero, no moment is computed about this axis.
RPIDS	Rotational motion about the local s-axis is governed by part ID RPIDS. If zero, no moment is computed about this axis.
RPIDT	Rotational motion about the local t-axis is governed by part ID RPIDT. If zero, no moment is computed about this axis.

*MAT_BRITTLE_DAMAGE

This is Material Type 96.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	TLIMIT	SLIMIT	FTOUGH	SRETEN
Type	A8	F	F	F	F	F	F	F

Card 2

Variable	VISC	FRA_RF	E_RF	YS_RF	EH_RF	FS_RF	SIGY	
Type	F	F	F	F	F	F	F	

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.
TLIMIT	Tensile limit.
SLIMIT	Shear limit.
FTOUGH	Fracture toughness.
SRETEN	Shear retention.
VISC	Viscosity.
FRA_RF	Fraction of reinforcement in section.
E_RF	Young's modulus of reinforcement.
YS_RF	Yield stress of reinforcement.

VARIABLE	DESCRIPTION
EH_RF	Hardening modulus of reinforcement.
FS_RF	Failure strain (true) of reinforcement.
SIGY	Compressive yield stress. EQ.0: no compressive yield

Remarks:

A full description of the tensile and shear damage parts of this material model is given in Govindjee, Kay and Simo [1994,1995]. It is an anisotropic brittle damage model designed primarily for concrete though it can be applied to a wide variety of brittle materials. It admits progressive degradation of tensile and shear strengths across smeared cracks that are initiated under tensile loadings. Compressive failure is governed by a simplistic J2 flow correction that can be disabled if not desired. Damage is handled by treating the rank 4 elastic stiffness tensor as an evolving internal variable for the material. Softening induced mesh dependencies are handled by a characteristic length method [Oliver 1989].

Description of properties:

1. E is the Young's modulus of the undamaged material also known as the virgin modulus.

2. υ is the Poisson's ratio of the undamaged material also known as the virgin Poisson's ratio.

3. f_n is the initial principal tensile strength (stress) of the material. Once this stress has been reached at a point in the body a smeared crack is initiated there with a normal that is co-linear with the 1st principal direction. Once initiated, the crack is fixed at that location, though it will convect with the motion of the body. As the loading progresses the allowed tensile traction normal to the crack plane is progressively degraded to a small machine dependent constant.

   The degradation is implemented by reducing the material's modulus normal to the smeared crack plane according to a maximum dissipation law that incorporates exponential softening. The restriction on the normal tractions is given by

$$\phi_t = (\mathbf{n} \otimes \mathbf{n}) : \boldsymbol{\sigma} - f_n + (1-\varepsilon)\, f_n \left(1 - \exp\left[-H\alpha\right]\right) \le 0$$

   where $\mathbf{n}$ is the smeared crack normal, ε is the small constant, H is the softening modulus, and α is an internal variable. H is set automatically by the program; see g_c below. α measures the crack field intensity and is output in the equivalent plastic strain field, $\bar{\varepsilon}^p$, in a normalized fashion.

   The evolution of alpha is governed by a maximum dissipation argument. When the normalized value reaches unity it means that the material's strength has been reduced to

2% of its original value in the normal and parallel directions to the smeared crack. Note that for plotting purposes it is never output greater than 5.

4. f_s is the initial shear traction that may be transmitted across a smeared crack plane. The shear traction is limited to be less than or equal to $f_s(1-\beta)(1-\exp[-H\alpha])$, through the use of two orthogonal shear damage surfaces. Note that the shear degradation is coupled to the tensile degradation through the internal variable alpha which measures the intensity of the crack field. β is the shear retention factor defined below. The shear degradation is taken care of by reducing the material's shear stiffness parallel to the smeared crack plane.

5. g_c is the fracture toughness of the material. It should be entered as fracture energy per unit area crack advance. Once entered the softening modulus is automatically calculated based on element and crack geometries.

6. β is the shear retention factor. As the damage progresses the shear tractions allowed across the smeared crack plane asymptote to the product βf_s.

7. η represents the viscosity of the material. Viscous behavior is implemented as a simple Perzyna regularization method. This allows for the inclusion of first order rate effects. The use of some viscosity is recommend as it serves as regularizing parameter that increases the stability of calculations.

8. σ_y is a uniaxial compressive yield stress. A check on compressive stresses is made using the J2 yield function $\mathbf{s}:\mathbf{s}-\sqrt{\frac{2}{3}}\sigma_y \le 0$, where $\mathbf{s}$ is the stress deviator. If violated, a J2 return mapping correction is executed. This check is executed when (1) no damage has taken place at an integration point yet, (2) when damage has taken place at a point but the crack is currently closed, and (3) during active damage after the damage integration (i.e. as an operator split). Note that if the crack is open the plasticity correction is done in the plane-stress subspace of the crack plane.

Remark: A variety of experimental data has been replicated using this model from quasi-static to explosive situations. Reasonable properties for a standard grade concrete would be E=3.15x10^6 psi, f_n=450 psi, f_s=2100 psi, ν = 0.2, g_c = 0.8 lbs/in, β =0.03, η = 0.0 psi-sec, σ_y = 4200 psi. For stability, values of η between 104 to 106 psi/sec are recommended. Our limited experience thus far has shown that many problems require nonzero values of η to run to avoid error terminations.

Remark: Various other internal variables such as crack orientations and degraded stiffness tensors are internally calculated but currently not available for output.

*MAT_GENERAL_JOINT_DISCRETE_BEAM

This is Material Type 97. This model is used to define a general joint constraining any combination of degrees of freedom between two nodes. The nodes may belong to rigid or deformable bodies. In most applications the end nodes of the beam are coincident and the local coordinate system (r,s,t axes) is defined by CID (see *SECTION_BEAM).

Card 1

Variable	MID	RO	TR	TS	TT	RR	RS	RT
Type	A8	F	I	I	I	I	I	
Remarks	1							

Card 2

Variable	RPST	RPSR						
Type	F	F						
Remarks	2							

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density, see also volume in *SECTION_BEAM definition.
TR	Translational constraint code along the r-axis (0 => free, 1 => constrained)
TS	Translational constraint code along the s-axis (0 => free, 1 => constrained)
TT	Translational constraint code along the y-axis (0 => free, 1 =>constrained)
RR	Rotational constraint code along the r-axis (0 => free, 1 => constrained)
RS	Rotational constraint code along the r-axis (0 => free, 1 => constrained)

VARIABLE	DESCRIPTION
RT	Rotational constraint code about the t-axis (0 => free, 1 => constrained)
RPST	Penalty stiffness scale factor for translational constraints.
RPSR	Penalty stiffness scale factor for rotational constraints.

Remarks:

1. For explicit calculations, the additional stiffness due to this joint may require addition mass and inertia for stability. Mass and rotary inertia for this beam element is based on the defined mass density, the volume, and the mass moment of inertia defined in the *SECTION_BEAM input.

2. The penalty stiffness applies to explicit calculations. For implicit calculations, constraint equations are generated and imposed on the system equations; therefore, these constants, RPST and RPSR, are not used.

*MAT_SIMPLIFIED_JOHNSON_COOK

This is Material Type 98. The Johnson/Cook strain sensitive plasticity is used for problems where the strain rates vary over a large range. In this simplified model, thermal effects and damage are ignored, and the maximum stress is directly limited since thermal softening which is very significant in reducing the yield stress under adiabatic loading is not available. An iterative plane stress update is used for the shell elements, but due to the simplifications related to thermal softening and damage, this model is 50% faster than the full Johnson/Cook implementation. To compensate for the lack of thermal softening, limiting stress values are used to keep the stresses within reasonable limits. A resultant formulation for the Belytschko-Tsay, the C0 Triangle, and the fully integrated type 16 shell elements is activated by specifying either zero or one through thickness integration point on the *SHELL_SECTION card. This latter option is less accurate than through thickness integration but is somewhat faster. Since the stresses are not computed in the resultant formulation, the stress output to the databases for the resultant elements are zero. This model is also available for the Hughes-Liu beam, the Belytschko-Schwer beam, and the truss element. For the resultant beam formulation, the rate effects are approximated by the axial rate since the thickness of the beam about it bending axes is unknown. The linear bulk modulus is used to determine the pressure in the elements, since the use of this model is primarily for structural analysis.

Card 1

Variable	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	VP			
Type	A8	F	F	F	F			
Default	none	none	none	none	0.0			

Card 2

Variable	A	B	N	C	PSFAIL	SIGMAX	SIGSAT	EPSO
Type	F	F	F	F	F	F	F	F
Default	none	0.0	0.0	0.0	1.0E+17	SIGSAT	1.0E+28	1.0

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
E	Young's modulus
PR	Poisson's ratio
VP	Formulation for rate effects: EQ.0.0: Scale yield stress (default), EQ.1.0: Viscoplastic formulation. This option applies only to the 4-node shell and 8-node thick shell if and only if through thickness integration is used.
A	See equations below.
B	See equations below.
N	See equations below.
C	See equations below.
PSFAIL	Effective plastic strain at failure. If zero failure is not considered.
SIGMAX	Maximum stress obtainable from work hardening before rate effects are added (optional). This option is ignored if VP=1.0. This option is ignored if VP=1.0.
SIGSAT	Saturation stress which limits the maximum value of effective stress which can develop after rate effects are added (optional).
εPSO	Strain rate normalization factor. This value depends on the time units. Typically, input 1 for units of seconds, 0.001 for units of milliseconds, 0.000001 for microseconds, etc.

Remarks:

Johnson and Cook express the flow stress as

$$\sigma_y = \left(A + B\,\bar{\varepsilon}^{\,p^n} \right) \left(1 + c\,\ln \dot{\varepsilon}^* \right)$$

where

A, B, C and n are input constants

$\bar{\varepsilon}^{\,p}$ effective plastic strain

$\dot{\varepsilon}* = \dfrac{\dot{\bar{\varepsilon}}}{\dot{\varepsilon}_0}$ effective strain rate for $\dot{\varepsilon}_0 = 1\, s^{-1}$

The maximum stress is limited by *sigmax* and *sigsat* by:

$$\sigma_y = min\left\{ \min\left[A + B\bar{\varepsilon}^{p^n}, sigmax \right]\left(1 + c \ln \dot{\varepsilon}* \right), sigsat \right\}$$

Failure occurs when the effective plastic strain exceeds *psfail*.

If the viscoplastic option is active, VP=1.0, the parameters SIGMAX and SIGSAT are ignored since these parameters make convergence of the viscoplastic strain iteration loop difficult to achieve. The viscoplastic option replaces the plastic strain in the forgoing equations by the viscoplastic strain and the strain rate by the viscoplastic strain rate. Numerical noise is substantially reduced by the viscoplastic formulation.

*MAT_SIMPLIFIED_JOHNSON_COOK_ORTHOTROPIC_DAMAGE

This is Material Type 99. This model, which is implemented with multiple through thickness integration points, is an extension of model 98 to include orthotropic damage as a means of treating failure in aluminum panels. Directional damage begins after a defined failure strain is reached in tension and continues to evolve until a tensile rupture strain is reached in either one of the two orthogonal directions. After rupture is detected at NUMINT integration points, the element is deleted.

Card 1

Variable	MID	RO	E	PR	VP	EPPFR	LCDM	NUMINT
Type	A8	F	F	F	F	F	I	I
Default	none	none	none	none	0.0	1.e+16	optional	all intg. pt

Card 2

Variable	A	B	N	C	PSFAIL	SIGMAX	SIGSAT	EPSO
Type	F	F	F	F	F	F	F	F
Default	none	0.0	0.0	0.0	1.0E+17	SIGSAT	1.0E+28	1.0

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
E	Young's modulus
PR	Poisson's ratio
VP	Formulation for rate effects: EQ.0.0: Scale yield stress (default), EQ.1.0: Viscoplastic formulation. This option applies only to the 4-node shell and 8-node thick shell if and only if through thickness integration is used.

VARIABLE	DESCRIPTION
EPPFR	Plastic strain at which material ruptures (logarithmic).
LCDM	Load curve ID defining nonlinear damage curve. See Figure 81-82.2.
NUMINT	Number of through thickness integration points which must fail before the element is deleted. (If zero, all points must fail.) The default of all integration points is not recommended since elements undergoing large strain are often not deleted due to nodal fiber rotations which limit 0strains at active integration points after most points have failed. Better results are obtained if NUMINT is set to 1 or a number less than one half of the number of through thickness points. For example, if four through thickness points are used, NUMINT should not exceed 2, even for fully integrated shells which have 16 integration points.
A	See equations below.
B	See equations below.
N	See equations below.
C	See equations below.
PSFAIL	Principal plastic strain at failure. If zero failure is not considered.
SIGMAX	Maximum stress obtainable from work hardening before rate effects are added (optional). This option is ignored if VP=1.0. This option is ignored if VP=1.0.
SIGSAT	Saturation stress which limits the maximum value of effective stress which can develop after rate effects are added (optional).
εPSO	Effective plastic strain rate. This value depends on the time units. Typically, input 1 for units of seconds, 0.001 for units of milliseconds, 0.000001 for microseconds, etc.

Remarks:

See the description for the SIMPLIFIED_JOHNSON_COOK model above.

***MAT_SPOTWELD_**{*OPTION*}

This is Material Type 100. The material model applies to beam element type 9 and to solid element type 1 with type 6 hourglass controls. The failure models apply to both beam and solid elements.

The beam elements, based on the Hughes-Liu beam formulation, may be placed between any two deformable shell surfaces and tied with constraint contact, *CONTACT_SPOTWELD, which eliminates the need to have adjacent nodes at spot weld locations. Beam spot welds may be placed between rigid bodies and rigid/deformable bodies by making the node on one end of the spot weld a rigid body node which can be an extra node for the rigid body, see *CONSTRAINED_ EXTRA_NODES_*OPTION*. In the same way rigid bodies may also be tied together with this spot weld option. This weld option should not be used with rigid body switching. The foregoing advice is valid if solid element spot welds are used; however, since the solid elements have just three degrees-of-freedom at each node, *CONTACT_TIED_SURFACE_TO_SURFACE must be used instead of *CONTACT_SPOTWELD.

In flat topologies the shell elements have an unconstrained drilling degree-of-freedom which prevents torsional forces from being transmitted. If the torsional forces are deemed to be important, brick elements should be used to model the spot welds.

Beam and solid element force resultants for MAT_SPOTWELD are written to the spot weld force file, SWFORC, and the file for element stresses and resultants for designated elements, ELOUT.

It is advisable to include all spot welds, which provide the slave nodes, and spot welded materials, which define the master segments, within a single *CONTACT_SPOTWELD interface for beam element spot welds or a *CONTACT_TIED_SURFACE_TO_SURFACE interface for solid element spot welds. As a constraint method these interfaces are treated independently which can lead to significant problems if such interfaces share common nodal points. An added benefit is that memory usage can be substantially less with a single interface.

Available options include:

<BLANK>

DAMAGE-FAILURE

The DAMAGE-FAILURE option causes one additional line to be read with the damage parameter and a flag that determines how failure is computed from the resultants. On this line the parameter, RS, if nonzero, invokes damage mechanics combined with the plasticity model to achieve a smooth drop off of the resultant forces prior to the removal of the spot weld. The parameter OPT determines the method used in computing resultant based failure, which is unrelated to damage.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	SIGY	ET	DT	TFAIL
Type	A8	F	F	F	F	F	F	F

Define this card as Card 2 when the DAMAGE-FAILURE option is inactive. Card 3 is not defined

Card 2	1	2	3	4	5	6	7	8
Variable	EFAIL	NRR	NRS	NRT	MRR	MSS	MTT	NF
Type	F	F	F	F	F	F	F	F

Define cards 2 and 3 below if the DAMAGE-FAILURE option is active.
OPT=-1.0 and 0.0, Resultant based failure.

Card 2	1	2	3	4	5	6	7	8
Variable	EFAIL	NRR	NRS	NRT	MRR	MSS	MTT	NF
Type	F	F	F	F	F	F	F	F

OPT=1.0,.Stress based failure.

Card 2	1	2	3	4	5	6	7	8
Variable	EFAIL	SIGAX	SIGTAU					NF
Type	F	F	F					F

OPT=1.0, Stress based failure if strain rate effects are included

Card 2	1	2	3	4	5	6	7	8
Variable	EFAIL	-LCAX	-LCTAU					NF
Type	F	F	F					F

OPT=2.0, User subroutine for failure

Card 2	1	2	3	4	5	6	7	8
Variable	EFAIL	USERV1	USERV2	USERV3	USERV4	USERV5	USERV6	NF
Type	F	F	F	F	F	F	F	F

OPT=3.0, 4.0

Card 2	1	2	3	4	5	6	7	8
Variable	EFAIL	ZD	ZT	ZALP1	ZALP2	ZALP3	ZRRAD	NF
Type	F	F	F	F	F	F	F	F

OPT=5.0

Card 2	1	2	3	4	5	6	7	8
Variable	EFAIL	ZD	ZT	ZT2				
Type	F	F	F	F				

OPT=6.0, 7.0, and 9.0

Card 2	1	2	3	4	5	6	7	8
Variable	EFAIL							NF
Type	F							F

Card 3 is defined only for the DAMAGE-FAILURE option.

Card 3	1	2	3	4	5	6	7	8
Variable	RS	OPT	FVAL	TRUE_T		BETA		
Type	F	F	F	F		F		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
E	Young's modulus
PR	Poisson's ratio
SIGY	SIGY.GT.0: Initial yield stress. SIGY.LT.0: A yield curve or table is assigned by |SIGY|. This option is available for beams and starting with release 971 R5.
ET	Hardening modulus, E_t
DT	Time step size for mass scaling, Δt
TFAIL	Failure time if nonzero. If zero this option is ignored.
EFAIL	Effective plastic strain in weld material at failure. If the damage option is inactive, the spot weld element is deleted when the plastic strain at each integration point exceeds EFAIL. If the damage option is active, the plastic strain must exceed the rupture strain at each integration point before deletion occurs.
NRR	Axial force resultant N_{rr_F} or maximum axial stress σ_{rr}^F at failure depending on the value of OPT (see below). If zero, failure due to this component is not considered. If negative, |NRR| is the load curve ID defining the maximum axial stress at failure as a function of the effective strain rate.
NRS	Force resultant N_{rs_F} or maximum shear stress τ^F at failure depending on the value of OPT (see below). If zero, failure due to this component is not considered. If negative, |NRS| is the load curve ID defining the maximum shear stress at failure as a function of the effective strain rate.

VARIABLE	DESCRIPTION
NRT	Force resultant N_{rt_F} at failure. If zero, failure due to this component is not considered.
MRR	Torsional moment resultant M_{rr_F} at failure. If zero, failure due to this component is not considered.
MSS	Moment resultant M_{ss_F} at failure. If zero, failure due to this component is not considered.
MTT	Moment resultant M_{tt_F} at failure. If zero, failure due to this component is not considered.
NF	Number of force vectors stored for filtering. The default value is set to zero which is generally recommended unless oscillatory resultant forces are observed in the time history databases. Even though these welds should not oscillate significantly, this option was added for consistency with the other spot weld options. NF affects the storage since it is necessary to store the resultant forces as history variables. When NF is nonzero, the resultants in the output databases are filtered. NF cannot exceed 30.
SIGAX	Maximum axial stress σ_{rr}^F at failure. If zero, failure due to this component is not considered.
SIGTAU	Maximum shear stress τ^F at failure. If zero, failure due to this component is not considered.
LCAX	Load curve ID defining the maximum axial stress at failure as a function of the effective strain rate. Input as a negative number.
LCTAU	Load curve ID defining the maximum shear stress at failure as a function of the effective strain rate. Input as a negative number.
USERVn	Failure constants for user failure subroutine, n=1,2,...6.
ZD	Notch diameter
ZT	Sheet thickness.
ZALP1	Correction factor alpha1
ZALP2	Correction factor alpha2
ZALP3	Correction factor alpha3

VARIABLE	DESCRIPTION
ZRRAD	Notch root radius (OPT=3.0 only).
ZT2	Second sheet thickness (OPT = 5.0 only)
RS	Rupture strain. Define if and only if damage is active.
OPT	Failure option: EQ.-2: same as option –1 but in addition, the peak value of the failure criteria and the time it occurs is stored and is written into the SWFORC database. This information may be necessary since the instantaneous values written at specified time intervals may miss the peaks. Additional storage is allocated to store this information. EQ.-1: resultant based failure criteria, *FC*, is computed based on the force and moment resultants and is written into the SWFORC file. *Failure is not allowed*. This allows easy identification of vulnerable spot welds in the post-processing. Failure is likely to occur if *FC* >1.0. Only the terms where the corresponding failure resultant is nonzero are included when FC is calculated. This option applies to both solid and beam elements.

$$FC = \sqrt{\left(\frac{max\left(N_{rr},0\right)}{N_{rr_F}}\right)^2 + \left(\frac{N_{rs}}{N_{rs_F}}\right)^2 + \left(\frac{N_{rt}}{N_{rt_F}}\right)^2 + \left(\frac{M_{rr}}{M_{rr_F}}\right)^2 + \left(\frac{M_{ss}}{M_{ss_F}}\right)^2 + \left(\frac{M_{tt}}{M_{tt_F}}\right)^2}$$

EQ. 0: resultant based failure
EQ. 1: stress based failure computed from resultants (Toyota)
EQ. 2: user subroutine uweldfail to determine failure
EQ. 3: notch stress based failure (beam weld only)
EQ. 4: stress intensity factor at failure (beam weld only)
EQ. 5: structural stress at failure (beam weld only).
EQ. 6: stress based failure computed from resultants (Toyota). In this option a shell strain rate dependent failure model is used (beam weld only). The static failure stresses are defined by part ID using the input given in the keyword definition *DEFINE_SPOTWELD_ RUPTURE_STRESS input.
EQ. 7: stress based failure for solid elements (Toyota) with peak stresses computed from resultants, and strength values input for pairs of parts, see *DEFINE_SPOTWELD_FAILURE_ RESULTANTS. Strain rate effects are optional.
EQ. 8: Not used.
EQ. 9: Stress based failure from resultants (Toyota). In this option a shell strain rate dependent failure model is used (beam weld only). The static failure stresses are defined by part ID using the input given in the keyword definition *DEFINE_SPOTWELD_ RUPTURE_PARAMETER input.

VARIABLE	DESCRIPTION
OPT	EQ. 12. user subroutine uweldfail12 with 22 material constants to determine damage and failure. EQ. 14. user subroutine uweldfail22 with 22 material constants to determine failure.
FVAL	Failure parameter. If OPT: EQ.-2: Not used. EQ.-1: Not used. EQ. 0: Not used. EQ. 1: Not used. EQ. 2: Not used. EQ. 3: Notch stress value at failure (σ_{KF}). EQ. 4: Stress intensity factor value at failure (K_{eqF}). EQ. 5: Structural stress value at failure (σ_{sF}). EQ. 6: Number of cycles that failure condition must be met to trigger beam deletion. EQ. 7: Not used. EQ. 9: Number of cycles that failure condition must be met to trigger beam deletion.
TRUE_T	True weld thickness. This optional value is available for solid element failure by OPT=0,1,7, or –2. TRUE_T is used to reduce the moment contribution to the failure calculation from artificially thick weld elements so shear failure can be modeled more accurately. See comments under the remarks for *MAT_SPOTWELD_DAIMLER CHRYSLER
BETA	Damage model decay rate.

Remarks:

The weld material is modeled with isotropic hardening plasticity coupled to failure models. EFAIL specifies a failure strain which fails each integration point in the spot weld independently. The resultant-based failure model fails the entire weld if the resultants are outside of the failure surface defined by:

$$\left(\frac{max\left(N_{rr},0\right)}{N_{rr_F}}\right)^2+\left(\frac{N_{rs}}{N_{rs_F}}\right)^2+\left(\frac{N_{rt}}{N_{rt_F}}\right)^2+\left(\frac{M_{rr}}{M_{rr_F}}\right)^2+\left(\frac{M_{ss}}{M_{ss_F}}\right)^2+\left(\frac{M_{tt}}{M_{tt_F}}\right)^2-1=0$$

where the *numerators* in the equation are the resultants calculated in the local coordinates of the cross section, and the **denominators** are the values specified in the input. If NF is nonzero the resultants are filtered before failure is checked. The stress based failure model (OPT=1), which was developed by *Toyota Motor Corporation* and is based on the peak axial and transverse shear stresses, fails the entire weld if the stresses are outside of the failure surface defined by

$$\left(\frac{\sigma_{rr}}{\sigma_{rr}^F}\right)^2+\left(\frac{\tau}{\tau^F}\right)^2-1=0$$

If strain rates are considered then the failure criteria becomes:

$$\left(\frac{\sigma_{rr}}{\sigma_{rr}^F\left(\dot{\varepsilon}_{eff}\right)}\right)^2+\left(\frac{\tau}{\tau^F\left(\dot{\varepsilon}_{eff}\right)}\right)^2-1=0$$

where $\sigma_{rr}^F\left(\dot{\varepsilon}_{eff}\right)$ and $\tau^F\left(\dot{\varepsilon}_{eff}\right)$ are defined by load curves LCAX and LCTAU. The peak stresses are calculated from the resultants using simple beam theory.

$$\sigma_{rr}=\frac{N_{rr}}{A}+\frac{\sqrt{M_{ss}^2+M_{tt}^2}}{Z}\quad \tau=\frac{M_{rr}}{2Z}+\frac{\sqrt{N_{rs}^2+N_{rt}^2}}{A}$$

where the area and section modulus are given by:

$$A=\pi\frac{d^2}{4}$$
$$Z=\pi\frac{d^3}{32}$$

and *d* is the equivalent diameter of the beam element or solid element used as a spot weld.

OPT=2 invokes a user-written subroutine *uweldfail*, documented in Appendix Q.

The failure based on notch stress (OPT=3), see Zhang [1999], occurs when the failure criterion:

$$\sigma_k-\sigma_{kF}\ge 0$$

is satisfied. The notch stress is give by the equation:

$$\sigma_k=\alpha_1\frac{4F}{\pi dt}\left(1+\frac{\sqrt{3}+\sqrt{19}}{8\sqrt{\pi}}\sqrt{\frac{t}{\rho}}\right)+\alpha_2\frac{6M}{\pi dt^2}\left(1+\frac{2}{\sqrt{3\pi}}\sqrt{\frac{t}{\rho}}\right)+\alpha_3\frac{4F_{rr}}{\pi d^2}\left(1+\frac{5}{3\sqrt{2\pi}}\frac{d}{t}\sqrt{\frac{t}{\rho}}\right)$$

Here,

$$F=\sqrt{F_{rs}^2+F_{rt}^2}$$
$$M=\sqrt{M_{ss}^2+M_{tt}^2}$$

and $\alpha_i\quad i=1,2,3$ are input corrections factors with default values of unity. If spot welds are between sheets of unequal thickness, the minimum thickness of the spot welded sheets may be introduced as a crude approximation.

The failure based on structural stress intensity (OPT=4) occurs, see Zhang [1999], when the failure criterion:

$$K_{eq} - K_{eqF} \geq 0$$

is satisfied where

$$K_{eq} = \sqrt{K_I^2 + K_{II}^2}$$

and

$$K_I = \alpha_1 \frac{\sqrt{3}F}{2\pi d\sqrt{t}} + \alpha_2 \frac{2\sqrt{3}M}{\pi dt\sqrt{t}} + \alpha_3 \frac{5\sqrt{2}F_{rr}}{3\pi d\sqrt{t}}$$

$$K_{II} = \alpha_1 \frac{2F}{\pi d\sqrt{t}}$$

Here, F and M are as defined above for the notch stress formulas and again, $\alpha_i \quad i = 1,2,3$ are input corrections factors with default values of unity. If spot welds are between sheets of unequal thickness, the minimum thickness of the spot welded sheets may be used as a crude approximation.

The maximum structural stress at the spot weld was utilized successfully for predicting the fatigue failure of spot welds, see Rupp, et. al. [1994] and Sheppard [1993]. The corresponding results according to Rupp, et. al. are listed below where it is assumed that they may be suitable for crash conditions.

The failure criterion invoked by OPT=5 is given by:

$$\max\left(\sigma_{v1}, \sigma_{v2}, \sigma_{v3}\right) - \sigma_{sF} = 0$$

where σ_{sF} is the critical value of structural stress at failure. It is noted that the forces and moments in the equations below are referred to the beam nodes 1, 2, and to the midpoint, respectively. The three stress values, $\sigma_{v1}, \sigma_{v2}, \sigma_{v3}$, are defined by:

$$\sigma_{v1}(\zeta) = \frac{F_{rs1}}{\pi dt_1}\cos\zeta + \frac{F_{rt1}}{\pi dt_1}\sin\zeta - \frac{1.046\beta_1 F_{rr1}}{t_1\sqrt{t_1}} - \frac{1.123M_{ss1}}{dt_1\sqrt{t_1}}\sin\zeta + \frac{1.123M_{tt1}}{dt_1\sqrt{t_1}}\cos\zeta \quad \text{with}$$

$$\beta_1 = 0 \quad if \quad F_{rr1} \leq 0$$
$$\beta_1 = 1 \quad if \quad F_{rr1} > 0$$

$$\sigma_{v2}(\zeta) = \frac{F_{rs2}}{\pi dt_2}\cos\zeta + \frac{F_{rt2}}{\pi dt_2}\sin\zeta - \frac{1.046\beta_1 F_{rr2}}{t_2\sqrt{t_2}} + \frac{1.123M_{ss2}}{dt_2\sqrt{t_2}}\sin\zeta - \frac{1.123M_{tt2}}{dt_2\sqrt{t_2}}\cos\zeta \quad \text{with}$$

$$\beta_2 = 0 \quad if \quad F_{rr2} \leq 0$$
$$\beta_2 = 1 \quad if \quad F_{rr2} > 0$$

$$\sigma_{v3}(\zeta) = 0.5\sigma(\zeta) + 0.5\sigma(\zeta)\cos(2\alpha) + 0.5\tau(\zeta)\sin(2\alpha)$$

where

$$\sigma(\zeta) = \frac{4\beta_3 F_{rr}}{\pi d^2} + \frac{32M_{ss}}{\pi d^3}\sin\zeta - \frac{32M_{tt}}{\pi d^3}\cos\zeta$$

$$\tau(\zeta) = \frac{16F_{rs}}{3\pi d^2}\sin^2\zeta + \frac{16F_{rt}}{3\pi d^2}\cos^2\zeta \qquad \text{with} \quad \begin{matrix}\beta_3 = 0 \ \ if \ \ F_{rr} \le 0 \\ \beta_3 = 1 \ \ if \ \ F_{rr} > 0\end{matrix}$$

$$\alpha = \frac{1}{2}\tan^{-1}\frac{2\tau(\zeta)}{\sigma(\zeta)}$$

The stresses are calculated for all directions, $0° \le \zeta \le 90°$, in order to find the maximum.

If the failure strain EFAIL is set to zero, the failure strain model is not used. In a similar manner, when the value of a resultant at failure is set to zero, the corresponding term in the failure surface is ignored. For example, if only N_{rr_F} is nonzero, the failure surface is reduced to $|N_{rr}| = N_{rr_F}$. None, either, or both of the failure models may be active depending on the specified input values.

The inertias of the spot welds are scaled during the first time step so that their stable time step size is Δt. A strong compressive load on the spot weld at a later time may reduce the length of the spot weld so that stable time step size drops below Δt. If the value of Δt is zero, mass scaling is not performed, and the spot welds will probably limit the time step size. Under most circumstances, the inertias of the spot welds are small enough that scaling them will have a negligible effect on the structural response and the use of this option is encouraged.

Spot weld force history data is written into the SWFORC ascii file. In this database the resultant moments are not available, but they are in the binary time history database and in the ASCII elout file.

When the DAMAGE-FAILURE option is invoked, the constitutive properties for the damaged material are obtained from the undamaged material properties. The amount of damage evolved is represented by the constant, ω, which varies from zero if no damage has occurred to unity for complete rupture. For uniaxial loading, the nominal stress in the damaged material is given by

$$\sigma_{nominal} = \frac{P}{A}$$

where P is the applied load and A is the surface area. The true stress is given by:

$$\sigma_{true} = \frac{P}{A - A_{loss}}$$

where A_{loss} is the void area. The damage variable can then be defined:

$$\omega = \frac{A_{loss}}{A} \quad 0 \le \omega \le 1$$

In this model damage is defined in terms of plastic strain after the failure strain is exceeded:

$$\omega = \frac{\varepsilon_{eff}^{p} - \varepsilon_{failure}^{p}}{\varepsilon_{rupture}^{p} - \varepsilon_{failure}^{p}} \quad if \quad \varepsilon_{failure}^{p} \le \varepsilon_{eff}^{p} \le \varepsilon_{rupture}^{p}$$

After exceeding the failure strain EFAIL, softening begins and continues until the rupture strain RS is reached.

If BETA is specified, the stress is multiplied by an exponential using ω defined in the previous equation,

$$\sigma_d = \sigma \cdot \exp(-\beta\omega) .$$

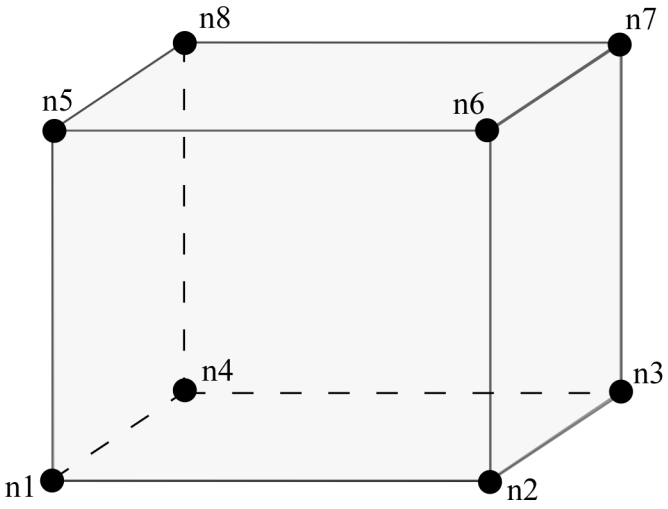


Figure 100.1. A solid element used as spot weld is shown. When resultant based failure is used orientation is very important. Nodes n1-n4 attach to the lower shell mid-surface and nodes n5-n8 attach to the upper shell mid-surface. The resultant forces and moments are computed based on the assumption that the brick element is properly oriented.

*MAT_SPOTWELD_DAIMLERCHRYSLER

This is Material Type 100. The material model applies to solid element type 1 with type 6 hourglass control. Spot weld elements may be placed between any two deformable shell surfaces and tied with constraint contact, *CONTACT_TIED_SURFACE_TO_SURFACE, which eliminates the need to have adjacent nodes at spot weld locations. Spot weld failure is modeled using this card and *DEFINE_CONNECTION_PROPERTIES data. Details of the failure model can be found in Seeger, Feucht, Frank, Haufe, and Keding [2005].

It is advisable to include all spot welds, which provide the slave nodes, and spot welded materials, which define the master segments, within a single *CONTACT_TIED_SURFACE_TO_SURFACE interface. This contact type uses constraint equations. If multiple interfaces are treated independently, significant problems can occur if such interfaces share common nodes. An added benefit is that memory usage can be substantially less with a single interface.

Card 1

1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR			DT	TFAIL
Type	A8	F	F	F			F	F

Card 2

Variable	EFAIL							NF
Type	F							F

Card 3

Variable	RS			TRUE_T	CON_ID			
Type	F			F	F			

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.
DT	Time step size for mass scaling, Δt.
TFAIL	Failure time if nonzero. If zero this option is ignored.
EFAIL	Effective plastic strain in weld material at failure. See remark below.
NF	Number of failure function evaluations stored for filtering by time averaging. The default value is set to zero which is generally recommended unless oscillatory resultant forces are observed in the time history databases. Even though these welds should not oscillate significantly, this option was added for consistency with the other spot weld options. NF affects the storage since it is necessary to store the failure terms. When NF is nonzero, the resultants in the output databases are filtered. NF cannot exceed 30.
RS	Rupture strain. See Remarks below.
TRUE_T	True weld thickness for single hexahedron solid weld elements. See comments below.
CON_ID	Connection ID of *DEFINE_CONNECTION card.

Remarks:

This weld material is modeled with isotropic hardening plasticity. The yield stress and constant hardening modulus are assumed to be those of the welded shell elements as defined in a *DEFINE_CONNECTION_PROPERTIES table. A failure function and damage type is also defined by *DEFINE_ CONNECTION_PROPERTIES data. The interpretation of EFAIL and RS is determined by the choice of damage type. This is discussed in remark 4 on *DEFINE_CONNECTION_PROPERTIES.

Solid weld elements are tied to the mid-plane of shell materials and so typically have a thickness that is half the sum of the thicknesses of the welded shell sections. As a result, a weld under shear loading can be subject to an artificially large moment which will be balanced by normal forces transferred through the tied contact. These normal forces will cause the normal term in the failure calculation to be artificially high. Inputting a TRUE_T that is smaller than the modeled thickness, for example, 10%-30% of true thickness, will scale down the normal force

that results from the balancing moment and provide more realistic failure calculations. TRUE_T effects only the failure calculation, not the weld element behavior. If TRUE_T=0 or data is omitted, the modeled weld element thickness is used.

Solid element force resultants for MAT_SPOTWELD are written to the spot weld force file, SWFORC, and the file for element stresses and resultants for designated elements, ELOUT. Also, spot weld failure data is written to the file, DCFAIL.

***MAT_GEPLASTIC_SRATE_2000a**

This is Material Type 101. The GEPLASTIC_SRATE_2000a material model characterizes General Electric's commercially available engineering thermoplastics subjected to high strain rate events. This material model features the variation of yield stress as a function of strain rate, cavitation effects of rubber modified materials and automatic element deletion of either ductile or brittle materials.

Card 1

1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	RATESF	EDOT0	ALPHA	
Type	A8	F	F	F	F	F	F	

Card 2

Variable	LCSS	LCFEPS	LCFSIG	LCE				
Type	F	F	F	F				

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
E	Young's Modulus.
PR	Poisson's ratio.
RATESF	Constant in plastic strain rate equation.
EDOT0	Reference strain rate
ALPHA	Pressure sensitivity factor
LCSS	Load curve ID or Table ID that defines the post yield material behavior. The values of this stress-strain curve are the difference of the yield stress and strain respectively. This means the first values for both stress and strain should be zero. All subsequent values will define softening or hardening.

VARIABLE	DESCRIPTION
LCFEPS	Load curve ID that defines the plastic failure strain as a function of strain rate.
LCFSIG	Load curve ID that defines the Maximum principal failure stress as a function of strain rate.
LCE	Load curve ID that defines the Unloading moduli as a function of plastic strain.

Remarks:

The constitutive model for this approach is:

$$\dot{\varepsilon}_p = \dot{\varepsilon}_0 \exp(A\{\sigma - S(\varepsilon_p)\}) \times \exp(-p\alpha A)$$

where $\dot{\varepsilon}_0$ and A are rate dependent yield stress parameters, $S(\varepsilon_p)$ internal resistance (strain hardening) and α is a pressure dependence parameter.

In this material the yield stress may vary throughout the finite element model as a function of strain rate and hydrostatic stress. Post yield stress behavior is captured in material softening and hardening values. Finally, ductile or brittle failure measured by plastic strain or maximum principal stress respectively is accounted for by automatic element deletion.

Although this may be applied to a variety of engineering thermoplastics, GE Plastics have constants available for use in a wide range of commercially available grades of their engineering thermoplastics.

***MAT_INV_HYPERBOLIC_SIN**

This is Material Type 102. It allows the modeling of temperature and rate dependent plasticity, Sheppard and Wright [1979].

Card 1

Variable	MID	RO	E	PR	T	HC	VP	
Type	A8	F	F	F	F	F	F	

Card 2

Variable	ALPHA	N	A	Q	G	EPS0		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
E	Young's Modulus.
PR	Poisson's ratio
T	Initial Temperature.
HC	Heat generation coefficient.
VP	Formulation for rate effects: EQ.0.0: Scale yield stress (default) EQ.1.0: Viscoplastic formulation.
ALPHA	See Remarks.
N	See Remarks.
A	See Remarks.
Q	See Remarks.
G	See Remarks.

Remarks:

Resistance to deformation is both temperature and strain rate dependent. The flow stress equation is:

$$\sigma = \frac{1}{\alpha}\sinh^{-1}\left(\left[\frac{Z}{A}\right]^{\frac{1}{N}}\right)$$

where Z, the Zener-Holloman temperature compensated strain rate, is:

$$Z = \dot{\varepsilon}\exp\left(\frac{Q}{GT}\right)$$

The units of the material constitutive constants are as follows: A (1/sec), N (dimensionless), α (1/MPa), the activation energy for flow, Q(J/mol), and the universal gas constant, G (J/mol K). The value of G will only vary with the unit system chosen. Typically it will be either 8.3144 J/mol ∞ K, or 40.8825 lb in/mol ∞ R.

The final equation necessary to complete our description of high strain rate deformation is one that allows us to compute the temperature change during the deformation. In the absence of a couples thermo-mechanical finite element code we assume adiabatic temperature change and follow the empirical assumption that 90-95% of the plastic work is dissipated as heat. Thus the heat generation coefficient is

$$HC \approx \frac{0.9}{\rho C_v}$$

where ρ is the density of the material and C_v is the specific heat.

*MAT_ANISOTROPIC_VISCOPLASTIC

This is Material Type 103. This anisotropic-viscoplastic material model applies to shell and brick elements. The material constants may be fit directly or, if desired, stress versus strain data may be input and a least squares fit will be performed by LS-DYNA to determine the constants. Kinematic or isotopic or a combination of kinematic and isotropic hardening may be used. A detailed description of this model can be found in the following references: Berstad, Langseth, and Hopperstad [1994]; Hopperstad and Remseth [1995]; and Berstad [1996]. Failure is based on effective plastic strain or by a user defined subroutine.

Card 1

1	2	3	4	5	6	7	8
MID	RO	E	PR	SIGY	FLAG	LCSS	ALPHA
A8	F	F	F	F	F	F	F

Card 2

QR1	CR1	QR2	CR2	QX1	CX1	QX2	CX2
F	F	F	F	F	F	F	F

Card 3

VK	VM	R00 or F	R45 or G	R90 or H	L	M	N
F	F	F	F	F	F	F	F

Card 4

AOPT	FAIL	NUMINT	MACF				
F	F	F	I				

Card 5

	1	2	3	4	5	6	7	8
Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card 6

Variable	V1	V2	V3	D1	D2	D3	BETA	
Type	F	F	F	F	F	F	F	

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
E	Young's modulus
PR	Poisson's ratio
SIGY	Initial yield stress
FLAG	Flag EQ.0: Give all material parameters EQ.1: Material parameters are fit in LS-DYNA to Load curve or Table given below. The parameters Q_{r1}, C_{r1}, Q_{r2}, and C_{r2} for isotropic hardening are determined by the fit and those for kinematic hardening are found by scaling those for isotropic hardening by $(1-\alpha)$ where α is defined below in columns 51-60. EQ.2: Use load curve directly, i.e., no fitting is required for the parameters Q_{r1}, C_{r1}, Q_{r2}, and C_{r2}. A table is not allowed.
LCSS	Load curve ID or Table ID. The load curve ID defines effective stress versus effective plastic strain. Card 2 is ignored with this option. The table ID, see Figure 24.1, defines for each strain rate value a load curve ID giving the stress versus effective plastic strain for that rate. If the load curve only is used, then the coefficients V_k and V_m must be given if viscoplastic behavior is desired. If a Table ID is given these coefficients are determined internally during initialization.

VARIABLE	DESCRIPTION
ALPHA	α distribution of hardening used in the curve-fitting. $\alpha = 0$ pure kinematic hardening and $\alpha = 1$ provides pure isotropic hardening
QR1	Isotropic hardening parameter Q_{r1}
CR1	Isotropic hardening parameter C_{r1}
QR2	Isotropic hardening parameter Q_{r2}
CR2	Isotropic hardening parameter C_{r2}
QX1	Kinematic hardening parameter $Q_{\chi 1}$
CX1	Kinematic hardening parameter $C_{\chi 1}$
QX2	Kinematic hardening parameter $Q_{\chi 2}$
CX2	Kinematic hardening parameter $C_{\chi 2}$
VK	Viscous material parameter V_k
VM	Viscous material parameter V_m
R00	R_{00} for shell (Default=1.0)
R45	R_{45} for shell (Default=1.0)
R90	R_{90} for shell (Default=1.0)
F	F for brick (Default =1/2)
G	G for brick (Default =1/2)
H	H for brick (Default =1/2)
L	L for brick (Default =3/2)
M	M for brick (Default =3/2)
N	N for brick (Default =3/2)

VARIABLE	DESCRIPTION
AOPT	Material axes option (see MAT_*OPTION* TROPIC_ELASTIC for a more complete description): EQ.0.0: locally orthotropic with material axes determined by element nodes 1, 2, and 4, as with *DEFINE_COORDINATE_NODES. EQ.1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center; this is the a-direction. This option is for solid elements only. EQ.2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR. EQ.3.0: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle, BETA, from a line in the plane of the element defined by the cross product of the vector **v** with the element normal. EQ.4.0: locally orthotropic in cylindrical coordinate system with the material axes determined by a vector **v**, and an originating point, P, which define the centerline axis. This option is for solid elements only. LT.0.0: the absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_NODES, *DEFINE_COORDINATE_SYSTEM or *DEFINE_COORDINATE_VECTOR). Available in R3 version of 971 and later.
FAIL	Failure flag. LT.0.0: User defined failure subroutine is called to determine failure. This is subroutine named, MATUSR_103, in DYN21.F. EQ.0.0: Failure is not considered. This option is recommended if failure is not of interest since many calculations will be saved. GT.0.0: Plastic strain to failure. When the plastic strain reaches this value, the element is deleted from the calculation.
NUMINT	Number of integration points which must fail before element deletion. If zero, all points must fail. This option applies to shell elements only. For the case of one point shells, NUMINT should be set to a value that is less than the number of through thickness integration points. Nonphysical stretching can sometimes appear in the results if all integration points have failed except for one point away from the midsurface. This is due to the fact that unconstrained nodal rotations will prevent strains from developing at the remaining integration point. In fully integrated shells, similar problems can occur.

VARIABLE	DESCRIPTION
MACF	Material axes change flag for brick elements: EQ.1: No change, default, EQ.2: switch material axes a and b, EQ.3: switch material axes a and c, EQ.4: switch material axes b and c.
XP,YP,ZP	x_p y_p z_p, define coordinates of point **p** for AOPT = 1 and 4.
A1,A2,A3	a_1 a_2 a_3, define components of vector **a** for AOPT = 2.
V1,V2,V3	v_1 v_2 v_3, define components of vector **v** for AOPT = 3 and 4.
D1,D2,D3	d_1 d_2 d_3, define components of vector **d** for AOPT = 2.
BETA	Material angle in degrees for AOPT = 3, may be overridden on the element card, see *ELEMENT_SHELL_BETA or *ELEMENT_SOLID_ORTHO.

Remarks:

The uniaxial stress-strain curve is given on the following form

$$\sigma(\varepsilon_{eff}^p, \dot{\varepsilon}_{eff}^p) = \sigma_0 + Q_{r1}(1-\exp(-C_{r1}\varepsilon_{eff}^p)) + Q_{r2}(1-\exp(-C_{r2}\varepsilon_{eff}^p)) + Q_{\chi 1}(1-\exp(-C_{\chi 1}\varepsilon_{eff}^p)) + Q_{\chi 2}(1-\exp(-C_{\chi 2}\varepsilon_{eff}^p)) + V_k \dot{\varepsilon}_{eff}^{p\ V_m}$$

For bricks the following yield criteria is used

$$F(\sigma_{22}-\sigma_{33})^2 + G(\sigma_{33}-\sigma_{11})^2 + H(\sigma_{11}-\sigma_{22})^2 + 2L\sigma_{23}^2 + 2M\sigma_{31}^2 + 2N\sigma_{12}^2 = \left(\sigma(\varepsilon_{eff}^p, \dot{\varepsilon}_{eff}^p)\right)^2$$

where ε_{eff}^p is the effective plastic strain and $\dot{\varepsilon}_{eff}^p$ is the effective plastic strain rate. For shells the anisotropic behavior is given by R_{00}, R_{45} and R_{90}. The model will work when the three first parameters in card 3 are given values. When $V_k = 0$ the material will behave elasto-plastically. Default values are given by:

$$F = G = H = \frac{1}{2}$$

$$L = M = N = \frac{3}{2}$$

$$R_{00} = R_{45} = R_{90} = 1$$

Strain rate of accounted for using the Cowper and Symonds model which, e.g., model 3, scales the yield stress with the factor:

$$1+\left(\frac{\dot{\varepsilon}_{eff}^{p}}{C}\right)^{1/p}$$

To convert these constants set the viscoelastic constants, V_k and V_m, to the following values:

$$V_k = \sigma\left(\frac{1}{C}\right)^{\frac{1}{p}}$$

$$V_m = \frac{1}{p}$$

This model properly treats rate effects. The viscoplastic rate formulation is an option in other plasticity models in LS-DYNA, e.g., mat_3 and mat_24, invoked by setting the parameter VP to 1.

***MAT_ANISOTROPIC_PLASTIC**

This is Material Type 103. This anisotropic-plastic material model is a simplified version of the MAT_ANISOTROPIC_VISCOPLASTIC above. This material model applies only to shell elements.

Card 1

Variable	MID	RO	E	PR	SIGY	LCSS		
Type	A8	F	F	F	F	F		

Card 2

Variable	QR1	CR1	QR2	CR2				
Type	F	F	F	F				

Card 3

Variable	R00	R45	R90	S11	S22	S33	S12	
Type	F	F	F	F	F	F	F	

Card 4

Variable	AOPT							
Type	F							

Card 5	1	2	3	4	5	6	7	8
Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card 6

Variable	V1	V2	V3	D1	D2	D3	BETA	
Type	F	F	F	F	F	F	F	

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
E	Young's modulus
PR	Poisson's ratio
SIGY	Initial yield stress
LCSS	Load curve ID. The load curve ID defines effective stress versus effective plastic strain. Card 2 is ignored with this option.
QR1	Isotropic hardening parameter Q_{r1}
CR1	Isotropic hardening parameter C_{r1}
QR2	Isotropic hardening parameter Q_{r2}
CR2	Isotropic hardening parameter C_{r2}
R00	R_{00} for anisotropic hardening
R45	R_{45} for anisotropic hardening
R90	R_{90} for anisotropic hardening
S11	Yield stress in local x-direction. This input is ignored if $(R_{00}, R_{45}, R_{90}) > 0$.

VARIABLE	DESCRIPTION
S22	Yield stress in local y-direction. This input is ignored if $(R_{00}, R_{45}, R_{90}) > 0$.
S33	Yield stress in local z-direction. This input is ignored if $(R_{00}, R_{45}, R_{90}) > 0$.
S12	Yield stress in local xy-direction. This input is ignored if $(R_{00}, R_{45}, R_{90}) > 0$.
AOPT	Material axes option (see MAT_*OPTION* TROPIC_ELASTIC for a more complete description): EQ.0.0: locally orthotropic with material axes determined by element nodes 1, 2, and 4, as with *DEFINE_COORDINATE_NODES. EQ.1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center; this is the a-direction. This option is for solid elements only. EQ.2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR. EQ.3.0: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle, BETA, from a line in the plane of the element defined by the cross product of the vector **v** with the element normal. EQ.4.0: locally orthotropic in cylindrical coordinate system with the material axes determined by a vector **v**, and an originating point, P, which define the centerline axis. This option is for solid elements only. LT.0.0: the absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_NODES, *DEFINE_COORDINATE_SYSTEM or *DEFINE_COORDINATE_VECTOR). Available in R3 version of 971 and later.
XP,YP,ZP	x_p y_p z_p, define coordinates of point **p** for AOPT = 1 and 4.
A1,A2,A3	a_1 a_2 a_3, define components of vector **a** for AOPT = 2.
D1,D2,D3	d_1 d_2 d_3, define components of vector **d** for AOPT = 2.
V1,V2,V3	v_1 v_2 v_3, define components of vector **v** for AOPT = 3 and 4.
BETA	Material angle in degrees for AOPT = 3, may be overridden on the element card, see *ELEMENT_SHELL_BETA or *ELEMENT_SOLID_ORTHO.

Remarks:

If no load curve is defined for the effective stress versus effective plastic strain, the uniaxial stress-strain curve is given on the following form

$$\sigma(\varepsilon_{eff}^{p}) = \sigma_0 + Q_{r1}(1 - \exp(-C_{r1}\varepsilon_{eff}^{p})) + \ Q_{r2}(1 - \exp(-C_{r2}\varepsilon_{eff}^{p}))$$

where ε_{eff}^{p} is the effective plastic strain. For shells the anisotropic behavior is given by R_{00}, R_{45} and R_{90}, or the yield stress in the different direction. Default values are given by:

$$R_{00} = R_{45} = R_{90} = 1$$

if the variables R00, R45, R90, S11, S22, S33 and S12 are set to zero.

***MAT_DAMAGE_1**

This is Material Type 104. This is a continuum damage mechanics (CDM) model which includes anisotropy and viscoplasticity. The CDM model applies to shell, thick shell, and brick elements. A more detailed description of this model can be found in the paper by Berstad, Hopperstad, Lademo, and Malo [1999]. This material model can also model anisotropic damage behavior by setting the FLAG to -1 in Card 2.

Card 1

Variable	MID	RO	E	PR	SIGY	LCSS	LCDS	
Type	A8	F	F	F	F			

Card 2

Variable	Q1	C1	Q2	C2	EPSD	S or EPSR	DC	FLAG
Type	F	F	F	F	F	F	F	F

Card 3

Variable	VK	VM	R00 or F	R45 or G	R90 or H	L	M	N
Type	F	F	F	F	F	F	F	F

Card 4

Variable	AOPT			MACF				
Type	F			I				

Card 5	1	2	3	4	5	6	7	8
Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card 6

Variable	V1	V2	V3	D1	D2	D3	BETA	
Type	F	F	F	F	F	F	F	

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
E	Young's modulus
PR	Poisson's ratio
SIGY	Initial yield stress, σ_0.
LCSS	Load curve ID. Load curve ID defining effective stress versus effective plastic strain. For FLAG = -1.
LCDS	Load curve ID defining nonlinear damage curve. For FLAG = -1.
Q1	Isotropic hardening parameter Q_1
C1	Isotropic hardening parameter C_1
Q2	Isotropic hardening parameter Q_2
C2	Isotropic hardening parameter C_2
EPSD	Damage threshold r_d Damage effective plastic strain when material softening begin. (Default=0.0)
S	Damage material constant S. (Default= $\frac{\sigma_0}{200}$). For FLAG≥0.

VARIABLE	DESCRIPTION
EPSR	Plastic strain at which material ruptures (logarithmic).
DC	Critical damage value D_C. When the damage value D reaches this value, the element is deleted from the calculation. (Default=0.5) For FLAG≥0.
FLAG	Flag EQ.-1: Anisotropic damage EQ.0: No calculation of localization due to damage EQ.1: The model flags element where strain localization occur
VK	Viscous material parameter V_k
VM	Viscous material parameter V_m
R00	R_{00} for shell (Default=1.0)
R45	R_{45} for shell (Default=1.0)
R90	R_{90} for shell (Default=1.0)
F	F for brick (Default =1/2)
G	G for brick (Default =1/2)
H	H for brick (Default =1/2)
L	L for brick (Default =3/2)
M	M for brick (Default =3/2)
N	N for brick (Default =3/2)
AOPT	Material axes option (see MAT_*OPTION* TROPIC_ELASTIC for a more complete description): EQ.0.0: locally orthotropic with material axes determined by element nodes 1, 2, and 4, as with *DEFINE_COORDINATE_NODES. EQ.1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center; this is the a-direction. This option is for solid elements only. EQ.2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR.

VARIABLE	DESCRIPTION
	EQ.3.0: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle, BETA, from a line in the plane of the element defined by the cross product of the vector **v** with the element normal. EQ.4.0: locally orthotropic in cylindrical coordinate system with the material axes determined by a vector **v**, and an originating point, P, which define the centerline axis. This option is for solid elements only. LT.0.0: the absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_NODES, *DEFINE_COORDINATE_SYSTEM or *DEFINE_COORDINATE_VECTOR). Available in R3 version of 971 and later.
MACF	Material axes change flag for brick elements: EQ.1: No change, default, EQ.2: switch material axes a and b, EQ.3: switch material axes a and c, EQ.4: switch material axes b and c.
XP,YP,ZP	x_p y_p z_p, define coordinates of point **p** for AOPT = 1 and 4.
A1,A2,A3	a_1 a_2 a_3, define components of vector **a** for AOPT = 2.
D1,D2,D3	d_1 d_2 d_3, define components of vector **d** for AOPT = 2.
V1,V2,V3	v_1 v_2 v_3, define components of vector **v** for AOPT = 3 and 4.
BETA	Material angle in degrees for AOPT = 3, may be overridden on the element card, see *ELEMENT_SHELL_BETA or *ELEMENT_SOLID_ORTHO.

Remarks:

Anisotropic Damage model (FLAG = -1). At each thickness integration points, an anisotropic damage law acts on the plane stress tensor in the directions of the principal total shell strains, ε_1 and ε_2, as follows:

$$\sigma_{11} = (1 - D_1(\varepsilon_1))\sigma_{110}$$
$$\sigma_{22} = (1 - D_2(\varepsilon_2))\sigma_{220}$$
$$\sigma_{12} = (1 - (D_1 + D_2)/2)\sigma_{120}$$

The transverse plate shear stresses in the principal strain directions are assumed to be damaged as follows:

$$\sigma_{13} = (1 - D_1/2)\sigma_{130}$$
$$\sigma_{23} = (1 - D_2/2)\sigma_{230}$$

In the anisotropic damage formulation, $D_1(\varepsilon_1)$ and $D_2(\varepsilon_2)$ are anisotropic damage functions for the loading directions 1 and 2, respectively. Stresses $\sigma_{110}, \sigma_{220}, \sigma_{120}, \sigma_{130}$ and σ_{230} are stresses in the principal shell strain directions as calculated from the undamaged elastic-plastic material behavior. The strains ε_1 and ε_2 are the magnitude of the principal strains calculated upon reaching the damage thresholds. Damage can only develop for tensile stresses, and the damage functions $D_1(\varepsilon_1)$ and $D_2(\varepsilon_2)$ are identical to zero for negative strains ε_1 and ε_2. The principal strain directions are fixed within an integration point as soon as either principal strain exceeds the initial threshold strain in tension. A more detailed description of the damage evolution for this material model is given in the description of Material 81.

The Continuum Damage Mechanics (CDM) model (FLAG≥0) is based on a CDM model proposed by Lemaitre [1992]. The effective stress $\tilde{\sigma}$, which is the stress calculated over the section that effectively resist the forces and reads.

$$\tilde{\sigma} = \frac{\sigma}{1-D}$$

where D is the damage variable. The evolution equation for the damage variable is defined as

$$\dot{D} = \begin{cases} 0 & \text{for} \quad r \le r_D \\ \dfrac{Y}{S(1-D)}\dot{r} & \text{for} \quad r > r_D \quad \text{and} \quad \sigma_1 > 0 \end{cases}$$

where r_D is the damage threshold,

is a positive material constant , S is the so-called strain energy release rate and σ_1 is the maximal principal stress. The strain energy density release rate is

$$Y = \frac{1}{2}\mathbf{e_e} : \mathbf{C} : \mathbf{e_e} = \frac{\sigma_{vm}^2 R_v}{2E(1-D)^2}$$

where σ_{vm} is the equivalent von Mises stress. The triaxiality function R_v is defined as

$$R_v = \frac{2}{3}(1+\nu) + 3(1-2\nu)\left(\frac{\sigma_H}{\sigma_{vm}}\right)^2$$

The uniaxial stress-strain curve is given in the following form

$$\sigma(r, \dot{\varepsilon}_{eff}^p) = \sigma_0 + Q_1(1-\exp(-C_1 r)) + Q_2(1-\exp(-C_2 r)) + V_k \dot{\varepsilon}_{eff}^{p\ V_m}$$

where r is the damage accumulated plastic strain, which can be calculated by

$$\dot{r} = \dot{\varepsilon}_{eff}^p (1-D)$$

For bricks the following yield criteria is used

$$F(\tilde{\sigma}_{22}-\tilde{\sigma}_{33})^2+G(\tilde{\sigma}_{33}-\tilde{\sigma}_{11})^2+H(\tilde{\sigma}_{11}-\tilde{\sigma}_{22})^2 + 2L\tilde{\sigma}_{23}^2+2M\tilde{\sigma}_{31}^2+2N\tilde{\sigma}_{12}^2=\sigma(r,\dot{\varepsilon}_{eff}^p)$$

where r is the damage effective viscoplastic strain and $\dot{\varepsilon}_{eff}^p$ is the effective viscoplastic strain rate. For shells the anisotropic behavior is given by the R-values: R_{00}, R_{45}, and R_{90}. When $V_k = 0$ the material will behave as an elastoplastic material without rate effects. Default values for the anisotropic constants are given by:

$$F = G = H = \frac{1}{2}$$

$$L = M = N = \frac{3}{2}$$

$$R_{00} = R_{45} = R_{90} = 1$$

so that isotropic behavior is obtained.

Strain rate is accounted for using the Cowper and Symonds model which scales the yield stress with the factor:

$$1+\left(\frac{\dot{\varepsilon}}{C}\right)^{1/p}$$

To convert these constants, set the viscoelastic constants, V_k and V_m, to the following values:

$$V_k = \sigma\left(\frac{1}{C}\right)^{\frac{1}{p}}$$

$$V_m = \frac{1}{p}$$

***MAT_DAMAGE_2**

This is Material Type 105. This is an elastic viscoplastic material model combined with continuum damage mechanics (CDM). This material model applies to shell, thick shell, and brick elements. The elastoplastic behavior is described in the description of material model 24. A more detailed description of the CDM model is given in the description of material model 104 above.

Card 1

Variable	MID	RO	E	PR	SIGY	ETAN	FAIL	TDEL
Type	A8	F	F	F	F	F	F	F
Default	none	none	none	none	none	0.0	10.E+20	0

Card 2

Variable	C	P	LCSS	LCSR				
Type	F	F	F	F				
Default	0	0	0	0				

Card 3

Variable	EPSD	S	DC					
Type	F	F	F					
Default	none	none	none					

Card 4	1	2	3	4	5	6	7	8
Variable	EPS1	EPS2	EPS3	EPS4	EPS5	EPS6	EPS7	EPS8
Type	F	F	\F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

Card 5

Variable	ES1	ES2	ES3	ES4	ES5	ES6	ES7	ES8
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.
SIGY	Yield stress.
ETAN	Tangent modulus, ignored if (LCSS.GT.0) is defined.
FAIL	Failure flag. EQ.0.0: Failure due to plastic strain is not considered. GT.0.0: Plastic strain to failure. When the plastic strain reaches this value, the element is deleted from the calculation.
TDEL	Minimum time step size for automatic element deletion.
C	Strain rate parameter, C, see formula below.
P	Strain rate parameter, P, see formula below.

VARIABLE	DESCRIPTION
LCSS	Load curve ID or Table ID. Load curve ID defining effective stress versus effective plastic strain. If defined EPS1-EPS8 and ES1-ES8 are ignored. The table ID defines for each strain rate value a load curve ID giving the stress versus effective plastic strain for that rate, See Figure 24.1. The stress versus effective plastic strain curve for the lowest value of strain rate is used if the strain rate falls below the minimum value. Likewise, the stress versus effective plastic strain curve for the highest value of strain rate is used if the strain rate exceeds the maximum value. The strain rate parameters: C and P; the curve ID, LCSR; EPS1-EPS8 and ES1-ES8 are ignored if a Table ID is defined.
LCSR	Load curve ID defining strain rate scaling effect on yield stress.
EPSD	Damage threshold r_d Damage effective plastic strain when material softening begin. (Default=0.0)
S	Damage material constant S. (Default= $\frac{\sigma_0}{200}$)
DC	Critical damage value D_C. When the damage value D reaches this value, the element is deleted from the calculation. (Default=0.5)
EPS1-EPS8	Effective plastic strain values (optional if SIGY is defined). At least 2 points should be defined.
ES1-ES8	Corresponding yield stress values to EPS1 - EPS8.

Remarks:

The stress-strain behavior may be treated by a bilinear curve by defining the tangent modulus, ETAN. Alternately, a curve similar to that shown in Figure 10.1 is expected to be defined by (EPS1,ES1) - (EPS8,ES8); however, an effective stress versus effective plastic strain curve ID (LCSS) may be input instead if eight points are insufficient. The cost is roughly the same for either approach. The most general approach is to use the table definition with table ID, LCSR, discussed below.

Three options to account for strain rate effects are possible.

I. Strain rate may be accounted for using the Cowper and Symonds model which scales the yield stress with the factor

$$1+\left(\frac{\dot{\varepsilon}}{C}\right)^{1/p}$$

where $\dot{\varepsilon}$ is the strain rate, $\dot{\varepsilon} = \sqrt{\dot{\varepsilon}_{ij}\dot{\varepsilon}_{ij}}$

II. For complete generality a load curve (LCSR) to scale the yield stress may be input instead. In this curve the scale factor versus strain rate is defined.

III. If different stress versus strain curves can be provided for various strain rates, the option using the reference to a table (LCSS) can be used. Then the table input in *DEFINE_TABLE has to be used, see Figure 24.1.

A fully viscoplastic formulation is used in this model.

*MAT_ELASTIC_VISCOPLASTIC_THERMAL

This is Material Type 106. This is an elastic viscoplastic material with thermal effects.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	SIGY	ALPHA	LCSS	
Type	A8	F	F	F	F	F	F	

Card 2

Variable	QR1	CR1	QR2	CR2	QX1	CX1	QX2	CX2
Type	F	F	F	F	F	F	F	F

Card 3

Variable	C	P	LCE	LCPR	LCSIGY	LCR	LCX	LCALPH
Type	F	F	F	F	F	F	F	F

Card 4

Variable	LCC	LCP	TREF					
Type	F	F	F					

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
E	Young's modulus
PR	Poisson's ratio

VARIABLE	DESCRIPTION
SIGY	Initial yield stress
LCSS	Load curve ID or Table ID. The load curve ID defines effective stress versus effective plastic strain. The table ID defines for each temperature value a load curve ID giving the stress versus effective plastic strain for that temperature. The stress versus effective plastic strain curve for the lowest value of temperature is used if the temperature falls below the minimum value. Likewise, the stress versus effective plastic strain curve for the highest value of temperature is used if the temperature exceeds the maximum value. Card 2 is ignored with this option.
ALPHA	Coefficient of thermal expansion.
QR1	Isotropic hardening parameter Q_{r1}
CR1	Isotropic hardening parameter C_{r1}
QR2	Isotropic hardening parameter Q_{r2}
CR2	Isotropic hardening parameter C_{r2}
QX1	Kinematic hardening parameter $Q_{\chi 1}$
CX1	Kinematic hardening parameter $C_{\chi 1}$
QX2	Kinematic hardening parameter $Q_{\chi 2}$
CX2	Kinematic hardening parameter $C_{\chi 2}$
C	Viscous material parameter C
P	Viscous material parameter P
LCE	Load curve defining Young's modulus as a function of temperature. E on card 1 is ignored with this option.
LCPR	Load curve defining Poisson's ratio as a function of temperature. PR on card 1 is ignored with this option.
LCSIGY	Load curve defining the initial yield stress as a function of temperature. SIGY on card 1 is ignored with this option.
LCR	Load curve for scaling the isotropic hardening parameters QR1 and QR2 or the stress given by the load curve LCSS as a function of temperature.

VARIABLE	DESCRIPTION
LCX	Load curve for scaling the kinematic hardening parameters QX1 and QX2 as a function of temperature.
LCALPH	Load curve ID defining the instantaneous coefficient of thermal expansion as a function of temperature: $$d\varepsilon_{ij}^{thermal} = \alpha(T)dT\delta_{ij}$$ ALPHA on card 1 is ignored with this option. If LCALPH is defined as the negative of the load curve ID, the curve is assumed to define the coefficient relative to a reference temperature, TREF below, such that the total thermal strain is give by $$\varepsilon_{ij}^{thermal} = \alpha(T)\left(T - T_{ref}\right)\delta_{ij}$$
LCC	Load curve for scaling the viscous material parameter C as a function of temperature.
LCP	Load curve for scaling the viscous material parameter P as a function of temperature.
TREF	Reference temperature required if and only if LCALPH is given with a negative curve ID.

Remarks:

If LCSS is not given any value the uniaxial stress-strain curve has the form

$$\sigma(\varepsilon_{eff}^{p}) = \sigma_0 + Q_{r1}(1-\exp(-C_{r1}\varepsilon_{eff}^{p})) + Q_{r2}(1-\exp(-C_{r2}\varepsilon_{eff}^{p}))$$
$$+Q_{\chi 1}(1-\exp(-C_{\chi 1}\varepsilon_{eff}^{p})) + Q_{\chi 2}(1-\exp(-C_{\chi 2}\varepsilon_{eff}^{p}))$$

Viscous effects are accounted for using the Cowper and Symonds model, which scales the yield stress with the factor:

$$1+\left(\frac{\dot{\varepsilon}_{eff}^{p}}{C}\right)^{1/p}$$

*MAT_MODIFIED_JOHNSON_COOK

This is Material Type 107.

Define the following two cards with general material parameters

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	BETA	XS1	CP	ALPHA
Type	A8	F	F	F	F	F	F	F

Card 2

Variable	E0DOT	Tr	Tm	T0	FLAG1	FLAG2		
Type	F	F	F	F	F	F		

Define the following two cards if and only if FLAG1=0.
Card 3 is the modified Johnson-Cook constitutive relation (FLAG1=0)

Card 3	1	2	3	4	5	6	7	8
Variable	A	B	N	C	m			
Type	F	F	F	F	F			

Card 4 is the modified Johnson-Cook constitutive relation with Voce hardening (FLAG=0)

Card 4	1	2	3	4	5	6	7	8
Variable	Q1	C1	Q2	C2				
Type	F	F	F	F				

Define the following two cards if and only if FLAG1=1:
Card 3 is the Zerilli-Armstrong constitutive relation (FLAG1=1)

Card 3	1	2	3	4	5	6	7	8
Variable	SIGA	B	BETA0	BETA1				
Type	F	F	F	F				

Card 4 is the Zerilli-Armstrong constitutive relation (FLAG1=1)

Card 4	1	2	3	4	5	6	7	8
Variable	A	N	ALPHA0	ALPHA1				
Type	F	F	F	F				

Define the following card if and only if FLAG2=0:
Card 5 is the modified Johnson-Cook fracture criterion (FLAG2=0)

Card 5	1	2	3	4	5	6	7	8
Variable	DC	PD	D1	D2	D3	D4	D5	
Type	F	F	F	F	F	F	F	

Define the following card if and only if FLAG2=1
Card 5 is the Cockcroft-Latham fracture criterion (FLAG2=1)

Card 5	1	2	3	4	5	6	7	8
Variable	DC	WC						
Type	F	F						

Card 6 includes additional element erosion criteria

Card 6	1	2	3	4	5	6	7	8
Variable	TC	TAUC						
Type	F	F						

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
E	Young's modulus, E.
PR	Poisson's ratio, ν.
BETA	Damage coupling parameter; see Eq. (107.3). EQ.0.0: No coupling between ductile damage and the constitutive relation. EQ.1.0: Full coupling between ductile damage and the constitutive relation.
XS1	Taylor-Quinney coefficient χ, see Eq. (107.20). Gives the portion of plastic work converted into heat (normally taken to be 0.9)
CP	Specific heat C_p, see Eq. (107.20)
ALPHA	Thermal expansion coefficient, α.
E0DOT	User-defined strain rate normalization factor $(\dot{\varepsilon}_0 = \dot{p}_0 = \dot{r}_0)$, see Eq. (107.12)
Tr	Room temperature, see Eq. (107.13)
Tm	Melt temperature, see Eq. (107.13)
T0	Initial temperature
FLAG1	Constitutive relation flag; see Eq. (107.11) and (107.14) EQ.0.0: Modified Johnson-Cook constitutive relation, see Eq. (107.11). EQ.1.0: Zerilli-Armstrong constitutive relation, see Eq. (107.14).

VARIABLE	DESCRIPTION
FLAG2	Fracture criterion flag; see Eq. (107.15) and (107.19). EQ.0.0: Modified Johnson-Cook fracture criterion; see Eq. (107.15). EQ.1.0: Cockcroft-Latham fracture criterion; see Eq. (107.19).
K	Bulk modulus
G	Shear modulus
A	Johnson-Cook yield stress A, see Eq. (107.11).
B	Johnson-Cook hardening parameter B, see Eq. (107.11).
N	Johnson-Cook hardening parameter n, see Eq. (107.11).
C	Johnson-Cook strain rate sensitivity parameter C, see Eq. (107.11).
M	Johnson-Cook thermal softening parameter m, see Eq. (107.11).
Q1	Voce hardening parameter Q_1 (when B = n = 0), see Eq. (107.11).
C1	Voce hardening parameter C_1 (when B = n = 0), see Eq. (107.11).
Q2	Voce hardening parameter Q_2 (when B = n = 0), see Eq. (107.11).
C2	Voce hardening parameter C_2 (when B = n = 0), see Eq. (107.11).
SIGA	Zerilli-Armstrong parameter α_a, see Eq. (107.14).
B	Zerilli-Armstrong parameter B, see Eq. (107.14).
BETA0	Zerilli-Armstrong parameter β_0, see Eq. (107.14).
BETA1	Zerilli-Armstrong parameter β_1, see Eq. (107.14).
A	Zerilli-Armstrong parameter A, see Eq. (107.14).
N	Zerilli-Armstrong parameter n, see Eq. (107.14).
ALPHA0	Zerilli-Armstrong parameter α_0, see Eq. (107.14).
ALPHA1	Zerilli-Armstrong parameter α_1, see Eq. (107.14).

VARIABLE	DESCRIPTION
DC	Critical damage parameter D_c, see Eq. (107.15) and (107.21). When the damage value D reaches this value, the element is eroded from the calculation.
PD	Damage threshold, see Eq. (107.15).
D1-D5	Fracture parameters in the Johnson-Cook fracture criterion, see Eq. (107.16).
WC	Critical Cockcroft-Latham parameter W_c, see Eq. (107.19). When the plastic work per volume reaches this value, the element is eroded from the simulation.
TC	Critical temperature parameter T_c, see Eq. (107.23). When the temperature T, reaches this value, the element is eroded from the simulation.
TAUC	Critical shear stress parameter τ_c. When the maximum shear stress τ reaches this value, the element is eroded from the simulation.

Remarks:

An additive decomposition of the rate-of-deformation tensor $\mathbf{d}$ is assumed, i.e.

$$\mathbf{d} = \mathbf{d}^e + \mathbf{d}^p + \mathbf{d}^t \tag{107.1}$$

Where $\mathbf{d}^e$ is the elastic part, $\mathbf{d}^p$ is the plastic part and $\mathbf{d}^t$ is the thermal part.

The elastic rate-of-deformation $\mathbf{d}^e$ is defined by a linear hypo-elastic relation

$$\tilde{\boldsymbol{\sigma}}^{\nabla J} = \left(K - \frac{2}{3}G \right) tr\left(\mathbf{d}^e\right)\mathbf{I} + 2G\mathbf{d}^e \tag{107.2}$$

Where $\mathbf{I}$ is the unit tensor, K is the bulk modulus and G is the shear modulus. The effective stress tensor is defined by

$$\tilde{\boldsymbol{\sigma}} = \frac{\boldsymbol{\sigma}}{1 - \beta D} \tag{107.3}$$

Where $\boldsymbol{\sigma}$ is the Cauchy-stress and D is the damage variable, while the Jaumann rate of the effective stress reads

$$\tilde{\boldsymbol{\sigma}}^{\nabla J} = \dot{\tilde{\boldsymbol{\sigma}}} - \mathbf{W}\cdot\tilde{\boldsymbol{\sigma}} - \tilde{\boldsymbol{\sigma}}\cdot\mathbf{W}^T \tag{107.4}$$

Where $\mathbf{W}$ is the spin tensor. The parameter β is equal to unity for coupled damage and equal to zero for uncoupled damage.

The thermal rate-of-deformation $\mathbf{d}^T$ is defined by

$$\mathbf{d}^T = \alpha\dot{T}\mathbf{I} \tag{107.5}$$

Where α is the linear thermal expansion coefficient and T is the temperature.

The plastic rate-of-deformation is defined by the associated flow rule as

$$\mathbf{d}^p = \dot{r}\frac{\partial f}{\partial \boldsymbol{\sigma}} = \frac{3}{2}\frac{\dot{r}}{1-\beta D}\frac{\tilde{\boldsymbol{\sigma}}'}{\tilde{\sigma}_{eq}} \tag{107.6}$$

Where $(\bullet)'$ means the deviatoric part of the tensor, r is the damage-equivalent plastic strain, f is the dynamic yield function, i.e.

$$f = \sqrt{\frac{3}{2}\tilde{\boldsymbol{\sigma}}':\tilde{\boldsymbol{\sigma}}'} - \sigma_Y(r,\dot{r},T) \le 0, \quad \dot{r} \ge 0, \quad \dot{r}f = 0 \tag{107.7}$$

And $\tilde{\sigma}_{eq}$ is the damage-equivalent stress.

$$\tilde{\sigma}_{eq} = \sqrt{\frac{3}{2}\tilde{\boldsymbol{\sigma}}':\tilde{\boldsymbol{\sigma}}'} \tag{107.8}$$

The following plastic work conjugate pairs are identified

$$\dot{W}^p = \boldsymbol{\sigma}:\mathbf{d}^p = \tilde{\sigma}_{eq}\dot{r} = \sigma_{eq}\dot{p} \tag{107.9}$$

Where $\dot{W}^p$ is the specific plastic work rate, and the equivalent stress σ_{eq} and the equivalent plastic strain p are defined as

$$\sigma_{eq} = \sqrt{\frac{3}{2}\tilde{\boldsymbol{\sigma}}':\tilde{\boldsymbol{\sigma}}'} = (1-\beta D)\tilde{\sigma}_{eq}, \quad \dot{p} = \sqrt{\frac{2}{3}\mathbf{d}^p:\mathbf{d}^p} = \frac{\dot{r}}{(1-\beta D)} \tag{107.10}$$

The material strength σ_Y is defined by

1. The modified Johnson-Cook constitutive relation

$$\sigma_Y = \left(A + Br^n = \sum_{i=1}^{2} Q_i \left(1 - \exp\left(-C_i r\right)\right)\right)\left(1 + \dot{r}^*\right)^C \left(1 - T^m\right) \qquad (107.11)$$

Where $A, B, C, m, n, Q_1, C_1, Q_2, C_2$ are material parameters; the normalized damage-equivalent plastic strain rate $\dot{r}^*$ is defined by

$$\dot{r}^* = \frac{\dot{r}}{\dot{\varepsilon}_0} \qquad (107.12)$$

In which $\dot{\varepsilon}_0$ is a user-defined reference strain rate; and the homologous temperature reads

$$T^* = \frac{T - T_r}{T_m - T_r} \qquad (107.13)$$

In which T_r is the room temperature (or initial temperature) and T_m is the melting temperature.

2. The Zerilli-Armstrong constitutive relation

$$\sigma_Y = \left[\sigma_a + B\exp\left(-\left(\beta_0 - \beta_1 \ln \dot{r}\right)T\right) + Ar^n \exp\left(-\left(\alpha_0 - \alpha_1 \ln \dot{r}\right)T\right)\right] \qquad (107.14)$$

Where $\sigma_a, B, \beta_0, \beta_1, A, n, \alpha_0, \alpha_1$ are material parameters.

Damage evolution is defined by:

1. The extended Johnson-Cook damage evolution rule:

$$\dot{D} = \begin{cases} 0 & \text{for } p \le p_d \\ \dfrac{D_C}{p_f - p_d}\dot{p} & \text{for } p > p_d \end{cases} \qquad (107.15)$$

Where the current equivalent fracture strain $p_f = p_f\left(\sigma^*, \dot{p}^*, T^*\right)$ is defined as

$$p_f = \left(D_1 + D_2 \exp\left(-D_3\sigma^*\right)\right)\left(1 + \dot{p}^*\right)^{D_4}\left(1 + D_5 T^*\right) \qquad (107.16)$$

And $D_1, D_2, D_3, D_4, D_5, D_C, p_d$ are material parameters; the normalized equivalent plastic strain rate $\dot{p}^*$ is defined by

$$\dot{p}^* = \frac{\dot{p}}{\dot{\varepsilon}_0} \tag{107.17}$$

And the stress triaxiality σ^* reads

$$\sigma^* = \frac{\sigma_H}{\sigma_{eq}}, \quad \sigma_H = \frac{1}{3} tr(\boldsymbol{\sigma}) \tag{107.18}$$

2. The Cockcroft-Latham damage evolution rule:

$$\dot{D} = \frac{D_C}{W_C} \max(\sigma_1, 0)\, \dot{p} \tag{107.19}$$

Where D_C , W_C are material parameters.

Adiabatic heating is calculated as

$$\dot{T} = \chi \frac{\boldsymbol{\sigma} : \mathbf{d}^p}{\rho C_p} = \chi \frac{\tilde{\sigma}_{eq} \dot{r}}{\rho C_p} \tag{107.20}$$

Where χ is the Taylor-Quinney parameter, ρ is the density and C_p is the specific heat. The initial value of the temperature T_0 may be specified by the user.

Element erosion occurs when one of the following several criteria are fulfilled:

1. The damage is greater than the critical value

$$D \geq D_C \tag{107.21}$$

2. The maximum shear stress is greater than a critical value

$$\tau_{\max} = \frac{1}{2} \max\{|\sigma_1 - \sigma_2|, |\sigma_2 - \sigma_3|, |\sigma_3 - \sigma_1|\} \geq \tau_C \tag{107.22}$$

3. The temperature is greater than a critical value

$$T \geq T_C \tag{107.23}$$

History Variable	*Description*
1	Evaluation of damage D
2	Evaluation of stress triaxiality σ^*
3	Evaluation of damaged plastic strain r
4	Evaluation of temperature T
5	Evaluation of damaged plastic strain rate $\dot{r}$
8	Evaluation of plastic work per volume W
9	Evaluation of maximum shear stress τ_{max}

*MAT_ORTHO_ELASTIC_PLASTIC

This is Material Type 108. This model combines orthotropic elastic plastic behavior with an anisotropic yield criterion. This model is implemented only for shell elements.

Card 1

Variable	MID	RO	E11	E22	G12	PR12	PR23	PR31
Type	A8	F	F	F	F	F	F	F

Card 2

Variable	SIGMA0	LC	QR1	CR1	QR2	CR2		
Type	F	I	F	F	F	F		

Card 3

Variable	R11	R22	R33	R12				
Type	F	F	F	F				

Card 4

Variable	AOPY	BETA						
Type	F	F						

Card 5	1	2	3	4	5	6	7	8
Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card 6

Variable	V1	V2	V3	D1	D2	D3		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass Density
E11	Young's Modulus in 11-direction
E22	Young's Modulus in 22-direction
G12	Shear modulus in 12-direction
PR12	Poisson's ratio 12
PR23	Poisson's ratio
PR31	Poisson's ration
LC	Load curve ID. The load curve ID defines effective stress versus effective plastic strain. Values on Card 2 are ignored if this value is defined.
SIGMA0	Initial yield stress, σ_0
QR1	Isotropic hardening parameter, Q_{R1}
CR1	Isotropic hardening parameter, C_{R1}
QR2	Isotropic hardening parameter, Q_{R2}

VARIABLE	DESCRIPTION
CR2	Isotropic hardening parameter, C_{R2}
R11	Yield criteria parameter, R_{11}
R22	Yield criteria parameter, R_{22}
R33	Yield criteria parameter, R_{33}
R12	Yield criteria parameter, R_{12}
AOPT	Material axes option (see Mat_*OPTION* TROPIC_ELASTIC for a more complete description) EQ.0.0: Locally orthotropic with material axes determined by element nodes as shown in Figure 2.1. Nodes 1, 2 and 4 of an element are identical to the node used for the definition of a coordinate system as by *DEFINE_COORDINATE_NODES. EQ.2.0: Globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR. EQ.3.0: Locally orthotropic material axes determined by offsetting the material axes by an angle, OFFANG, from a line determined by taking the cross product of the vector v with the normal to the plane of the element. LT.0.0: the absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_NODES, *DEFINE_COORDINATE_SYSTEM or *DEFINE_COORDINATE_VECTOR). Available in R3 version of 971 and later.
BETA	Material angle in degrees for AOPT=3, may be overwritten on the element card, see *ELEMENT_SHELL_BETA or *ELEMENT_SOLID_ORTHO.
XP YP ZP	Coordinates of point **p** for AOPT=1.
A1 A2 A3	Components of vector **a** for AOPT=2.
V1 V2 V3	Components of vector **v** for AOPT=3.
D1 D2 D3	Components of vector **d** for AOPT=2.

Remarks:

The yield function is defined as

$$f = \bar{f}(\boldsymbol{\sigma}) - \left[\sigma_0 + R\left(\varepsilon^p\right)\right]$$

where the equivalent stress σ_{eq} is defined as an anisotropic yield criterion

$$\sigma_{eq} = \sqrt{F\left(\sigma_{22}-\sigma_{33}\right)^2 + G\left(\sigma_{33}-\sigma_{11}\right)^2 + H\left(\sigma_{11}-\sigma_{22}\right)^2 + 2L\sigma_{23}^2 + 2M\sigma_{31}^2 + 2N\sigma_{12}^2}$$

Where F, G, H, L, M and N are constants obtained by test of the material in different orientations. They are defined as

$$\begin{aligned}
F &= \frac{1}{2}\left(\frac{1}{R_{22}^2} + \frac{1}{R_{33}^2} - \frac{1}{R_{11}^2}\right) \\
G &= \frac{1}{2}\left(\frac{1}{R_{33}^2} + \frac{1}{R_{11}^2} - \frac{1}{R_{22}^2}\right) \\
H &= \frac{1}{2}\left(\frac{1}{R_{11}^2} + \frac{1}{R_{22}^2} - \frac{1}{R_{33}^2}\right) \\
L &= \frac{3}{2R_{23}^2} \\
M &= \frac{3}{2R_{13}^2} \\
N &= \frac{3}{2R_{31}^2}
\end{aligned}$$

The yield stress ratios are defined as follows

$$R_{11} = \frac{\bar{\sigma}_{11}}{\sigma_0}$$

$$R_{22} = \frac{\bar{\sigma}_{22}}{\sigma_0}$$

$$R_{33} = \frac{\bar{\sigma}_{33}}{\sigma_0}$$

$$R_{12} = \frac{\bar{\sigma}_{12}}{\tau_0}$$

$$R_{23} = \frac{\bar{\sigma}_{23}}{\tau_0}$$

$$R_{31} = \frac{\bar{\sigma}_{31}}{\tau_0}$$

where σ_{ij} is the measured yield stress values, σ_0 is the reference yield stress and $\tau_0 = \sigma_0 / \sqrt{3}$.

The strain hardening is either defined by the load curve or the strain hardening *R* is defined by the extended Voce law,

$$R\left(\varepsilon^p\right) = \sum_{i=1}^{2} Q_{Ri}\left(1 - \exp\left(-C_{Ri}\varepsilon^p\right)\right)$$

where ε^p is the effective (or accumulated) plastic strain, and Q_{Ri} and C_{Ri} are strain hardening parameters.

*MAT_JOHNSON_HOLMQUIST_CERAMICS

This is Material Type 110. This Johnson-Holmquist Plasticity Damage Model is useful for modeling ceramics, glass and other brittle materials. A more detailed description can be found in a paper by Johnson and Holmquist [1993].

Card 1

1	2	3	4	5	6	7	8
MID	RO	G	A	B	C	M	N
A8	F	F	F	F	F	F	F

Card 2

1	2	3	4	5	6	7	8
EPSI	T	SFMAX	HEL	PHEL	BETA		
F	F	F	F	F	F		

Card 3

1	2	3	4	5	6	7	8
D1	D2	K1	K2	K3	FS		
F	F	F	F	F	F		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Density
G	Shear modulus
A	Intact normalized strength parameter
B	Fractured normalized strength parameter
C	Strength parameter (for strain rate dependence)
M	Fractured strength parameter (pressure exponent)

VARIABLE	DESCRIPTION
N	Intact strength parameter (pressure exponent).
EPSI	Reference strain rate.
T	Maximum tensile pressure strength.
SFMAX	Maximum normalized fractured strength (if Eq.0, defaults to 1e20).
HEL	Hugoniot elastic limit.
PHEL	Pressure component at the Hugoniot elastic limit.
BETA	Fraction of elastic energy loss converted to hydrostatic energy (affects bulking pressure (history variable 1) that accompanies damage).
D1	Parameter for plastic strain to fracture.
D2	Parameter for plastic strain to fracture (exponent).
K1	First pressure coefficient (equivalent to the bulk modulus).
K2	Second pressure coefficient.
K3	Third pressure coefficient.
FS	Failure criteria. FS < 0 0 fail if p* + t* < 0 (tensile failure). FS = 0 no failure (default). FS > 0 fail if the effective plastic strain > FS.

Remarks:

The equivalent stress for a ceramic-type material is given by

$$\sigma^* = \sigma_i^* - D\left(\sigma_i^* - \sigma_f^*\right)$$

where

$$\sigma_i^* = a\left(p^* + t^*\right)^n \left(1 + c \ln \dot{\varepsilon}\right)$$

represents the intact, undamaged behavior,

$$D = \sum \Delta\varepsilon^p / \varepsilon_f^p$$

represents the accumulated damage (history variable 2) based upon the increase in plastic strain per computational cycle and the plastic strain to fracture

$$\varepsilon_f^p = d_1\left(p^* + t^*\right)^{d_2}$$

and

$$\sigma_f^* = b\left(p^*\right)^m \left(1 + c\ \ln\dot{\varepsilon}\right) \le SFMAX$$

represents the damaged behavior. In each case, the '*' indicates a normalized quantity, the stresses being normalized by the equivalent stress at the Hugoniot elastic limit (see below), the pressures by the pressure at the Hugoniot elastic limit (see below) and the strain rate by the reference strain rate. The parameter d1 controls the rate at which damage accumulates. If it is made 0, full damage occurs in one time step i.e. instantaneously. It is also the best parameter to vary if one attempts to reproduce results generated by another finite element program.

In undamaged material, the hydrostatic pressure is given by

$$P = k_1\mu + k_2\mu^2 + k_3\mu^3$$

in compression and

$$P = k_1\mu$$

in tension where $\mu = \rho/\rho_0 - 1$. When damage starts to occur, there is an increase in pressure. A fraction, between 0 and 1, of the elastic energy loss, β, is converted into hydrostatic potential energy (pressure). The details of this pressure increase are given in the reference.

Given *hel* and *g*, μ_{hel} can be found iteratively from

$$hel = k_1\mu_{hel} + k_2\mu_{hel}^2 + k_3\mu_{hel}^3 + (4/3)\,g(\mu_{hel}/(1+\mu_{hel})$$

and, subsequently, for normalization purposes,

$$P_{hel} = k_1\mu_{hel} + k_2\mu_{hel}^2 + k_3\mu_{hel}^{3\ 3}$$

and

$$\sigma_{hel} = 1.5\left(hel - p_{hel}\right)$$

These are calculated automatically by LS-DYNA if ρ_{f0} is zero on input.

*MAT_JOHNSON_HOLMQUIST_CONCRETE

This is Material Type 111. This model can be used for concrete subjected to large strains, high strain rates and high pressures. The equivalent strength is expressed as a function of the pressure, strain rate, and damage. The pressure is expressed as a function of the volumetric strain and includes the effect of permanent crushing. The damage is accumulated as a function of the plastic volumetric strain, equivalent plastic strain and pressure. A more detailed description of this model can be found in the paper by Holmquist, Johnson, and Cook [1993].

Card 1

1	2	3	4	5	6	7	8
MID	RO	G	A	B	C	N	FC
A8	F	F	F	F	F	F	F

Card 2

1	2	3	4	5	6	7	8
T	EPS0	EFMIN	SFMAX	PC	UC	PL	UL
F	F	F	F	F	F	F	F

Card 3

1	2	3	4	5	6	7	8
D1	D2	K1	K2	K3	FS		
F	F	F	F	F	F		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
G	Shear modulus.
A	Normalized cohesive strength.
B	Normalized pressure hardening.

VARIABLE	DESCRIPTION
C	Strain rate coefficient.
N	Pressure hardening exponent.
FC	Quasi-static uniaxial compressive strength.
T	Maximum tensile hydrostatic pressure.
EPS0	Reference strain rate.
EFMIN	Amount of plastic strain before fracture.
SFMAX	Normalized maximum strength.
PC	Crushing pressure.
UC	Crushing volumetric strain.
PL	Locking pressure.
UL	Locking volumetric strain.
D1	Damage constant.
D2	Damage constant.
K1	Pressure constant.
K2	Pressure constant.
K3	Pressure constant.
FS	Failure type: FS < 0 fail if damage strength < *0* FS = 0 fail if $P^* + T^* \le 0$ (tensile failure). FS > 0 fail if the effective plastic strain > FS.

Remarks:

The normalized equivalent stress is defined as

$$\sigma^* = \frac{\sigma}{f_c'}$$

where σ is the actual equivalent stress, and f' is the quasi-static uniaxial compressive strength. The expression is defined as:

$$\sigma^* = \left[A(1-D) + BP^{*N} \right]\left[1 + C \ln\left(\dot{\varepsilon}^*\right)\right]$$

where D is the damage parameter, $P^* = P/f_c'$ is the normalized pressure and $\dot{\varepsilon}^* = \dot{\varepsilon}/\dot{\varepsilon}_0$ is the dimensionless strain rate. The model incrementally accumulates damage, *D*, both from equivalent plastic strain and plastic volumetric strain, and is expressed as

$$D = \sum \frac{\Delta\varepsilon_p + \Delta\mu_p}{D_1\left(P^* + T^*\right)^{D_2}}$$

where $\Delta\varepsilon_p$ and $\Delta\mu_p$ are the equivalent plastic strain and plastic volumetric strain, D_1 and D_2 are material constants and $T^* = T/f_c'$ is the normalized maximum tensile hydrostatic pressure.

The damage strength, *DS*, is defined in compression when P^* > 0 as

$$DS = f_c^{'} \cdot MIN\left[SFMAX, A(1-D) + BP^{*N} \right]\left[1 + C * \ln\left(\dot{\varepsilon}^*\right)\right]$$

Or in tension if P^* < 0, as

$$DS = f_c^{'} \cdot MAX\left[0, A(1-D) - A\left(\frac{P^*}{T}\right)\right]\left[1 + C * \ln\left(\dot{\varepsilon}^*\right)\right]$$

The pressure for fully dense material is expressed as

$$P = K_1\bar{\mu} + K_2\bar{\mu}^2 + K_3\bar{\mu}^3$$

where K_1 , K_2 and K_3 are material constants and the modified volumetric strain is defined as

$$\bar{\mu} = \frac{\mu - \mu_{lock}}{1 + \mu_{lock}}$$

where μ_{lock} is the locking volumetric strain.

*MAT_FINITE_ELASTIC_STRAIN_PLASTICITY

This is Material Type 112. An elasto-plastic material with an arbitrary stress versus strain curve and arbitrary strain rate dependency can be defined. The elastic response of this model uses a finite strain formulation so that large elastic strains can develop before yielding occurs. This model is available for solid elements only. See Remarks below.

Card 1

Variable	MID	RO	E	PR	SIGY	ETAN		
Type	A8	F	F	F	F	F		
Default	none	none	none	none	none	0.0		

Card 2

Variable	C	P	LCSS	LCSR				
Type	F	F	F	F				
Default	0	0	0	0				

Card 3

Variable	EPS1	EPS2	EPS3	EPS4	EPS5	EPS6	EPS7	EPS8
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

Card 4	1	2	3	4	5	6	7	8
Variable	ES1	ES2	ES3	ES4	ES5	ES6	ES7	ES8
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.
SIGY	Yield stress.
ETAN	Tangent modulus, ignored if (LCSS.GT.0) is defined.
C	Strain rate parameter, C, see formula below.
P	Strain rate parameter, P, see formula below.
LCSS	Load curve ID or Table ID. Load curve ID defining effective stress versus effective plastic strain. If defined EPS1-EPS8 and ES1-ES8 are ignored. The table ID defines for each strain rate value a load curve ID giving the stress versus effective plastic strain for that rate, See Figure 24.1. The stress versus effective plastic strain curve for the lowest value of strain rate is used if the strain rate falls below the minimum value. Likewise, the stress versus effective plastic strain curve for the highest value of strain rate is used if the strain rate exceeds the maximum value. The strain rate parameters: C and P; the curve ID, LCSR; EPS1-EPS8 and ES1-ES8 are ignored if a Table ID is defined.
LCSR	Load curve ID defining strain rate scaling effect on yield stress.
EPS1-EPS8	Effective plastic strain values (optional if SIGY is defined). At least 2 points should be defined. The first point must be zero corresponding to the initial yield stress. WARNING: If the first point is nonzero the yield stress is extrapolated to determine the initial yield. If this option is used SIGY and ETAN are ignored and may be input as zero.
ES1-ES8	Corresponding yield stress values to EPS1 - EPS8.

Remarks:

The stress strain behavior may be treated by a bilinear stress strain curve by defining the tangent modulus, ETAN. Alternately, a curve similar to that shown in Figure 10.1 is expected to be defined by (EPS1,ES1) - (EPS8,ES8); however, an effective stress versus effective plastic strain curve (LCSS) may be input instead if eight points are insufficient. The cost is roughly the same for either approach. The most general approach is to use the table definition (LCSS) discussed below.

Three options to account for strain rate effects are possible.

I. Strain rate may be accounted for using the Cowper and Symonds model which scales the yield stress with the factor

$$1+\left(\frac{\dot{\varepsilon}}{C}\right)^{1/p}$$

where $\dot{\varepsilon}$ is the strain rate, $\dot{\varepsilon}=\sqrt{\dot{\varepsilon}_{ij}\dot{\varepsilon}_{ij}}$.

II. For complete generality a load curve (LCSR) to scale the yield stress may be input instead. In this curve the scale factor versus strain rate is defined.

III. If different stress versus strain curves can be provided for various strain rates, the option using the reference to a table (LCSS) can be used. Then the table input in *DEFINE_TABLE has to be used, see Figure 24.1.

***MAT_TRIP**

This is Material Type 113. This isotropic elasto-plastic material model applies to shell elements only. It features a special hardening law aimed at modelling the temperature dependent hardening behavior of austenitic stainless TRIP-steels. TRIP stands for Transformation Induced Plasticity. A detailed description of this material model can be found in Hänsel, Hora, and Reissner [1998] and Schedin, Prentzas, and Hilding [2004].

Card Format (I10, 7E10.0)

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	CP	T0	TREF	TA0
Type	A8	F	F					
Default								

Card Format (8E10.0)

Card 2	1	2	3	4	5	6	7	8
Variable	A	B	C	D	P	Q	E0MART	VM0
Type	F	F						
Default								

Card Format (8E10.0)

Card 3	1	2	3	4	5	6	7	8
Variable	AHS	BHS	M	N	EPS0	HMART	K1	K2
Type								
Default								

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.
CP	Adiabatic temperature calculation option: EQ.0.0: Adiabatic temperature calculation is disabled. GT.0.0: CP is the specific heat C_p. Adiabatic temperature calculation is enabled.
T0	Initial temperature T_0 of the material if adiabatic temperature calculation is enabled.
TREF	Reference temperature for output of the yield stress as history variable 1.
TA0	Reference temperature T_{A0}, the absolute zero for the used temperature scale, e.g. -273.15 if the Celsius scale is used and 0.0 if the Kelvin scale is used.
A	Martensite rate equation parameter *A*, see equations below.
B	Martensite rate equation parameter *B*, see equations below.
C	Martensite rate equation parameter *C*, see equations below.
D	Martensite rate equation parameter *D*, see equations below.
P	Martensite rate equation parameter *p*, see equations below.
Q	Martensite rate equation parameter *Q*, see equations below.
E0MART	Martensite rate equation parameter $E_{0(mart)}$, see equations below.
VM0	The initial volume fraction of martensite $0.0<V_{m0}<1.0$ may be initialised using two different methods: GT.0.0: V_{m0} is set to VM0. LT.0.0: Can be used only when there are initial plastic strains ε^p present, e.g. when using *INITIAL_STRESS_SHELL. The absolute value of VM0 is then the load curve ID for a function *f* that sets $V_{m0} = f\left(\varepsilon^p\right)$. The function f must be a monotonically nondecreasing function of ε^p.

VARIABLE	DESCRIPTION
AHS	Hardening law parameter A_{HS}, see equations below.
BHS	Hardening law parameter B_{HS}, see equations below.
M	Hardening law parameter m, see equations below.
N	Hardening law parameter n, see equations below.
EPS0	Hardening law parameter ε_0, see equations below.
HMART	Hardening law parameter $\Delta H_{\gamma\rightarrow\alpha'}$, see equations below.
K1	Hardening law parameter K_1, see equations below.
K2	Hardening law parameter K_2, see equations below.

Remarks:

Here a short description is given of the TRIP-material model. The material model uses the von Mises yield surface in combination with isotropic hardening. The hardening is temperature dependent and therefore this material model must be run either in a coupled thermo-mechanical solution, using prescribed temperatures or using the adiabatic temperature calculation option. Setting the parameter CP to the specific heat C_p of the material activates the adiabatic temperature calculation that calculates the temperature rate from the equation

$$\dot{T} = \frac{\sigma \cdot D^p}{\rho C_p},$$

where $\sigma \cdot D^p$ is the plastically dissipated heat. Using the Kelvin scale is recommended, even though other scales may be used without problems.

The hardening behavior is described by the following equations. The Martensite rate equation is

$$\frac{\partial V_m}{\partial \bar{\varepsilon}^p} = \begin{cases} 0 \text{ , if } \varepsilon < E_{0(mart)} \\ \\ \dfrac{B}{A}\exp\left(\dfrac{Q}{T - T_{A0}}\right)\left(\dfrac{1-V_m}{V_m}\right)^{(B+1)/B} {V_m}^p \dfrac{1}{2}(1-\tanh(C + D \cdot T)), \text{ if } \bar{\varepsilon}^p \geq E_{0(mart)} \end{cases},$$

where

$\bar{\varepsilon}^p$ = effective plastic strain and

T = temperature.

The martensite fraction is integrated from the above rate equation:

$$V_m = \int_0^{\varepsilon} \frac{\partial V_m}{\partial \bar{\varepsilon}^p} d\bar{\varepsilon}^p .$$

It always holds that $0.0<V_m<1.0$. The initial martensite content is V_{m0} and must be greater than zero and less than 1.0. Note that V_{m0} is not used during a restart or when initializing the V_m history variable using *INITIAL_STRESS_SHELL.

The yield stress σ_y is

$$\sigma_y = \left\{ B_{HS} - (B_{HS} - A_{HS}) \exp\left(-m \left[\bar{\varepsilon}^p + \varepsilon_0 \right]^n \right) \right\} (K_1 + K_2 T) + \Delta H_{\gamma \to \alpha'} V_m .$$

The parameters p and B should fulfill the following condition

$$(1+B)/B<p,$$

if not fulfilled then the martensite rate will approach infinity as V_m approaches zero. Setting the parameter ε_0 larger than zero, typical range 0.001-0.02 is recommended. A part from the effective true strain a few additional history variables are output, see below.

History variables that are output for post-processing:

Variable	*Description*
1	Yield stress of material at temperature TREF. Useful to evaluate the strength of the material after e.g., a simulated forming operation.
2	Volume fraction martensite, V_m
3	CP EQ.0.0: Not used CP GT.0.0: Temperature from adiabatic temperature calculation

*MAT_LAYERED_LINEAR_PLASTICITY

This is Material Type 114. A layered elastoplastic material with an arbitrary stress versus strain curve and an arbitrary strain rate dependency can be defined. This material must be used with the user defined integration rules, see *INTEGRATION-SHELL, for modeling laminated composite and sandwich shells where each layer can be represented by elastoplastic behavior with constitutive constants that vary from layer to layer. Lamination theory is applied to correct for the assumption of a uniform constant shear strain through the thickness of the shell. Unless this correction is applied, the stiffness of the shell can be grossly incorrect leading to poor results. Generally, without the correction the results are too stiff. This model is available for shell elements only. Also, see Remarks below.

Card 1

Variable	MID	RO	E	PR	SIGY	ETAN	FAIL	TDEL
Type	A8	F	F	F	F	F	F	F
Default	none	none	none	none	none	0.0	10.E+20	0

Card 2

Variable	C	P	LCSS	LCSR				
Type	F	F	F	F				
Default	0	0	0	0				

Card 3

Variable	EPS1	EPS2	EPS3	EPS4	EPS5	EPS6	EPS7	EPS8
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

Card 4	1	2	3	4	5	6	7	8
Variable	ES1	ES2	ES3	ES4	ES5	ES6	ES7	ES8
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.
SIGY	Yield stress.
ETAN	Tangent modulus, ignored if (LCSS.GT.0) is defined.
FAIL	Failure flag. LT.0.0: User defined failure subroutine is called to determine failure EQ.0.0: Failure is not considered. This option is recommended if failure is not of interest since many calculations will be saved. GT.0.0: Plastic strain to failure. When the plastic strain reaches this value, the element is deleted from the calculation.
TDEL	Minimum time step size for automatic element deletion.
C	Strain rate parameter, C, see formula below.
P	Strain rate parameter, P, see formula below.
LCSS	Load curve ID or Table ID. Load curve ID defining effective stress versus effective plastic strain. If defined EPS1-EPS8 and ES1-ES8 are ignored. The table ID defines for each strain rate value a load curve ID giving the stress versus effective plastic strain for that rate, See Figure 24.1. The stress versus effective plastic strain curve for the lowest value of strain rate is used if the strain rate falls below the minimum value. Likewise, the stress versus effective plastic strain curve for the

VARIABLE	DESCRIPTION
	highest value of strain rate is used if the strain rate exceeds the maximum value. The strain rate parameters: C and P; the curve ID, LCSR; EPS1-EPS8 and ES1-ES8 are ignored if a Table ID is defined.
LCSR	Load curve ID defining strain rate scaling effect on yield stress.
EPS1-EPS8	Effective plastic strain values (optional if SIGY is defined). At least 2 points should be defined. The first point must be zero corresponding to the initial yield stress. WARNING: If the first point is nonzero the yield stress is extrapolated to determine the initial yield. If this option is used SIGY and ETAN are ignored and may be input as zero.
ES1-ES8	Corresponding yield stress values to EPS1 - EPS8.

Remarks:

The stress strain behavior may be treated by a bilinear stress strain curve by defining the tangent modulus, ETAN. Alternately, a curve similar to that shown in Figure 10.1 is expected to be defined by (EPS1,ES1) - (EPS8,ES8); however, an effective stress versus effective plastic strain curve (LCSS) may be input instead if eight points are insufficient. The cost is roughly the same for either approach. The most general approach is to use the table definition (LCSS) discussed below.

Three options to account for strain rate effects are possible.

I. Strain rate may be accounted for using the Cowper and Symonds model which scales the yield stress with the factor

$$1+\left(\frac{\dot{\varepsilon}}{C}\right)^{1/p}$$

where $\dot{\varepsilon}$ is the strain rate, $\dot{\varepsilon}=\sqrt{\dot{\varepsilon}_{ij}\dot{\varepsilon}_{ij}}$.

II. For complete generality a load curve (LCSR) to scale the yield stress may be input instead. In this curve the scale factor versus strain rate is defined.

III. If different stress versus strain curves can be provided for various strain rates, the option using the reference to a table (LCSS) can be used. Then the table input in *DEFINE_TABLE has to be used, see Figure 24.1.

*MAT_UNIFIED_CREEP

This is Material Type 115. This is an elastic creep model for modeling creep behavior when plastic behavior is not considered.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	A	N	M	
Type	A8	F	F	F	F	F	F	
Default	none	none	none	none	none	none	none	

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.
A	Stress coefficient.
N	Stress exponent.
M	Time exponent.

Remarks:

The effective creep strain, $\bar{\varepsilon}^c$, given as:

$$\bar{\varepsilon}^c = A\bar{\sigma}^n \bar{t}^m$$

where *A*, *n*, and *m* are constants and $\bar{t}$ is the effective time. The effective stress, $\bar{\sigma}$, is defined as:

$$\bar{\sigma} = \sqrt{\frac{3}{2}\sigma_{ij}\sigma_{ij}}$$

The creep strain, therefore, is only a function of the deviatoric stresses. The volumetric behavior for this material is assumed to be elastic. By varying the time constant m primary creep (*m<1*), secondary creep (*m=1*), and tertiary creep (*m>1*) can be modeled. This model is described by Whirley and Henshall [1992].

***MAT_COMPOSITE_LAYUP**

This is Material Type 116. This material is for modeling the elastic responses of composite layups that have an arbitrary number of layers through the shell thickness. A pre-integration is used to compute the extensional, bending, and coupling stiffness for use with the Belytschko-Tsay resultant shell formulation. The angles of the local material axes are specified from layer to layer in the *SECTION_SHELL input. This material model must be used with the user defined integration rule for shells, see *INTEGRATION_SHELL, which allows the elastic constants to change from integration point to integration point. Since the stresses are not computed in the resultant formulation, the stresses output to the binary databases for the resultant elements are zero. Note that this shell *does not use laminated shell theory* and that storage is allocated for just one integration point (as reported in D3HSP) regardless of the layers defined in the integration rule.

Card 1

Variable	MID	RO	EA	EB	EC	PRBA	PRCA	PRCB
Type	A8	F	F	F	F	F	F	F

Card 2

Variable	GAB	GBC	GCA	AOPT				
Type	F	F	F	F				

Card 3

Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card 4

Variable	V1	V2	V3	D1	D2	D3	BETA	
Type	F	F	F	F	F	F	F	

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
EA	E_a, Young's modulus in a-direction.
EB	E_b, Young's modulus in b-direction.
EC	E_c, Young's modulus in c-direction.
PRBA	ν_{ba}, Poisson's ratio ba.
PRCA	ν_{ca}, Poisson's ratio ca.
PRCB	ν_{cb}, Poisson's ratio cb.
GAB	G_{ab}, shear modulus ab.
GBC	G_{bc}, shear modulus bc.
GCA	G_{ca}, shear modulus ca.
AOPT	Material axes option, see Figure 2.1: EQ.0.0: locally orthotropic with material axes determined by element nodes as shown in Figure 2.1. Nodes 1, 2, and 4 of an element are identical to the nodes used for the definition of a coordinate system as by *DEFINE_COORDINATE_NODES. EQ.1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center; this is the a-direction. This option is for solid elements only. EQ.2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_ VECTOR. EQ.3.0: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle, BETA, from a line in the plane of the element defined by the cross product of the vector **v** with the element normal. The plane of a solid element is the midsurface between the inner surface and outer surface defined by the first four nodes and the last four nodes of the connectivity of the element, respectively. EQ.4.0: locally orthotropic in cylindrical coordinate system with the material axes determined by a vector **v**, and an originating point, P, which define the centerline axis. This option is for solid elements only.

VARIABLE	DESCRIPTION
	LT.0.0: the absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_NODES, *DEFINE_COORDINATE_SYSTEM or *DEFINE_COORDINATE_VECTOR). Available in R3 version of 971 and later.
XP YP ZP	Define coordinates of point **p** for AOPT = 1 and 4.
A1 A2 A3	Define components of vector **a** for AOPT = 2.
V1 V2 V3	Define components of vector **v** for AOPT = 3 and 4.
D1 D2 D3	Define components of vector **d** for AOPT = 2.
BETA	Material angle in degrees for AOPT = 3, may be overridden on the element card, see *ELEMENT_SHELL_BETA.

Remarks:

This material law is based on standard composite lay-up theory. The implementation, [Jones 1975], allows the calculation of the force, *N*, and moment, *M*, stress resultants from:

$$\begin{Bmatrix} N_x \\ N_y \\ N_{xy} \end{Bmatrix} = \begin{bmatrix} A_{11}A_{12}A_{16} \\ A_{21}A_{22}A_{26} \\ A_{16}A_{26}A_{66} \end{bmatrix} \begin{Bmatrix} \varepsilon_x^0 \\ \varepsilon_y^0 \\ \varepsilon_z^0 \end{Bmatrix} + \begin{bmatrix} B_{11}B_{12}B_{16} \\ B_{21}B_{22}B_{26} \\ B_{16}B_{26}B_{66} \end{bmatrix} \begin{Bmatrix} \kappa_x \\ \kappa_y \\ \kappa_{xy} \end{Bmatrix}$$

$$\begin{Bmatrix} M_x \\ M_y \\ M_{xy} \end{Bmatrix} = \begin{bmatrix} B_{11}\ B_{12}\ B_{16} \\ B_{21}\ B_{22}\ B_{26} \\ B_{16}\ B_{26}\ B_{66} \end{bmatrix} \begin{Bmatrix} \varepsilon_x^0 \\ \varepsilon_y^0 \\ \varepsilon_z^0 \end{Bmatrix} + \begin{bmatrix} D_{11}\ D_{12}\ D_{16} \\ D_{21}\ D_{22}\ D_{26} \\ D_{16}\ D_{26}\ D_{66} \end{bmatrix} \begin{Bmatrix} \kappa_x \\ \kappa_y \\ \kappa_{xy} \end{Bmatrix}$$

where A_{ij} is the extensional stiffness, D_{ij} is the bending stiffness, and B_{ij} is the coupling stiffness which is a null matrix for symmetric lay-ups. The mid-surface stains and curvatures are denoted by ε_{ij}^0 and κ_{ij} respectively. Since these stiffness matrices are symmetric, 18 terms are needed per shell element in addition to the shell resultants which are integrated in time. This is considerably less storage than would typically be required with through thickness integration which requires a minimum of eight history variables per integration point, e.g., if 100 layers are used 800 history variables would be stored. Not only is memory much less for this model, but the CPU time required is also considerably reduced.

*MAT_COMPOSITE_MATRIX

This is Material Type 117. This material is used for modeling the elastic responses of composites where a pre-integration is used to compute the extensional, bending, and coupling stiffness coefficients for use with the Belytschko-Tsay resultant shell formulation. Since the stresses are not computed in the resultant formulation, the stresses output to the binary databases for the resultant elements are zero.

Card 1

Variable	MID	RO						
Type	A8	F						

Card 2

Variable	C11	C12	C22	C13	C23	C33	C14	C24
Type	F	F	F	F	F	F	F	F

Card 3

Variable	C34	C44	C15	C25	C35	C45	C55	C16
Type	F	F	F	F	F	F	F	F

Card 4

Variable	C26	C36	C46	C56	C66	AOPT		
Type	F	F	F	F	F	F		

Card 5	1	2	3	4	5	6	7	8
Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card 6

Variable	V1	V2	V3	D1	D2	D3	BETA	
Type	F	F	F	F	F	F	F	

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
CIJ	C_{ij} coefficients of stiffness matrix.
AOPT	Material axes option, see Figure 2.1: EQ. 0.0: locally orthotropic with material axes determined by element nodes as shown in Figure 2.1. Nodes 1, 2, and 4 of an element are identical to the nodes used for the definition of a coordinate system as by *DEFINE_COORDINATE_NODES. EQ. 1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center; this is the a-direction. This option is for solid elements only. EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_ VECTOR. EQ. 3.0: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle, BETA, from a line in the plane of the element defined by the cross product of the vector **v** with the element normal. The plane of a solid element is the midsurface between the inner surface and outer surface defined by the first four nodes and the last four nodes of the connectivity of the element, respectively

VARIABLE	DESCRIPTION
	EQ. 4.0: locally orthotropic in cylindrical coordinate system with the material axes determined by a vector **v**, and an originating point, P, which define the centerline axis. This option is for solid elements only. LT.0.0: the absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_NODES, *DEFINE_COORDINATE_SYSTEM or *DEFINE_COORDINATE_VECTOR). Available in R3 version of 971 and later.
XP YP ZP	Define coordinates of point **p** for AOPT = 1 and 4.
A1 A2 A3	Define components of vector **a** for AOPT = 2.
V1 V2 V3	Define components of vector **v** for AOPT = 3 and 4.
D1 D2 D3	Define components of vector **d** for AOPT = 2.
BETA	Material angle in degrees for AOPT = 3, may be overridden on the element card, see *ELEMENT_SHELL_BETA.

Remarks:

The calculation of the force, N_{ij}, and moment, M_{ij}, stress resultants is given in terms of the membrane strains, ε_i^0, and shell curvatures, κ_i, as:

$$\begin{Bmatrix} N_x \\ N_y \\ N_{xy} \\ M_x \\ M_y \\ M_{xy} \end{Bmatrix} = \begin{bmatrix} C_{11} & C_{12} & C_{13} & C_{14} & C_{15} & C_{16} \\ C_{21} & C_{22} & C_{23} & C_{24} & C_{25} & C_{26} \\ C_{31} & C_{32} & C_{33} & C_{34} & C_{35} & C_{36} \\ C_{41} & C_{42} & C_{43} & C_{44} & C_{45} & C_{46} \\ C_{51} & C_{52} & C_{53} & C_{54} & C_{55} & C_{56} \\ C_{61} & C_{62} & C_{63} & C_{64} & C_{65} & C_{66} \end{bmatrix} \begin{Bmatrix} \varepsilon_x^0 \\ \varepsilon_y^0 \\ \varepsilon_z^0 \\ \kappa_x \\ \kappa_y \\ \kappa_{xy} \end{Bmatrix}$$

where $C_{ij} = C_{ji}$. In this model this symmetric matrix is transformed into the element local system and the coefficients are stored as element history variables. In model type *MAT_COMPOSITE_DIRECT below, the resultants are already assumed to be given in the element local system which reduces the storage since the 21 coefficients are not stored as history variables as part of the element data.

The shell thickness is built into the coefficient matrix and, consequently, within the part ID, which references this material ID, the thickness must be uniform.

*MAT_COMPOSITE_DIRECT

This is Material Type 118. This material is used for modeling the elastic responses of composites where a pre-integration is used to compute the extensional, bending, and coupling stiffness coefficients for use with the Belytschko-Tsay resultant shell formulation. Since the stresses are not computed in the resultant formulation, the stresses output to the binary databases for the resultant elements are zero.

Card 1

Variable	MID	RO						
Type	A8	F						

Card 2

Variable	C11	C12	C22	C13	C23	C33	C14	C24
Type	F	F	F	F	F	F	F	F

Card 3

Variable	C34	C44	C15	C25	C35	C45	C55	C16
Type	F	F	F	F	F	F	F	F

Card 4

Variable	C26	C36	C46	C56	C66			
Type	F	F	F	F	F			

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
CIJ	C_{ij} coefficients of the stiffness matrix.

Remarks:

The calculation of the force, N_{ij}, and moment, M_{ij}, stress resultants is given in terms of the membrane strains, ε_i^0, and shell curvatures, κ_i, as:

$$\begin{Bmatrix} N_x \\ N_y \\ N_{xy} \\ M_x \\ M_y \\ M_{xy} \end{Bmatrix} = \begin{bmatrix} C_{11} & C_{12} & C_{13} & C_{14} & C_{15} & C_{16} \\ C_{21} & C_{22} & C_{23} & C_{24} & C_{25} & C_{26} \\ C_{31} & C_{32} & C_{33} & C_{34} & C_{35} & C_{36} \\ C_{41} & C_{42} & C_{43} & C_{44} & C_{45} & C_{46} \\ C_{51} & C_{52} & C_{53} & C_{54} & C_{55} & C_{56} \\ C_{61} & C_{62} & C_{63} & C_{64} & C_{65} & C_{66} \end{bmatrix} \begin{Bmatrix} \varepsilon_x^0 \\ \varepsilon_y^0 \\ \varepsilon_z^0 \\ \kappa_x \\ \kappa_y \\ \kappa_{xy} \end{Bmatrix}$$

where $C_{ij} = C_{ji}$. In this model the stiffness coefficients are already assumed to be given in the element local system which reduces the storage. Great care in the element orientation and choice of the local element system, see *CONTROL_ACCURACY, must be observed if this model is used.

The shell thickness is built into the coefficient matrix and, consequently, within the part ID, which references this material ID, the thickness must be uniform.

*MAT_GENERAL_NONLINEAR_6DOF_DISCRETE_BEAM

This is Material Type 119. This is a very general spring and damper model. This beam is based on the MAT_SPRING_GENERAL_NONLINEAR option. Additional unloading options have been included. The two nodes defining the beam may be coincident to give a zero length beam, or offset to give a finite length beam. For finite length discrete beams the absolute value of the variable SCOOR in the SECTION_BEAM input should be set to a value of 2.0 or 3.0 to give physically correct behavior. A triad is used to orient the beam for the directional springs.

Card 1

1	2	3	4	5	6	7	8
MID	RO	KT	KR	UNLDOPT	OFFSET	DAMPF	IFLAG
A8	F	F	F	I	F	F	I

Card 2

1	2	3	4	5	6	7	8
LCIDTR	LCIDTS	LCIDTT	LCIDRR	LCIDRS	LCIDRT		
I	I	I	I	I	I		

Card 3

1	2	3	4	5	6	7	8
LCIDTUR	LCIDTUS	LCIDTUT	LCIDRUR	LCIDRUS	LCIDRUT		
I	I	I	I	I	I		

Card 4

1	2	3	4	5	6	7	8
LCIDTDR	LCIDTDS	LCIDTDT	LCIDRDR	LCIDRDS	LCIDRDT		
I	I	I	I	I	I		

Card 5

Variable	LCIDTER	LCIDTES	LCIDTET	LCIDRER	LCIDRES	LCIDRET		
Type	I	I	I	I	I	I		

Card 6

Variable	UTFAILR	UTFAILS	UTFAILT	WTFAILR	WTFAILS	WTFAILT		
Type	F	F	F	F	F	F		

Card 7

Variable	UCFAILR	UCFAILS	UCFAILT	WCFAILR	WCFAILS	WCFAILT		
Type	F	F	F	F	F	F		

Card 8

Variable	IUR	IUS	IUT	IWR	IWS	IWT		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density, see also volume in *SECTION_BEAM definition.
KT	Translational stiffness for unloading option 2.0.
KR	Rotational stiffness for unloading option 2.0.
DAMPF	Damping factor for stability. Values in the neighborhood of unity are recommended. This damping factor is properly scaled to eliminate time step size dependency. Also, it is active if and only if the local stiffness is defined.

VARIABLE	DESCRIPTION
IFLAG	Flag for switching between the displacement (default IFLAG=0) and linear strain (IFLAG=1) formulations. The displacement formulation is the one used in all other models. For the linear strain formulation, the displacements and velocities are divided by the initial length of the beam.
UNLDOPT	Unloading option (Also see Figure 119.1.): EQ.0.0: Loading and unloading follow loading curve EQ.1.0: Loading follows loading curve, unloading follows unloading curve. The unloading curve ID if undefined is taken as the loading curve. EQ.2.0: Loading follows loading curve, unloading follows unloading stiffness, KT or KR, to the unloading curve. The loading and unloading curves may only intersect at the origin of the axes. EQ.3.0: Quadratic unloading from peak displacement value to a permanent offset.
OFFSET	Offset factor between 0 and 1.0 to determine permanent set upon unloading if the UNLDOPT=3.0. The permanent sets in compression and tension are equal to the product of this offset value and the maximum compressive and tensile displacements, respectively.
LCIDTR	Load curve ID defining translational force resultant along local r-axis versus relative translational displacement. If zero, no stiffness related forces are generated for this degree of freedom. The loading curves must be defined from the most negative displacement to the most positive displacement. The force does not need to increase monotonically. The curves in this input are linearly extrapolated when the displacement range falls outside the curve definition.
LCIDTS	Load curve ID defining translational force resultant along local s-axis versus relative translational displacement.
LCIDTT	Load curve ID defining translational force resultant along local t-axis versus relative translational displacement.
LCIDRR	Load curve ID defining rotational moment resultant about local r-axis versus relative rotational displacement.
LCIDRS	Load curve ID defining rotational moment resultant about local s-axis versus relative rotational displacement.
LCIDRT	Load curve ID defining rotational moment resultant about local t-axis versus relative rotational displacement.

VARIABLE	DESCRIPTION
LCIDTUR	Load curve ID defining translational force resultant along local r-axis versus relative translational displacement during unloading. The force values defined by this curve must increase monotonically from the most negative displacement to the most positive displacement. For UNLDOPT=1.0, the slope of this curve must equal or exceed the loading curve for stability reasons. This is not the case for UNLDOPT=2.0. For loading and unloading to follow the same path simply set LCIDTUR=LCIDTR. For options UNLDOPT=0.0 or 3.0 the unloading curve is not required.
LCIDTUS	Load curve ID defining translational force resultant along local s-axis versus relative translational displacement during unloading.
LCIDTUT	Load curve ID defining translational force resultant along local t-axis versus relative translational displacement during unloading.
LCIDRUR	Load curve ID defining rotational moment resultant about local r-axis versus relative rotational displacement during unloading.
LCIDRUS	Load curve ID defining rotational moment resultant about local s-axis versus relative rotational displacement during unloading.
LCIDRUT	Load curve ID defining rotational moment resultant about local t-axis versus relative rotational displacement during unloading. If zero, no viscous forces are generated for this degree of freedom.
LCIDTDR	Load curve ID defining translational damping force resultant along local r-axis versus relative translational velocity.
LCIDTDS	Load curve ID defining translational damping force resultant along local s-axis versus relative translational velocity.
LCIDTDT	Load curve ID defining translational damping force resultant along local t-axis versus relative translational velocity.
LCIDRDR	Load curve ID defining rotational damping moment resultant about local r-axis versus relative rotational velocity.
LCIDRDS	Load curve ID defining rotational damping moment resultant about local s-axis versus relative rotational velocity.
LCIDRDT	Load curve ID defining rotational damping moment resultant about local t-axis versus relative rotational velocity.
LCIDTER	Load curve ID defining translational damping force scale factor versus relative displacement in local r-direction.

VARIABLE	DESCRIPTION
LCIDTES	Load curve ID defining translational damping force scale factor versus relative displacement in local s-direction.
LCIDTET	Load curve ID defining translational damping force scale factor versus relative displacement in local t-direction.
LCIDRER	Load curve ID defining rotational damping moment resultant scale factor versus relative displacement in local r-rotation.
LCIDRES	Load curve ID defining rotational damping moment resultant scale factor versus relative displacement in local s-rotation.
LCIDRET	Load curve ID defining rotational damping moment resultant scale factor versus relative displacement in local t-rotation.
UTFAILR	Optional, translational displacement at failure in tension. If zero, the corresponding displacement, u_r, is not considered in the failure calculation.
UTFAILS	Optional, translational displacement at failure in tension. If zero, the corresponding displacement, u_s, is not considered in the failure calculation.
UTFAILT	Optional, translational displacement at failure in tension. If zero, the corresponding displacement, u_t, is not considered in the failure calculation.
WTFAILR	Optional, rotational displacement at failure in tension. If zero, the corresponding rotation, θ_r, is not considered in the failure calculation.
WTFAILS	Optional, rotational displacement at failure in tension. If zero, the corresponding rotation, θ_s, is not considered in the failure calculation.
WTFAILT	Optional rotational displacement at failure in tension. If zero, the corresponding rotation, θ_t, is not considered in the failure calculation.
UCFAILR	Optional, translational displacement at failure in compression. If zero, the corresponding displacement, u_r, is not considered in the failure calculation. Define as a positive number.
UCFAILS	Optional, translational displacement at failure in compression. If zero, the corresponding displacement, u_s, is not considered in the failure calculation. Define as a positive number.

VARIABLE	DESCRIPTION
UCFAILT	Optional, translational displacement at failure in compression. If zero, the corresponding displacement, u_t, is not considered in the failure calculation. Define as a positive number.
WCFAILR	Optional, rotational displacement at failure in compression. If zero, the corresponding rotation, θ_r, is not considered in the failure calculation. Define as a positive number.
WCFAILS	Optional, rotational displacement at failure in compression. If zero, the corresponding rotation, θ_s, is not considered in the failure calculation. Define as a positive number.
WCFAILT	Optional, rotational displacement at failure in compression. If zero, the corresponding rotation, θ_t, is not considered in the failure calculation. Define as a positive number.
IUR	Initial translational displacement along local r-axis.
IUS	Initial translational displacement along local s-axis.
IUT	Initial translational displacement along local t-axis.
IWR	Initial rotational displacement about the local r-axis.
IWS	Initial rotational displacement about the local s-axis.
IWT	Initial rotational displacement about the local t-axis.

Remarks:

Catastrophic failure, which is based on displacement resultants, occurs if either of the following inequalities are satisfied:

$$\left(\frac{u_r}{u_r^{tfail}}\right)^2+\left(\frac{u_s}{u_s^{tfail}}\right)^2+\left(\frac{u_t}{u_t^{tfail}}\right)^2+\left(\frac{\theta_r}{\theta_r^{tfail}}\right)^2+\left(\frac{\theta_s}{\theta_s^{tfail}}\right)^2+\left(\frac{\theta_t}{\theta_t^{tfail}}\right)^2-1.\geq 0$$

$$\left(\frac{u_r}{u_r^{cfail}}\right)^2+\left(\frac{u_s}{u_s^{cfail}}\right)^2+\left(\frac{u_t}{u_t^{cfail}}\right)^2+\left(\frac{\theta_r}{\theta_r^{cfail}}\right)^2+\left(\frac{\theta_s}{\theta_s^{cfail}}\right)^2+\left(\frac{\theta_t}{\theta_t^{cfail}}\right)^2-1.\geq 0$$

After failure the discrete element is deleted. If failure is included either the tension failure or the compression failure or both may be used.

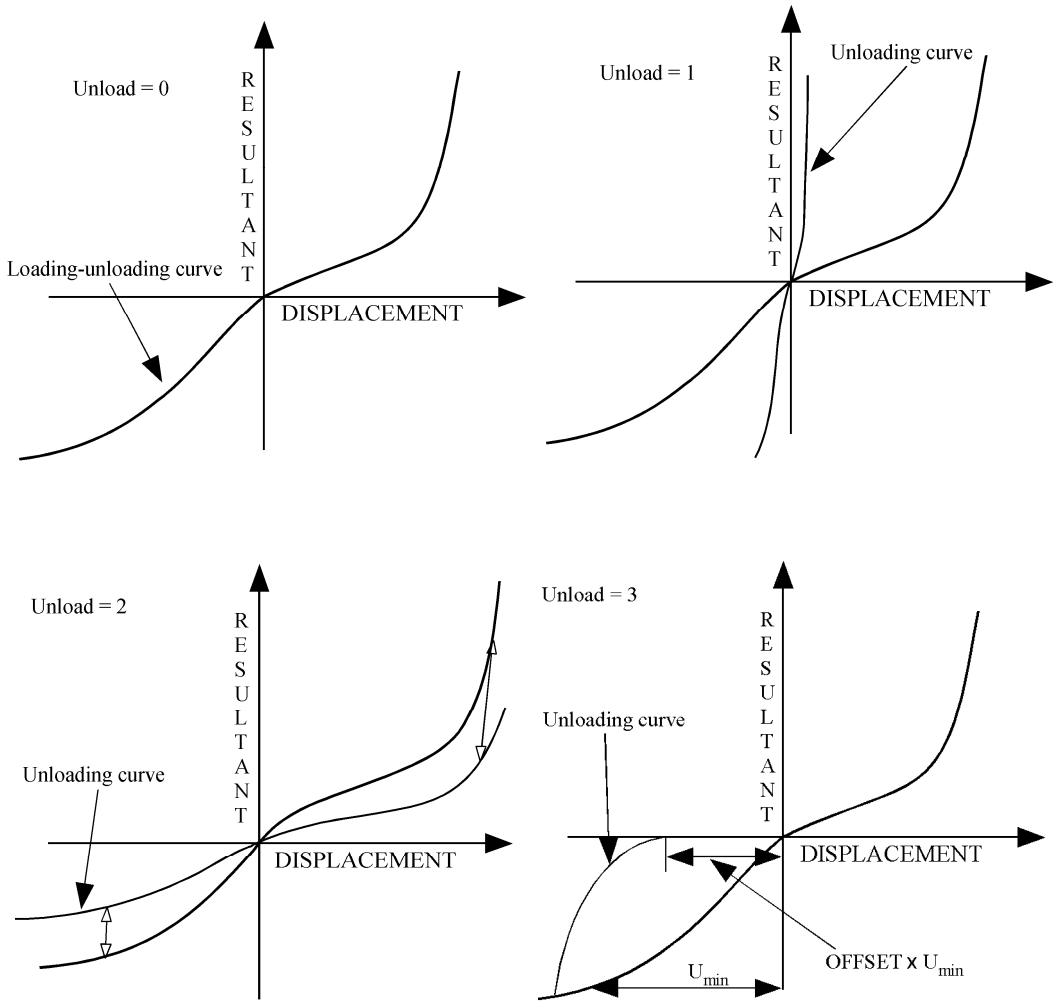


Figure 119.1. Load and unloading behavior.

There are two formulations for calculating the force. The first is the standard displacement formulation, where, for example, the force in a linear spring is

$$F = -K\Delta\ell$$

for a change in length of the beam of $\Delta\ell$. The second formulation is based on the linear strain, giving a force of

$$F = -K\frac{\Delta\ell}{\ell_0}$$

for a beam with an initial length of ℓ_0. This option is useful when there are springs of different lengths but otherwise similar construction since it automatically reduces the stiffness of the spring as the length increases, allowing an entire family of springs to be modeled with a single material. Note that all the displacement and velocity components are divided by the initial length, and therefore the scaling applies to the damping and rotational stiffness.

*MAT_GURSON

This is Material Type 120. This is the Gurson dilatational-plastic model. This model is available for shell and solid elements. A detailed description of this model can be found in the following references: Gurson [1975, 1977], Chu and Needleman [1980] and Tvergaard and Needleman [1984]. The implementation in LS-DYNA is based on the implementation of Feucht [1998] and Faßnacht [1999], which was recoded at LSTC. Strain rate dependency can be defined via a Table definition starting with the second formal release of version 971.

Card 1

Variable	MID	RO	E	PR	SIGY	N	Q1	Q2
Type	A8	F	F	F	F	F	F	F
Default	none	none	none	none	none	0.0	none	none

Card 2

Variable	FC	F0	EN	SN	FN	ETAN	ATYP	FF0
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

Card 3

Variable	EPS1	EPS2	EPS3	EPS4	EPS5	EPS6	EPS7	EPS8
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

Card 4

Variable	ES1	ES2	ES3	ES4	ES5	ES6	ES7	ES8
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

Card 5

Variable	L1	L2	L3	L4	FF1	FF2	FF3	FF4
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

Card 6

Variable	LCSS	LCLF	NUMINT	LCF0	LCFC	LCFN	VGTYP	
Type	F	F	F	F	F	F	F	
Default	0	0	1.0	0	0	0	0	

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.
SIGY	Yield stress.

VARIABLE	DESCRIPTION
N	Exponent for Power law. This value is only used if ATYP=1 and LCSS=0.
Q1	Gurson flow function parameter q_1.
Q2	Gurson flow function parameter q_2.
FC	Critical void volume fraction f_c where voids begin to aggregate. This value is only used if LCFC=0.
F0	Initial void volume fraction f_0. This value is only used if LCF0=0.
EN	Mean nucleation strain ε_N.
SN	Standard deviation s_N of the normal distribution of ε_N.
FN	Void volume fraction of nucleating particles f_N. This value is only used if LCFN=0.
ETAN	Hardening modulus. This value is only used if ATYP=2 and LCSS=0.
ATYP	Type of hardening. EQ.1.0: Power law. EQ.2.0: Linear hardening. EQ.3.0: 8 points curve.
FF0	Failure void volume fraction f_F. This value is only used if no curve is given by L1,FF1 – L4,FF4 and LCFF=0.
EPS1-EPS8	Effective plastic strain values. The first point must be zero corresponding to the initial yield stress. At least 2 points should be defined. These values are used if ATYP=3 and LCSS=0.
ES1-ES8	Corresponding yield stress values to EPS1 - EPS8. These values are used if ATYP=3 and LCSS=0.
L1-L4	Element length values. These values are only used if LCFF=0
FF1-FF4	Corresponding failure void volume fraction. These values are only used if LCFF=0.
LCSS	Load curve ID or Table ID. ATYP is ignored with this option. Load curve ID defining effective stress versus effective plastic strain. Table ID defines for each strain rate value a load curve ID giving the effective stress versus effective plastic strain for that rate (see MAT_024). The

VARIABLE	DESCRIPTION
	stress-strain curve for the lowest value of strain rate is used if the strain rate falls below the minimum value. Likewise, the stress-strain curve for the highest value of strain rate is used if the strain rate exceeds themaximum value. NOTE: The strain rate values defined in the table may be given as the natural logarithm of the strain rate. If the *first* stress-strain curve in the table corresponds to a negative strain rate, LS-DYNA assumes that the natural logarithm of the strain rate value is used. Since the tables are internally discretized to equally space the points, natural logarithms are necessary, for example, if the curves correspond to rates from 10.e-04 to 10.e+04. The table option is available starting with the second formal release of version 971.
LCFF	Load curve ID defining failure void volume fraction f_F versus element length.
NUMINT	Number of integration points which must fail before the element is deleted. This option is available for shells and solids. LT.0.0: |NUMINT| is percentage of integration points/layers which must fail before element fails. For fully integrated shells, a methodology is used where a layer fails if one integration point fails and then the given percentage of layers must fail before the element fails.
LCF0	Load curve ID defining initial void volume fraction f_0 versus element length. This option is available starting with the second formal release of version 971.
LCFC	Load curve ID defining critical void volume fraction f_c versus element length. This option is available starting with the second formal release of version 971.
LCFN	Load curve ID defining void volume fraction of nucleating particles f_N versus element length. This option is available starting with the second formal release of version 971.
VGTYP	Type of void growth behavior. EQ.0.0: Void growth in case of tension and void contraction in case of compression, but never below f_0 (default). EQ.1.0: Void growth only in case of tension. EQ.2.0: Void growth in case of tension and void contraction in case of compression, even below f_0.

Remarks:

The Gurson flow function is defined as:

$$\Phi = \frac{\sigma_M^2}{\sigma_Y^2} + 2q_1 f^* \cosh\left(\frac{3q_2\sigma_H}{2\sigma_Y}\right) - 1 - \left(q_1 f^*\right)^2 = 0$$

where σ_M is the equivalent von Mises stress, σ_Y is the yield stress, σ_H is the mean hydrostatic stress. The effective void volume fraction is defined as

$$f^*(f) = \begin{cases} f & f \le f_c \\ f_c + \dfrac{1/q_1 - f_c}{f_F - f_c}(f - f_c) & f > f_c \end{cases}$$

The growth of void volume fraction is defined as

$$\dot{f} = \dot{f}_G + \dot{f}_N$$

where the growth of existing voids is defined as

$$\dot{f}_G = (1 - f)\dot{\varepsilon}_{kk}^p$$

and nucleation of new voids is defined as

$$\dot{f}_N = A\dot{\varepsilon}_p$$

with function *A*

$$A = \frac{f_N}{S_N\sqrt{2\pi}} \exp\left(-\frac{1}{2}\left(\frac{\varepsilon_p - \varepsilon_N}{S_N}\right)^2\right)$$

Voids are nucleated only in tension.

History variables

Shell / Solid	*Description*
1 / 1	Void volume fraction
4 / 2	Triaxiality variable σ_H / σ_M
5 / 3	Effective strain rate
6 / 4	Growth of voids
7 / 5	Nucleation of voids
11 / 11	Dimensionless material damage value $= \begin{cases} (f - f_0)/(f_c - f_0) & \text{if } f \le f_c \\ 1 + (f - f_c)/(f_F - f_c) & \text{if } f > f_c \end{cases}$
13 / 13	Deviatoric part of microscopic plastic strain
14 / 14	Volumetric part of macroscopic plastic strain

***MAT_GURSON_JC**

This is an enhancement of Material Type 120. This is the Gurson model with additional Johnson-Cook failure criterion (parameters Card 5). This model is available for shell and solid elements. Strain rate dependency can be defined via Table. This model is available starting with the second formal release of version 971. An extension for void growth under shear-dominated states and for Johnson-Cook damage evolution is available starting with the fourth formal release of version 971 (optional Card 7).

Card 1

Variable	MID	RO	E	PR	SIGY	N	Q1	Q2
Type	A8	F	F	F	F	F	F	F
Default	none	none	none	none	None	0.0	none	none

Card 2

Variable	FC	F0	EN	SN	FN	ETAN	ATYP	FF0
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

Card 3

Variable	EPS1	EPS2	EPS3	EPS4	EPS5	EPS6	EPS7	EPS8
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

Card 4

Variable	SIG1	SIG2	SIG3	SIG4	SIG5	SIG6	SIG7	SIG8
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

Card 5

Variable	LCDAM	L1	L2	D1	D2	D3	D4	LCJC
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

Card 6

Variable	LCSS	LCFF	NUMINT	LCF0	LCFC	LCFN	VGTYP	
Type	F	F	F	F	F	F	F	
Default	0	0	1	0	0	0	0	

Optional Card (starting with version 971 release R4)

Card 7

Variable	KW	BETA	M					
Type	F	F	F					
Default	0	0	1.0					

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.
SIGY	Yield stress.
N	Exponent for Power law. This value is only used if ATYP=1 and LCSS=0.
Q1	Gurson flow function parameter q_1.
Q2	Gurson flow function parameter q_2.
FC	Critical void volume fraction f_c where voids begin to aggregate. This value is only used if LCFC=0.
F0	Initial void volume fraction f_0. This value is only used if LCF0=0.
EN	Mean nucleation strain ε_N.
SN	Standard deviation s_N of the normal distribution of ε_N.
FN	Void volume fraction of nucleating particles f_N. This value is only used if LCFN=0.
ETAN	Hardening modulus. This value is only used if ATYP=2 and LCSS=0.
ATYP	Type of hardening. EQ.1.0: Power law. EQ.2.0: Linear hardening. EQ.3.0: 8 points curve.
FF0	Failure void volume fraction f_F. This value is only used if LCFF=0.
EPS1-EPS8	Effective plastic strain values. The first point must be zero corresponding to the initial yield stress. At least 2 points should be defined. These values are used if ATYP=3 and LCSS=0.
ES1-ES8	Corresponding yield stress values to EPS1 - EPS8. These values are used if ATYP=3 and LCSS=0.

VARIABLE	DESCRIPTION
LCDAM	Load curve defining scaling factor Λ versus element length. Scales the Johnson-Cook failure strain (see remarks). If LCDAM=0, no scaling is performed.
L1	Lower triaxiality factor defining failure evolution (Johnson-Cook).
L2	Upper triaxiality factor defining failure evolution (Johnson-Cook).
D1-D4	Johnson-Cook damage parameters.
LCJC	Load curve defining scaling factor for Johnson-Cook failure versus triaxiality (see remarks). If LCJC > 0, parameters D1, D2 and D3 are ignored.
LCSS	Load curve ID or Table ID. ATYP is ignored with this option. Load curve ID defining effective stress versus effective plastic strain. Table ID defines for each strain rate value a load curve ID giving the effective stress versus effective plastic strain for that rate (see MAT_024). The stress-strain curve for the lowest value of strain rate is used if the strain rate falls below the minimum value. Likewise, the stress-strain curve for the highest value of strain rate is used if the strain rate exceeds the maximum value. NOTE: The strain rate values defined in the table may be given as the natural logarithm of the strain rate. If the *first* stress-strain curve in the table corresponds to a negative strain rate, LS-DYNA assumes that the natural logarithm of the strain rate value is used. Since the tables are internally discretized to equally space the points, natural logarithms are necessary, for example, if the curves correspond to rates from 10.e-04 to 10.e+04.
LCFF	Load curve ID defining failure void volume fraction f_F versus element length.
NUMINT	Number of through thickness integration points which must fail before the element is deleted. This option is available for shells and solids. LT.0.0: |NUMINT| is percentage of integration points/layers which must fail before element fails. For fully integrated shells, a methodology is used where a layer fails if one integration point fails and then the given percentage of layers must fail before the element fails.
LCF0	Load curve ID defining initial void volume fraction f_0 versus element length.
LCFC	Load curve ID defining critical void volume fraction f_c versus element length.

VARIABLE	DESCRIPTION
LCFN	Load curve ID defining void volume fraction of nucleating particles f_N versus element length.
VGTYP	Type of void growth behavior. EQ.0.0: Void growth in case of tension and void contraction in case of compression, but never below f_0 (default). EQ.1.0: Void growth only in case of tension. EQ.2.0: Void growth in case of tension and void contraction in case of compression, even below f_0.
KW	Parameter k_ω for void growth in shear-dominated states. See remarks.
BETA	Parameter β in Lode cosine function. See remarks.
M	Parameter for generalization of Johnson-Cook damage evolution. See remarks.

Remarks:

The Gurson flow function is defined as:

$$\Phi = \frac{\sigma_M^2}{\sigma_Y^2} + 2q_1 f^* \cosh\left(\frac{3q_2\sigma_H}{2\sigma_Y}\right) - 1 - \left(q_1 f^*\right)^2 = 0$$

where σ_M is the equivalent von Mises stress, σ_Y is the yield stress, σ_H is the mean hydrostatic stress. The effective void volume fraction is defined as

$$f^*(f) = \begin{cases} f & f \le f_c \\ f_c + \dfrac{1/q_1 - f_c}{f_F - f_c}(f - f_c) & f > f_c \end{cases}$$

The growth of void volume fraction is defined as

$$\dot{f} = \dot{f}_G + \dot{f}_N$$

where the growth of existing voids is defined as

$$\dot{f}_G = (1-f)\dot{\varepsilon}_{kk}^p + k_\omega \omega(\boldsymbol{\sigma})\, f\,(1-f)\dot{\varepsilon}_M^{pl}\frac{\sigma_Y}{\sigma_M}$$

The second term is an optional extension for shear failure proposed by Nahshon and Hutchinson [2008] with new parameter k_ω (=0 by default), effective plastic strain rate in the matrix $\dot{\varepsilon}_M^{pl}$, and Lode cosin function $\omega(\boldsymbol{\sigma})$:

$$\omega(\boldsymbol{\sigma}) = 1 - \xi^2 - \beta \cdot \xi(1-\xi), \qquad \xi = \cos(3\theta) = \frac{27}{2}\frac{J_3}{\sigma_M^3}$$

with parameter β, Lode angle θ and third deviatoric stress invariant J_3.

Nucleation of new voids is defined as

$$\dot{f}_N = A\dot{\varepsilon}_M^{pl}$$

with function *A*

$$A = \frac{f_N}{S_N\sqrt{2\pi}} \exp\left(-\frac{1}{2}\left(\frac{\varepsilon_M^{pl} - \varepsilon_N}{S_N} \right)^2 \right)$$

Voids are nucleated only in tension.

The Johnson-Cook failure criterion is added in this material model. Based on the triaxiality ratio σ_H / σ_M failure is calculated as:

a) $\sigma_H / \sigma_M > L_1$: Gurson model

b) $L_1 \geq \sigma_H / \sigma_M \geq L_2$: Gurson model and Johnson-Cook failure criteria

c) $\sigma_H / \sigma_M < L_2$: Gurson model

Johnson-Cook failure strain is defined as

$$\varepsilon_f = \left[D_1 + D_2 \exp\left(D_3 \frac{\sigma_H}{\sigma_M} \right) \right] (1 + D_4 \ln \dot{\varepsilon}) \Lambda$$

where D_1, D_2, D_3 and D_4 are the Johnson-Cook failure parameters and Λ is a function for including mesh-size dependency. An alternative expression can be used, where the first term of the above equation (including D1, D2 and D3) is replaced by a general function LCJC which depends on triaxiality

$$\varepsilon_f = LCJC\left(\frac{\sigma_H}{\sigma_M} \right) (1 + D_4 \ln \dot{\varepsilon}) \Lambda$$

The Johnson-Cook damage parameter D_f is calculated with the following evolution

$$\dot{D}_f = \frac{\dot{\varepsilon}^{pl}}{\varepsilon_f} \quad \rightarrow \quad D_f = \sum \frac{\Delta\varepsilon^{pl}}{\varepsilon_f} \begin{cases} <1 & \text{no failure} \\ \geq 1 & \text{failure} \end{cases}$$

where $\Delta\varepsilon^{pl}$ is the increment in effective plastic strain. A more general (non-linear) damage evolution is possible if M >1 is chosen:

$$\dot{D}_f = \frac{M}{\varepsilon_f} D_f^{\left(1-\frac{1}{M}\right)} \dot{\varepsilon}^{pl} \quad , \qquad M \geq 1.0$$

History variables

Shell / Solid	*Description*
1 / 1	Void volume fraction
4 / 2	Triaxiality variable σ_H / σ_M
5 / 3	Effective strain rate
6 / 4	Growth of voids
7 / 5	Nucleation of voids
8 / 6	Johnson-Cook failure strain ε_f
9 / 7	Johnson-Cook damage parameter D_f
10 / 8	Domain variable: = 0 elastic stress update = 1 region a) Gurson = 2 region b) Gurson + Johnson-Cook = 3 region c) Gurson
11 / 11	Dimensionless material damage value $= \begin{cases} (f - f_0)/(f_c - f_0) & \text{if } f \leq f_c \\ 1+(f - f_c)/(f_F - f_c) & \text{if } f > f_c \end{cases}$
13 / 13	Deviatoric part of microscopic plastic strain
14 / 14	Volumetric part of macroscopic plastic strain

*MAT_GURSON_RCDC

This is an enhancement of material Type 120. This is the Gurson model with the Wilkins Rc-Dc [Wilkins, et al., 1977] fracture model added. This model is available for shell and solid elements. A detailed description of this model can be found in the following references: Gurson [1975, 1977]; Chu and Needleman [1980]; and Tvergaard and Needleman [1984].

Card 1

Variable	MID	RO	E	PR	SIGY	N	Q1	Q2
Type	A8	F	F	F	F	F	F	F
Default	None	none	none	none	none	0.0	none	none

Card 2

Variable	FC	F0	EN	SN	FN	ETAN	ATYP	FF0
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

Card 3

Variable	EPS1	EPS2	EPS3	EPS4	EPS5	EPS6	EPS7	EPS8
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

Card 4

Variable	1	2	3	4	5	6	7	8
	ES1	ES2	ES3	ES4	ES5	ES6	ES7	ES8
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

Card 5

Variable								
	L1	L2	L3	L4	FF1	FF2	FF3	FF4
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

Card 6

Variable								
	LCSS	LCLF	NUMINT					
Type	F	F	F					
Default	0	0	1					

Card 7

Variable								
	ALPHA	BETA	GAMMA	D0	B	LAMBDA	DS	L
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.
SIGY	Yield stress.
N	Exponent for Power law. This value is only used if ATYP=1 and LCSS=0.
Q1	Parameter q_1.
Q1	Parameter q_2.
FC	Critical void volume fraction f_c
F0	Initial void volume fraction f_0.
EN	Mean nucleation strain ε_N.
SN	Standard deviation S_N of the normal distribution of ε_N.
FN	Void volume fraction of nucleating particles.
ETAN	Hardening modulus. This value is only used if ATYP=2 and LCSS=0.
ATYP	Type of hardening. EQ.1.0: Power law. EQ.2.0: Linear hardening. EQ.3.0: 8 points curve.
FF0	Failure void volume fraction. This value is used if no curve is given by the points L1,FF1 - L4,FF4 and LCLF=0.
EPS1-EPS8	Effective plastic strain values. The first point must be zero corresponding to the initial yield stress. This option is only used if ATYP equal to 3. At least 2 points should be defined. These values are used if ATYP=3 and LCSS=0.
ES1-ES8	Corresponding yield stress values to EPS1 - EPS8. These values are used if ATYP=3 and LCSS=0.

VARIABLE	DESCRIPTION
L1-L4	Element length values. These values are only used if LCLF=0.
FF1-FF4	Corresponding failure void volume fraction. These values are only used if LCLF=0.
LCSS	Load curve ID defining effective stress versus effective plastic strain. ATYP is ignored with this option.
LCLF	Load curve ID defining failure void volume fraction versus element length. The values L1-L4 and FF1-FF4 are ignored with this option.
NUMINT	Number of through thickness integration points which must fail before the element is deleted.
ALPHA	Parameter α. for the Rc-Dc model
BETA	Parameter β. for the Rc-Dc model
GAMMA	Parameter γ. for the Rc-Dc model
D0	Parameter D_0. for the Rc-Dc model
B	Parameter b. for the Rc-Dc model
LAMBDA	Parameter λ. for the Rc-Dc model
DS	Parameter D_s. for the Rc-Dc model
L	Characteristic element length for this material

Remarks:

The Gurson flow function is defined as:

$$\Phi = \frac{\sigma_M^2}{\sigma_Y^2} + 2q_1 f^* \cosh\left(\frac{3q_2\sigma_H}{2\sigma_Y}\right) - 1 - \left(q_1 f^*\right)^2 = 0$$

where σ_M is the equivalent von Mises stress, σ_Y is the Yield stress, σ_H is the mean hydrostatic stress. The effective void volume fraction is defined as

$$f^*(f) = \begin{cases} f & f \le f_c \\ f_c + \dfrac{1/q_1 - f_c}{f_F - f_c}(f - f_c) & f > f_c \end{cases}$$

The growth of the void volume fraction is defined as

$$\dot{f} = \dot{f}_G + \dot{f}_N$$

where the growth of existing voids is given as:

$$\dot{f}_G = (1-f)\dot{\varepsilon}_{kk}^p,$$

and nucleation of new voids as:

$$\dot{f}_N = A\dot{\varepsilon}_p$$

in which *A* is defined as

$$A = \frac{f_N}{S_N\sqrt{2\pi}}\exp\left(-\frac{1}{2}\left(\frac{\varepsilon_p - \varepsilon_N}{S_N}\right)^2\right)$$

The Rc-Dc model is defined as the following:

The damage D is given by

$$D = \int \omega_1\omega_2 d\varepsilon^p$$

where ε^p is the equivalent plastic strain,

$$\omega_1 = \left(\frac{1}{1-\gamma\sigma_m}\right)^\alpha$$

is a triaxial stress weighting term and

$$\omega_2 = (2 - A_D)^\beta$$

is a asymmetric strain weighting term.

In the above σ_m is the mean stress and

$$A_D = \max\left(\frac{S_2}{S_3}, \frac{S_2}{S_1}\right)$$

Fracture is initiated when the accumulation of damage is

$$\frac{D}{D_c} > 1$$

where D_c is the a critical damage given by

$$D_c = D_0\left(1 + b|\nabla D|^\lambda\right)$$

A fracture fraction

$$F = \frac{D - D_c}{D_s}$$

defines the degradations of the material by the Rc-Dc model.

The characteristic element length is used in the calculation of ∇D. Calculation of this factor is only done for element with smaller element length than this value.

*MAT_GENERAL_NONLINEAR_1DOF_DISCRETE_BEAM

This is Material Type 121. This is a very general spring and damper model. This beam is based on the MAT_SPRING_GENERAL_NONLINEAR option and is a one-dimensional version of the 6DOF_DISCRETE_BEAM above. The forces generated by this model act along a line between the two connected nodal points. Additional unloading options have been included.

Card 1

Variable	MID	RO	K	UNLDOPT	OFFSET	DAMPF		
Type	A8	F	F	I	F	F		

Card 2

Variable	LCIDT	LCIDTU	LCIDTD	LCIDTE				
Type	I	I	I	I				

Card 3

Variable	UTFAIL	UCFAIL	IU					
Type	F	F	F					

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density, see also volume in *SECTION_BEAM definition.
K	Translational stiffness for unloading option 2.0.
UNLDOPT	Unloading option (Also see Figure 119.1): EQ.0.0: Loading and unloading follow loading curve EQ.1.0: Loading follows loading curve, unloading follows unloading curve. The unloading curve ID if undefined is taken as the loading curve.

VARIABLE	DESCRIPTION
	EQ.2.0: Loading follows loading curve, unloading follows unloading stiffness, K, to the unloading curve. The loading and unloading curves may only intersect at the origin of the axes. EQ.3.0: Quadratic unloading from peak displacement value to a permanent offset.
OFFSET	Offset to determine permanent set upon unloading if the UNLDOPT=3.0. The permanent sets in compression and tension are equal to the product of this offset value and the maximum compressive and tensile displacements, respectively.
DAMPF	Damping factor for stability. Values in the neighborhood of unity are recommended. This damping factor is properly scaled to eliminate time step size dependency. Also, it is active if and only if the local stiffness is defined.
LCIDT	Load curve ID defining translational force resultant along the axis versus relative translational displacement. If zero, no stiffness related forces are generated for this degree of freedom. The loading curves must be defined from the most negative displacement to the most positive displacement. The force does not need to increase monotonically for the loading curve. The curves are extrapolated when the displacement range falls outside the curve definition.
LCIDTU	Load curve ID defining translational force resultant along the axis versus relative translational displacement during unloading. The force values defined by this curve must increase monotonically from the most negative displacement to the most positive displacement. For UNLDOPT=1.0, the slope of this curve must equal or exceed the loading curve for stability reasons. This is not the case for UNLDOPT=2.0. For loading and unloading to follow the same path simply set LCIDTU=LCIDT.
LCIDTD	Load curve ID defining translational damping force resultant along local the axis versus relative translational velocity.
LCIDTE	Load curve ID defining translational damping force scale factor versus relative displacement in along axis.
UTFAIL	Optional, translational displacement at failure in tension. If zero, failure in tension is not considered.
UCFAIL	Optional, translational displacement at failure in compression. If zero, failure in compression is not considered.
IU	Initial translational displacement along axis.

*MAT_HILL_3R

This is Material Type 122. This is Hill's 1948 planar anisotropic material model with 3 R values.

Card 1

1	2	3	4	5	6	7	8
MID	RO	E	PR	HR	P1	P2	
A8	F	F	F	F	F	F	

Card 2

R00	R45	R90	LCID	E0			
F	F	F	F	F			

Card 3

AOPT							
F							

Card 4

			A1	A2	A3		
			F	F	F		

Card 5

V1	V2	V3	D1	D2	D3	BETA	
F	F	F	F	F	F	F	

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
E	Young's modulus, E
PR	Poisson's ratio, ν
HR	Hardening rule: EQ.1.0: linear (default), EQ.2.0: exponential. EQ.3.0: load curve
P1	Material parameter: HR.EQ.1.0: Tangent modulus, HR.EQ.2.0: k, strength coefficient for exponential hardening
P2	Material parameter: HR.EQ.1.0: Yield stress HR.EQ.2.0: n, exponent
R00	R_{00}, Lankford parameter determined from experiments
R45	R_{45}, Lankford parameter determined from experiments
R90	R_{90}, Lankford parameter determined from experiments
LCID	load curve ID for the load curve hardening rule
E0	ε_0 for determining initial yield stress for exponential hardening. (Default=0.0)
AOPT	Material axes option (see **MAT_***OPTION* **TROPIC_ELASTIC** for a more complete description): EQ.0.0: locally orthotropic with material axes determined by element nodes 1, 2, and 4, as with *DEFINE_COORDINATE_NODES. EQ.2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR. EQ.3.0: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle, BETA, from a line in the plane of the element defined by the cross product of the vector **v** with the element normal.

VARIABLE	DESCRIPTION
	LT.0.0: the absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_NODES, *DEFINE_COORDINATE_SYSTEM or *DEFINE_COORDINATE_VECTOR). Available in R3 version of 971 and later.
XP YP ZP	Coordinates of point **p** for AOPT = 1.
A1 A2 A3	Components of vector **a** for AOPT = 2.
V1 V2 V3	Components of vector **v** for AOPT = 3.
D1 D2 D3	Components of vector **d** for AOPT = 2.
BETA	Material angle in degrees for AOPT = 3, may be overridden on the element card, see *ELEMENT_SHELL_BETA.

***MAT_MODIFIED_PIECEWISE_LINEAR_PLASTICITY_**{*OPTION*}

This is Material Type 123. An elasto-plastic material with an arbitrary stress versus strain curve and arbitrary strain rate dependency can be defined. This model is available for shell and solid elements. Another model, MAT_PIECEWISE_LINEAR_PLASTICITY, is similar but lacks the enhanced failure criteria. Failure is based on effective plastic strain, plastic thinning, the major principal in plane strain component, or a minimum time step size. See the discussion under the model description for MAT_PIECEWISE_LINEAR_PLASTICITY if more information is desired.

Available options include:

<BLANK>

RATE

RTCL

The "RATE" option is used to account for rate dependence of plastic thinning failure. The "RTCL" option is used to activate RTCL damage. One additional card is needed with either option.

Card 1

Variable	MID	RO	E	PR	SIGY	ETAN	FAIL	TDEL
Type	A8	F	F	F	F	F	F	F
Default	none	none	none	none	none	0.0	10.E+20	0

Card 2

Variable	C	P	LCSS	LCSR	VP	EPSTHIN	EPSMAJ	NUMINT
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

Card 3	1	2	3	4	5	6	7	8
Variable	EPS1	EPS2	EPS3	EPS4	EPS5	EPS6	EPS7	EPS8
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

Card 4

Variable	ES1	ES2	ES3	ES4	ES5	ES6	ES7	ES8
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

Card 5 is required if and only if either the RATE or RTCL option is active.

Card 5	1	2	3	4	5	6	7	8
Variable	LCTSRF	EPS0						
Type	I	F						
Default	0	0						

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.
SIGY	Yield stress.

VARIABLE	DESCRIPTION
ETAN	Tangent modulus, ignored if (LCSS.GT.0) is defined.
FAIL	Failure flag. LT.0.0:User defined failure subroutine is called to determine failure EQ.0.0: Failure is not considered. This option is recommended if failure is not of interest since many calculations will be saved. GT.0.0: Plastic strain to failure. When the plastic strain reaches this value, the element is deleted from the calculation.
TDEL	Minimum time step size for automatic element deletion.
C	Strain rate parameter, C, see formula below.
P	Strain rate parameter, P, see formula below.
LCSS	Load curve ID or Table ID. Load curve ID defining effective stress versus effective plastic strain. If defined EPS1-EPS8 and ES1-ES8 are ignored. The table ID defines for each strain rate value a load curve ID giving the stress versus effective plastic strain for that rate, See Figure 24.1. The stress versus effective plastic strain curve for the lowest value of strain rate is used if the strain rate falls below the minimum value. Likewise, the stress versus effective plastic strain curve for the highest value of strain rate is used if the strain rate exceeds the maximum value. The strain rate parameters: C and P, the curve ID, LCSR, EPS1-EPS8, and ES1-ES8 are ignored if a Table ID is defined. NOTE: The strain rate values defined in the table may be given as the natural logarithm of the strain rate. If the *first* stress-strain curve in the table corresponds to a negative strain rate, LS-DYNA assumes that the natural logarithm of the strain rate value is used. Since the tables are internally discretized to equally space the points, natural logarithms are necessary, for example, if the curves correspond to rates from 10.e-04 to 10.e+04. Computing the natural logarithm of the strain rate does slow the stress update down significantly on some computers.
LCSR	Load curve ID defining strain rate scaling effect on yield stress.
VP	Formulation for rate effects: EQ.0.0: Scale yield stress (default), EQ.1.0: Viscoplastic formulation (recommended).
EPSTHIN	Thinning plastic strain at failure. This number should be given as a positive number.
EPSMAJ	Major in plane strain at failure. LT.0: EPSMAJ=|EPSMAJ| and filtering is activated. The last twelve values of the major strain is stored at each integration point and the average value is used to determine failure.

NUMINT	Number of integration points which must fail before the element is deleted. (If zero, all points must fail.) For fully integrated shell formulations, each of the 4*NIP integration points are counted individually in determining a total for failed integration points. NIP is the number of through-thickness integration points.

VARIABLE	DESCRIPTION
NUMINT	Number of integration points which must fail before the element is deleted. (If zero, all points must fail.) For fully integrated shell formulations, each of the 4*NIP integration points are counted individually in determining a total for failed integration points. NIP is the number of through-thickness integration points. As NUMINT approaches the total number of integration points (NIP for under integrated shells, 4*NIP for fully integrated shells), the chance of instability increases. LT.0.0: |NUMINT| is percentage of integration points/layers which must fail before element fails. For fully integrated shells, a methodology is used where a layer fails if one integration point fails and then the given percentage of layers must fail before the element fails.
EPS1-EPS8	Effective plastic strain values (optional if SIGY is defined). At least 2 points should be defined. The first point must be zero corresponding to the initial yield stress. WARNING: If the first point is nonzero the yield stress is extrapolated to determine the initial yield. If this option is used SIGY and ETAN are ignored and may be input as zero.
ES1-ES8	Corresponding yield stress values to EPS1 - EPS8.
LCTSRF	Load curve that defines the thinning plastic strain at failure as a function of the plastic strain rate.
EPS0	EPS0 parameter for RTCL damage. EQ.0.0: (default) RTCL damage is inactive. GT.0.0: RTCL damage is active

*MAT_PLASTICITY_COMPRESSION_TENSION

This is Material Type 124. An isotropic elastic-plastic material where unique yield stress versus plastic strain curves can be defined for compression and tension. Also, failure can occur based on a plastic strain or a minimum time step size. Rate effects on the yield stress are modeled either by using the Cowper-Symonds strain rate model or by using two load curves that scale the yield stress values in compression and tension, respectively. Material rate effects, which are independent of the plasticity model, are based on a 6-term Prony series Maxwell mode that generates an additional stress tensor. The viscous stress tensor is superimposed on the stress tensor generated by the plasticity.

Card 1

Variable	MID	RO	E	PR	C	P	FAIL	TDEL
Type	A8	F	F	F	F	F	F	F
Default	none	none	none	none	0	0	10.E+20	0

Card 2

Variable	LCIDC	LCIDT	LCSRC	LCSRT	SRFLAG	LCFAIL		
Type	I	I	I	I	F	I		
Default	0	0	0	0	0	0		

Card 3

Variable	PC	PT	PCUTC	PCUTT	PCUTF			
Type	F	F	F	F	F			
Default	0	0	0	0	0			

Card 4	1	2	3	4	5	6	7	8
Variable	K							
Type	F							

Card Format for viscoelastic constants. Up to 6 cards may be input. A keyword card (with a "*" in column 1) terminates this input if less than 6 cards are used.

Optional Cards	1	2	3	4	5	6	7	8
Variable	GI	BETAI						
Type	F	F						

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.
C	Strain rate parameter, C, see formula below.
P	Strain rate parameter, P, see formula below.
FAIL	Failure flag. LT.0.0: User defined failure subroutine is called to determine failure EQ.0.0: Failure is not considered. This option is recommended if failure is not of interest since many calculations will be saved. GT.0.0: Plastic strain to failure. When the plastic strain reaches this value, the element is deleted from the calculation.
TDEL	Minimum time step size for automatic element deletion.
LCIDC	Load curve ID defining yield stress versus effective plastic strain in compression.

VARIABLE	DESCRIPTION
LCIDT	Load curve ID defining yield stress versus effective plastic strain in tension.
LCSRC	Optional load curve ID defining strain rate scaling effect on yield stress when the material is in compression.
LCSRT	Optional load curve ID defining strain rate scaling effect on yield stress when the material is in tension.
SRFLAG	Formulation for rate effects: EQ.0.0: Total strain rate, EQ.1.0: Deviatoric strain rate.
LCFAIL	Load curve ID defining failure strain versus strain rate.
PC	Compressive mean stress (pressure) at which the yield stress follows load curve ID, LCIDC. If the pressure falls between PC and PT a weighted average of the two load curves is used.
PT	Tensile mean stress at which the yield stress follows load curve ID, LCIDT.
PCUTC	Pressure cut-off in compression (PCUTC must be greater than or equal to zero). This option applies only to solid elements. When the pressure cut-off is reached the deviatoric stress tensor is set to zero and the pressure remains at its compressive value. Like the yield stress, PCUTC is scaled to account for rate effects.
PCUTT	Pressure cut-off in tension (PCUTT must be less than or equal to zero). This option applies only to solid elements. When the pressure cut-off is reached the deviatoric stress tensor and tensile pressure is set to zero. Like the yield stress, PCUTT is scaled to account for rate effects.
PCUTF	Pressure cut-off flag activation. EQ.0.0: Inactive, EQ.1.0: Active.
K	Optional bulk modulus for the viscoelastic material. If nonzero a Kelvin type behavior will be obtained. Generally, K is set to zero.
GI	Optional shear relaxation modulus for the ith term
BETAI	Optional shear decay constant for the ith term

Remarks:

The stress strain behavior follows a different curve in compression than it does in tension. Tension is determined by the sign of the mean stress where a positive mean stress (i.e., a negative pressure) is indicative of tension. Two curves must be defined giving the yield stress versus effective plastic strain for both the tension and compression regimes.

Strain rate may be accounted for using the Cowper and Symonds model which scales the yield stress with the factor:

$$1+\left(\frac{\dot{\varepsilon}}{C}\right)^{1/p}$$

where $\dot{\varepsilon}$ is the strain rate. $\dot{\varepsilon}=\sqrt{\dot{\varepsilon}_{ij}\dot{\varepsilon}_{ij}}$.

*MAT_KINEMATIC_HARDENING_TRANSVERSELY_ANISOTROPIC

This is Material Type 125. This material model combines Yoshida's non-linear kinematic hardening rule with material type 37. Yoshida's theory uses two surfaces to describe the hardening rule: the yield surface and the bounding surface. In the forming process, the yield surface does not change in size, but its center translates with deformation; the bounding surface changes both in size and location. This model allows the change of Young's modulus as a function of effective plastic strain as proposed by Yoshida [2003]. This material type is available for shells, thick shells and solid elements.

Card 1

Variable	MID	RO	E	PR	R			
Type	A8	F	F	F	F			
Default	none	None	none	none	none			

Card 2

Variable	CB	Y	SC	K	RSAT	SB	H	
Type	F	F	F	F	F	F	F	
Default	none	none	none	none	none	none	none	

Card 3

Variable	EA	COE	IOP	C1	C2			
Type	F	F	I	F	F			
Default	none	none	0	none	none			

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
E	Young's Modulus
PR	Poisson's ratio
R	Anisotropic hardening parameter
CB	The uppercase B defined in the following equations.
Y	Hardening parameter as defined in the following equations.
SC	The lowercase c defined in the following equations.
K	Hardening parameter as defined in the following equations.
RSAT	Hardening parameter as defined in the following equations.
SB	The lowercase b as defined in the following equations.
H	Anisotropic parameter associated with work-hardening stagnation.
EA	Variable controlling the change of Young's modulus, E^A in the following equations.
COE	Variable controlling the change of Young's modulus, ζ in the following equations.
SRFLAG	Modified kinematic hardening rule flag: EQ.0: Original Yoshida formulation, EQ.1: Modified formulation.
C1, C2	Constants used to modify R:

$$R = RSAT\left[(C_1 + \bar{\varepsilon}^p)^{c_2} - C_1^{c_2}\right]$$

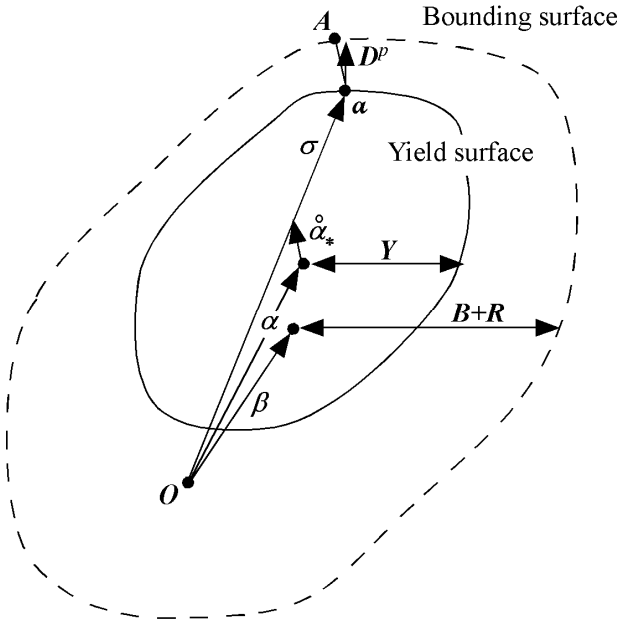


Figure 125.1 Schematic illustration of the two-surface model

The above figure is a schematic illustration of the two-surface kinematic model. O is the original center of the yield surface, α_* is the current center for the yield surface; α is the center of the bounding surface. β represents the relative position of the centers of the two surfaces. Y is the size of the yield surface and is constant throughout the deformation process. B+R represents the size of the bounding surface, with R being associated with isotropic hardening.

$$\alpha_* = \alpha - \beta$$

$$\alpha_* = c[(\frac{a}{Y})(\sigma - \alpha) - \sqrt{\frac{a}{\alpha_*}}\alpha_*]\bar{\varepsilon}^p$$

$$a = B + R - Y$$

The change of size and location for the bounding surface is defined as

$$\dot{R} = k(R_{sat} - R)\dot{\bar{\varepsilon}}^p,$$

$$\overset{o}{\beta}' = k(\frac{2}{3}bD - \beta'\dot{\bar{\varepsilon}}^p)$$

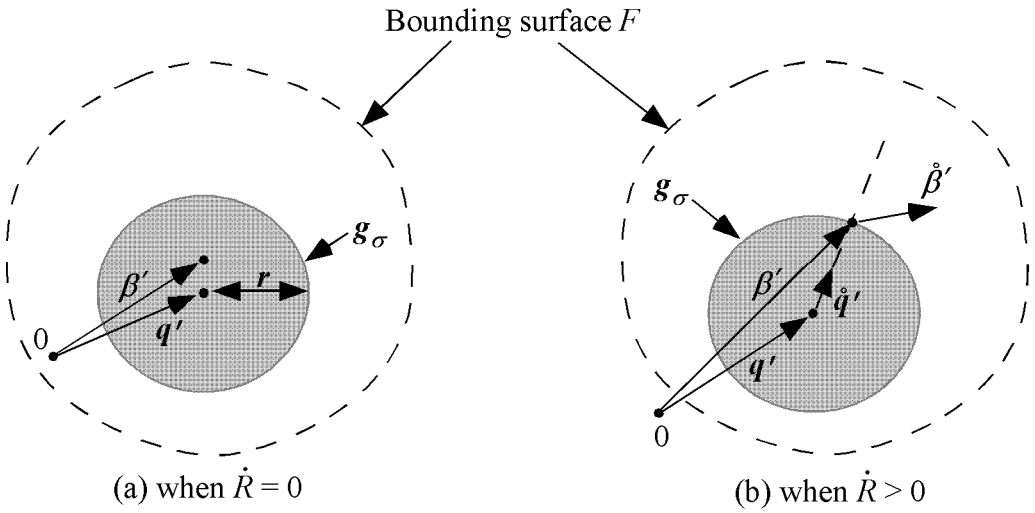
$$\sigma_{bound} = B + R + \beta$$

In Yoshida's model, this is work-hardening stagnation in the unloading process, and it is described as:

$$g_\sigma(\sigma',q',r') = \frac{3}{2}(\sigma'-q'):(\sigma'-q') - r^2$$

$$\overset{o}{q}' = \mu(\beta'-q')$$

$$r = h\Gamma, \Gamma = \frac{3(\beta'-q'):\overset{o}{\beta}'}{2r}$$



(a) when $\dot{R} = 0$ (b) when $\dot{R} > 0$

Young's modulus is defined as a function of effective strain:

$$E = E_0 - (E_0 - E_A)(1 - exp(-\zeta\bar{\varepsilon}^p))$$

*MAT_MODIFIED_HONEYCOMB

This is Material Type 126. The major use of this material model is for aluminum honeycomb crushable foam materials with anisotropic behavior. Three yield surfaces are available. In the first, nonlinear elastoplastic material behavior can be defined separately for all normal and shear stresses, which are considered to be fully uncoupled. In the second, a yield surface is defined that considers the effects of off-axis loading. The second yield surface is transversely isotropic. A drawback of this second yield surface is that the material can collapse in a shear mode due to low shear resistance. There was no obvious way of increasing the shear resistance without changing the behavior in purely uniaxial compression. Therefore, in the third option, the model has been modified so that the user can prescribe the shear and hydrostatic resistance in the material without affecting the uniaxial behavior. The choice of the second yield surface is flagged by the sign of the first load curve ID, LCA. The third yield surface is flagged by the sign of ECCU, which becomes the initial stress yield limit in simple shear. A description is given below.

The development of the second and third yield surfaces are based on experimental test results of aluminum honeycomb specimens at Toyota Motor Corporation.

The default element for this material is solid type 0, a nonlinear spring type brick element. *The recommended hourglass control is the type 2 viscous formulation for one point integrated solid elements. The stiffness form of the hourglass control when used with this constitutive model can lead to nonphysical results since strain localization in the shear modes can be inhibited.*

Card 1

Variable	MID	RO	E	PR	SIGY	VF	MU	BULK
Type	A8	F	F	F	F	F	F	F
Default	none	None	None	none	none	none	.05	0.0

Card 2

Variable	LCA	LCB	LCC	LCS	LCAB	LCBC	LCCA	LCSR
Type	F	F	F	F	F	F	F	F
Default	none	LCA	LCA	LCA	LCS	LCS	LCS	optional

Card 3

	1	2	3	4	5	6	7	8
Variable	EAAU	EBBU	ECCU	GABU	GBCU	GCAU	AOPT	MACF
Type	F	F	F	F	F	F		I

Card 4

Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card 5

Variable	D1	D2	D3	TSEF	SSEF	VREF	TREF	SHDFLG
Type	F	F	F	F	F	F	F	F

Define if and only if LCSR=-1.0

Card 6

Variable	LCSRA	LCSRB	LCSRC	LCSRAB	LCSRBC	LCSCA		
Type	F	F	F	F	F	F		

Define if and only if AOPT=3 or AOPT=4

Card 6

Variable	V1	V2	V3					
Type	F	F	F					

VARIABLE	DESCRIPTION

MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
E	Young's modulus for compacted honeycomb material.
PR	Poisson's ratio for compacted honeycomb material.
SIGY	Yield stress for fully compacted honeycomb.

VARIABLE	DESCRIPTION
VF	Relative volume at which the honeycomb is fully compacted. This parameter is ignored for corotational solid elements, types 0 and 9.
MU	μ, material viscosity coefficient. (default=.05) Recommended.
BULK	Bulk viscosity flag: EQ.0.0: bulk viscosity is not used. This is recommended. EQ.1.0: bulk viscosity is active and μ=0 This will give results identical to previous versions of LS-DYNA.
LCA	Load curve ID, see *DEFINE_CURVE: LCA.LT.0: Yield stress as a function of the angle off the material axis in degrees. LCA.GT.0: sigma-aa versus normal strain component aa. For the corotational solid elements, types 0 and 9, engineering strain is expected, but for all other solid element formulations a logarithmic strain is expected. See Remarks.
LCB	Load curve ID, see *DEFINE_CURVE: LCA.LT.0: strong axis hardening stress as a function of the volumetric strain. LCA.GT.0: sigma-bb versus normal strain component bb. For the corotational solid elements, types 0 and 9, engineering strain is expected, but for all other solid element formulations a logarithmic strain is expected. Default LCB=LCA. See Remarks.
LCC	Load curve ID, see *DEFINE_CURVE: LCA.LT.0: weak axis hardening stress as a function of the volumetric strain. LCA.GT.0: sigma-cc versus normal strain component cc. For the corotational solid elements, types 0 and 9, engineering strain is expected, but for all other solid element formulations a logarithmic strain is expected. Default LCC=LCA. See Remarks.
LCS	Load curve ID, see *DEFINE_CURVE: LCA.LT.0: damage curve giving shear stress multiplier as a function of the shear strain component. This curve definition is optional and may be used if damage is desired. IF SHDFLG=0 (the default), the damage value multiplies the stress every time step and the stress is updated incrementally. The damage curve should be set to unity until failure begins. After failure the value should drop to 0.999 or 0.99 or any number between zero and one depending on how many steps are needed to zero the stress. Alternatively, if SHDFLG=1, the damage value is treated as a factor that scales the shear stress compared to the undamaged value.

VARIABLE	DESCRIPTION
	LCA.GT.0: shear stress versus shear strain. For the corotational solid elements, types 0 and 9, engineering strain is expected, but for all other solid element formulations a shear strain based on the deformed configuration is used. Default LCS=LCA. Each component of shear stress may have its own load curve. See Remarks.
LCAB	Load curve ID, see *DEFINE_CURVE. Default LCAB=LCS: LCA.LT.0: damage curve giving shear ab-stress multiplier as a function of the ab-shear strain component. This curve definition is optional and may be used if damage is desired. See LCS above. LCA.GT.0: sigma-ab versus shear strain-ab. For the corotational solid elements, types 0 and 9, engineering strain is expected, but for all other solid element formulations a shear strain based on the deformed configuration is used. See Remarks.
LCBC	Load curve ID, see *DEFINE_CURVE. Default LCBC=LCS: LCA.LT.0: damage curve giving bc-shear stress multiplier as a function of the ab-shear strain component. This curve definition is optional and may be used if damage is desired. See LCS above. LCA.GT.0: sigma-bc versus shear strain-bc. For the corotational solid elements, types 0 and 9, engineering strain is expected, but for all other solid element formulations a shear strain based on the deformed configuration is used. See Remarks.
LCCA	Load curve ID, see *DEFINE_CURVE. Default LCCA=LCS: LCA.LT.0: damage curve giving ca-shear stress multiplier as a function of the ca-shear strain component. This curve definition is optional and may be used if damage is desired. See LCS above. LCA.GT.0: sigma-ca versus shear strain-ca. For the corotational solid elements, types 0 and 9, engineering strain is expected, but for all other solid element formulations a shear strain based on the deformed configuration is used. See Remarks.
LCSR	Load curve ID, see *DEFINE_CURVE, for strain-rate effects defining the scale factor versus effective strain rate $\dot{\bar{\varepsilon}} = \sqrt{\frac{2}{3}(\dot{\varepsilon}'_{ij}\dot{\varepsilon}'_{ij})}$. This is optional. The curves defined above are scaled using this curve.
EAAU	Elastic modulus E_{aau} in uncompressed configuration.
EBBU	Elastic modulus E_{bbu} in uncompressed configuration.
ECCU	Elastic modulus E_{ccu} in uncompressed configuration. LT.0.0: σ_d^Y, |ECCU| initial stress limit (yield) in simple shear. Also, LCA<0 to activate the transversely isotropic yield surface.

VARIABLE	DESCRIPTION
GABU	Shear modulus G_{abu} in uncompressed configuration.
GBCU	Shear modulus G_{bcu} in uncompressed configuration.
GCAU	Shear modulus G_{cau} in uncompressed configuration. ECCU.LT.0.0: σ_p^Y, GCAU initial stress limit (yield) in hydrostatic compression. Also, LCA<0 to activate the transversely isotropic yield surface.
AOPT	Material axes option (see MAT_*OPTION* TROPIC_ELASTIC for a more complete description): EQ.0.0: locally orthotropic with material axes determined by element nodes 1, 2, and 4, as with *DEFINE_COORDINATE_NODES. EQ.1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center; this is the a-direction. EQ.2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR. EQ.3.0: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle, BETA, from a line in the plane of the element defined by the cross product of the vector **v** with the element normal. The plane of a solid element is the midsurface between the inner surface and outer surface defined by the first four nodes and the last four nodes of the connectivity of the element, respectively. EQ.4.0: locally orthotropic in cylindrical coordinate system with the material axes determined by a vector **v**, and an originating point, P, which define the centerline axis. This option is for solid elements only. LT.0.0: the absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_NODES, *DEFINE_COORDINATE_SYSTEM or *DEFINE_COORDINATE_VECTOR). Available starting with the R3 release of Version 971.
MACF	Material axes change flag: EQ.1: No change, default, EQ.2: switch material axes a and b, EQ.3: switch material axes a and c, EQ.4: switch material axes b and c.
XP YP ZP	Coordinates of point **p** for AOPT = 1.
A1 A2 A3	Components of vector **a** for AOPT = 2.
D1 D2 D3	Components of vector **d** for AOPT = 2.

VARIABLE	DESCRIPTION
V1 V2 V3	Define components of vector **v** for AOPT = 3 and 4.
TSEF	Tensile strain at element failure (element will erode).
SSEF	Shear strain at element failure (element will erode).
VREF	This is an optional input parameter for solid elements types 1, 2, 3, 4, and 10. Relative volume at which the reference geometry is stored. At this time the element behaves like a nonlinear spring. The TREF, below, is reached first then VREF will have no effect.
TREF	This is an optional input parameter for solid elements types 1, 2, 3, 4, and 10. Element time step size at which the reference geometry is stored. When this time step size is reached the element behaves like a nonlinear spring. If VREF, above, is reached first then TREF will have no effect.
SHDFLG	Flag defining treatment of damage from curves LCS, LCAB, LCBC and LCCA (relevant only when LCA < 0): EQ.0.0: Damage reduces shear stress every time step, EQ.1.0: Damage = (shear stress)/(undamaged shear stress)
LCSRA	Optional load curve ID if LCSR=-1, see *DEFINE_CURVE, for strain rate effects defining the scale factor for the yield stress in the *a*-direction versus the ***natural logarithm*** of the absolute value of deviatoric strain rate in the *a*-direction. This curve is optional. The scale factor for the lowest value of strain rate defined by the curve is used if the strain rate is zero. The scale factor for the highest value of strain rate defined by the curve also defines the upper limit of the scale factor.
LCSRB	Optional load curve ID if LCSR=-1, see *DEFINE_CURVE, for strain rate effects defining the scale factor for the yield stress in the *b*-direction versus the ***natural logarithm*** of the absolute value of deviatoric strain rate in the *b*-direction. This curve is optional. The scale factor for the lowest value of strain rate defined by the curve is used if the strain rate is zero. The scale factor for the highest value of strain rate defined by the curve also defines the upper limit of the scale factor.
LCSRC	Similar definition as for LCSA and LCSB above.
LCSRAB	Similar definition as for LCSA and LCSB above.
LCSRBC	Similar definition as for LCSA and LCSB above.
LCSRCA	Similar definition as for LCSA and LCSB above.

Remarks:

For efficiency it is strongly recommended that the load curve ID's: LCA, LCB, LCC, LCS, LCAB, LCBC, and LCCA, contain exactly the same number of points with corresponding strain values on the abscissa. If this recommendation is followed the cost of the table lookup is insignificant. Conversely, the cost increases significantly if the abscissa strain values are not consistent between load curves.

For solid element formulations 1 and 2, the behavior before compaction is orthotropic where the components of the stress tensor are uncoupled, i.e., an a component of strain will generate resistance in the local *a*-direction with no coupling to the local *b* and *c* directions. The elastic moduli vary from their initial values to the fully compacted values linearly with the relative volume:

$$E_{aa} = E_{aau} + \beta\left(E - E_{aau}\right) \qquad G_{ab} = E_{abu} + \beta\left(G - G_{abu}\right)$$

$$E_{bb} = E_{bbu} + \beta\left(E - E_{bbu}\right) \qquad G_{bc} = G_{bcu} + \beta\left(G - G_{bcu}\right)$$

$$E_{cc} = E_{ccu} + \beta\left(E - E_{ccu}\right) \qquad G_{ca} = G_{cau} + \beta\left(G - G_{cau}\right)$$

where

$$\beta = \max\left[\min\left(\tfrac{1-V}{1-V_f},1\right),0\right]$$

and G is the elastic shear modulus for the fully compacted honeycomb material

$$G = \frac{E}{2\left(1+\nu\right)}$$

The relative volume, V, is defined as the ratio of the current volume over the initial volume, and typically, V=1 at the beginning of a calculation.

For corotational solid elements, types 0 and 9, the components of the stress tensor remain uncoupled and the uncompressed elastic moduli are used, that is, the fully compacted elastic moduli are ignored.

The load curves define the magnitude of the stress as the material undergoes deformation. The first value in the curve should be less than or equal to zero corresponding to tension and increase to full compaction. **Care should be taken when defining the curves so the extrapolated values do not lead to negative yield stresses.**

At the beginning of the stress update we transform each element's stresses and strain rates into the local element coordinate system. For the uncompacted material, the trial stress components are updated using the elastic interpolated moduli according to:

$$\sigma_{aa}^{n+1^{trial}} = \sigma_{aa}^{n} + E_{aa}\Delta\varepsilon_{aa} \qquad \sigma_{ab}^{n+1^{trial}} = \sigma_{ab}^{n} + 2G_{ab}\Delta\varepsilon_{ab}$$

$$\sigma_{bb}^{n+1^{trial}} = \sigma_{bb}^{n} + E_{bb}\Delta\varepsilon_{bb} \qquad \sigma_{bc}^{n+1^{trial}} = \sigma_{bc}^{n} + 2G_{bc}\Delta\varepsilon_{bc}$$

$$\sigma_{cc}^{n+1^{trial}} = \sigma_{cc}^{n} + E_{cc}\Delta\varepsilon_{cc} \qquad \sigma_{ca}^{n+1^{trial}} = \sigma_{ca}^{n} + 2G_{ca}\Delta\varepsilon_{ca}$$

If LCA>0, each component of the updated stress tensor is checked to ensure that it does not exceed the permissible value determined from the load curves, e.g., if

$$\left|\sigma_{ij}^{n+1^{trial}}\right| > \lambda\sigma_{ij}\left(\varepsilon_{ij}\right)$$

then

$$\sigma_{ij}^{n+1} = \sigma_{ij}\left(\varepsilon_{ij}\right)\frac{\lambda\sigma_{ij}^{n+1^{trial}}}{\left|\sigma_{ij}^{n+1^{trial}}\right|}$$

On Card 3 $\sigma_{ij}\left(\varepsilon_{ij}\right)$ is defined in the load curve specified in columns 31-40 for the aa stress component, 41-50 for the bb component, 51-60 for the cc component, and 61-70 for the ab, bc, cb shear stress components. The parameter λ is either unity or a value taken from the load curve number, LCSR, that defines λ as a function of strain-rate. Strain-rate is defined here as the Euclidean norm of the deviatoric strain-rate tensor.

If LCA<0, a transversely isotropic yield surface is obtained where the uniaxial limit stress, $\sigma^{y}\left(\varphi,\varepsilon^{vol}\right)$, can be defined as a function of angle φ with the strong axis and volumetric strain, ε^{vol}. In order to facilitate the input of data to such a limit stress surface, the limit stress is written as:

$$\sigma^{y}\left(\varphi,\varepsilon^{vol}\right) = \sigma^{b}\left(\varphi\right) + \left(\cos\varphi\right)^{2}\sigma^{s}\left(\varepsilon^{vol}\right) + \left(\sin\varphi\right)^{2}\sigma^{w}\left(\varepsilon^{vol}\right)$$

where the functions σ^{b}, σ^{s}, and σ^{w} are represented by load curves LCA, LCB, LCC, respectively. The latter two curves can be used to include the stiffening effects that are observed as the foam material crushes to the point where it begins to lock up. To ensure that the limit stress decreases with respect to the off-angle the curves should be defined such that following equations hold:

$$\frac{\partial\sigma^{b}\left(\varphi\right)}{\partial\varphi} \le 0$$

and

$$\sigma^{s}\left(\varepsilon^{vol}\right) - \sigma^{w}\left(\varepsilon^{vol}\right) \ge 0\,.$$

A drawback of this implementation was that the material often collapsed in shear mode due to low shear resistance. There was no way of increasing the shear resistance without changing the behavior in pure uniaxial compression. We have therefore modified the model so that the user can optionally prescribe the shear and hydrostatic resistance in the material without affecting the

uniaxial behavior. We introduce the parameters $\sigma_p^Y(\varepsilon^{\text{vol}})$ and $\sigma_d^Y(\varepsilon^{\text{vol}})$ as the *hydrostatic* and *shear limit stresses*, respectively. These are functions of the volumetric strain and are assumed given by

$$\sigma_p^Y(\varepsilon^{\text{vol}}) = \sigma_p^Y + \sigma^s(\varepsilon^{\text{vol}})$$
$$\sigma_d^Y(\varepsilon^{\text{vol}}) = \sigma_d^Y + \sigma^s(\varepsilon^{\text{vol}}),$$

where we have reused the densification function σ^s. The new parameters are the initial hydrostatic and shear limit stress values, σ_p^Y and σ_d^Y, and are provided by the user as GCAU and |ECCU|, respectively. The negative sign of ECCU flags the third yield surface option whenever LCA<0. The effect of the third formulation is that (i) for a uniaxial stress the stress limit is given by $\sigma^Y(\varphi,\varepsilon^{\text{vol}})$, (ii) for a pressure the stress limit is given by $\sigma_p^Y(\varepsilon^{\text{vol}})$ and (iii) for a simple shear the stress limit is given by $\sigma_d^Y(\varepsilon^{\text{vol}})$. Experiments have shown that the model may give noisy responses and inhomogeneous deformation modes if parameters are not chosen with care. We therefore recommend to (i) avoid large slopes in the function σ^P, (ii) let the functions σ^s and σ^w be slightly increasing and (iii) avoid large differences between the stress limit values $\sigma^y(\varphi,\varepsilon^{\text{vol}})$, $\sigma_p^Y(\varepsilon^{\text{vol}})$ and $\sigma_d^Y(\varepsilon^{\text{vol}})$. These guidelines are likely to contradict how one would interpret test data and it is up to the user to find a reasonable trade-off between matching experimental results and avoiding the mentioned numerical side effects.

For fully compacted material (element formulations 1 and 2), we assume that the material behavior is elastic-perfectly plastic and updated the stress components according to:

$$s_{ij}^{trial} = s_{ij}^n + 2G\Delta\varepsilon_{ij}^{dev^{n+1/2}}$$

where the deviatoric strain increment is defined as

$$\Delta\varepsilon_{ij}^{dev} = \Delta\varepsilon_{ij} - \tfrac{1}{3}\Delta\varepsilon_{kk}\delta_{ij}$$

We now check to see if the yield stress for the fully compacted material is exceeded by comparing

$$s_{eff}^{trial} = \left(\tfrac{3}{2}s_{ij}^{trial}s_{ij}^{trial}\right)^{1/2}$$

the effective trial stress to the yield stress, σ_y (Card 3, field 21-30). If the effective trial stress exceeds the yield stress we simply scale back the stress components to the yield surface

$$s_{ij}^{n+1} = \frac{\sigma_y}{s_{eff}^{trial}} s_{ij}^{trial}$$

We can now update the pressure using the elastic bulk modulus, K

$$p^{n+1} = p^n - K\Delta\varepsilon_{kk}^{n+\frac{1}{2}}$$

$$K = \frac{E}{3(1-2\nu)}$$

and obtain the final value for the Cauchy stress

$$\sigma_{ij}^{n+1} = s_{ij}^{n+1} - p^{n+1}\delta_{ij}$$

After completing the stress update we transform the stresses back to the global configuration.

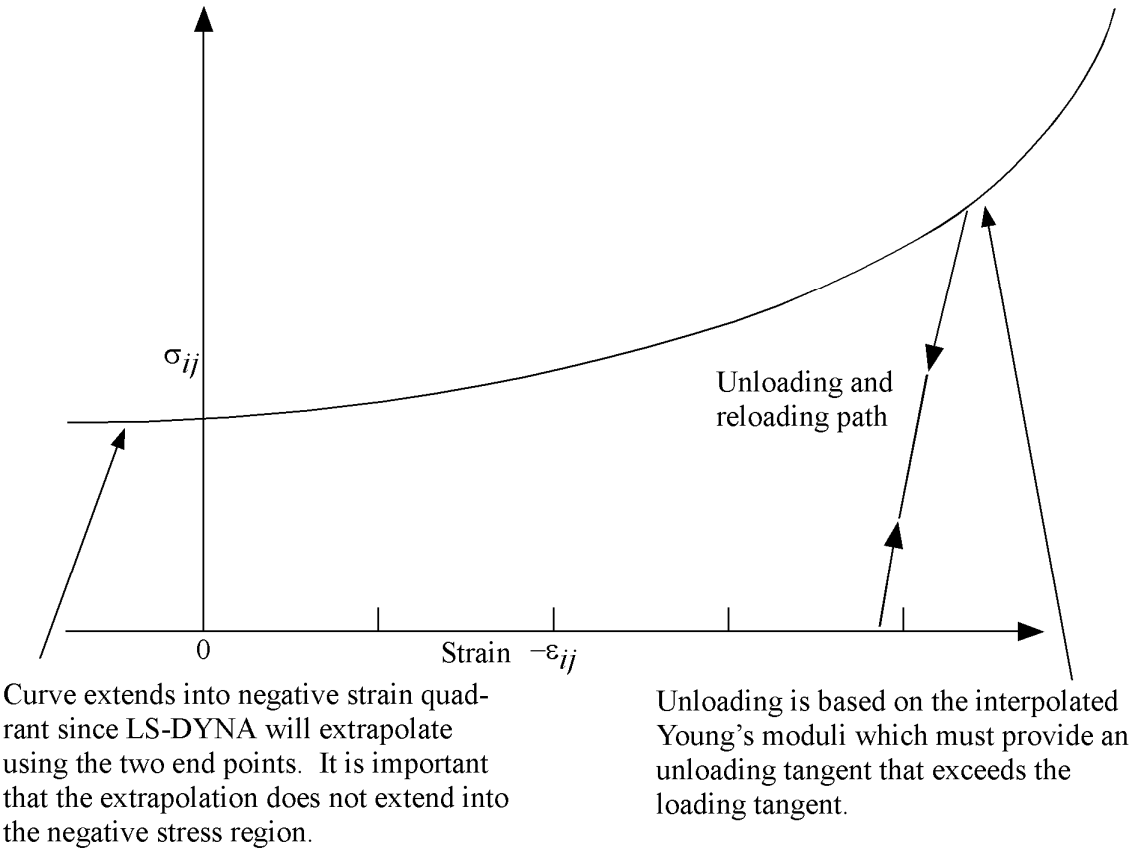


Figure 126.1. Stress quantity versus strain. Note that the "yield stress" at a strain of zero is nonzero. In the load curve definition the "time" value is the directional strain and the "function" value is the yield stress. Note that for element types 0 and 9 engineering strains are used, but for all other element types the rates are integrated in time.

***MAT_ARRUDA_BOYCE_RUBBER**

This is Material Type 127. This material model provides a hyperelastic rubber model, see [Arruda and Boyce 1993] combined optionally with linear viscoelasticity as outlined by [Christensen 1980].

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	K	G	N			
Type	A8	F	F	F	F			

Card 2

Variable	LCID	TRAMP	NT					
Type	F	F	F					

Card Format for Viscoelastic Constants. Up to 6 cards may be input. A keyword card (with a "*" in column 1) terminates this input if less than 6 cards are used.

Optional Cards	1	2	3	4	5	6	7	8
Variable	GI	BETAI						
Type	F	F						

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
K	Bulk modulus
G	Shear modulus
N	Number of statistical links

VARIABLE	DESCRIPTION
LCID	Optional load curve ID of relaxation curve If constants $\beta\iota$ are determined via a least squares fit. This relaxation curve is shown in Figure 76.1. This model ignores the constant stress.
TRAMP	Optional ramp time for loading.
NT	Number of Prony series terms in optional fit. If zero, the default is 6. Currently, the maximum number is 6. Values less than 6, possibly 3-5 are recommended, since each term used adds significantly to the cost. Caution should be exercised when taking the results from the fit. Always check the results of the fit in the output file. Preferably, all generated coefficients should be positive. Negative values may lead to unstable results. Once a satisfactory fit has been achieved it is recommended that the coefficients which are written into the output file be input in future runs.
GI	Optional shear relaxation modulus for the ith term
BETAI	Optional decay constant if ith term

Remarks:

Rubber is generally considered to be fully incompressible since the bulk modulus greatly exceeds the shear modulus in magnitude. To model the rubber as an unconstrained material a hydrostatic work term, $W_H(J)$, is included in the strain energy functional which is function of the relative volume, *J*, [Ogden 1984]:

$$W(J_1,J_2,J) = nk\theta\left[\tfrac{1}{2}(J_1-3)+\tfrac{1}{20N}\left(J_1^2-9\right)+\tfrac{11}{1050N^2}\left(J_1^3-27\right)\right]$$
$$+nk\theta\left[\tfrac{19}{7000N^3}\left(J_1^4-81\right)+\tfrac{519}{673750N^4}\left(J_1^5-243\right)\right]+W_H(J)$$

where the hydrostatic work term is in terms of the bulk modulus, *K*, and the third invariant, *J*, as:

$$W_H(J) = \frac{K}{2}(J-1)^2$$

Rate effects are taken into account through linear viscoelasticity by a convolution integral of the form:

$$\sigma_{ij} = \int_0^t g_{ijkl}(t-\tau)\frac{\partial \varepsilon_{kl}}{\partial \tau}d\tau$$

or in terms of the second Piola-Kirchhoff stress, S_{ij}, and Green's strain tensor, E_{ij},

$$S_{ij} = \int_0^t G_{ijkl}\left(t-\tau\right)\frac{\partial E_{kl}}{\partial \tau}d\tau$$

where $g_{ijkl}\left(t-\tau\right)$ and $G_{ijkl}\left(t-\tau\right)$ are the relaxation functions for the different stress measures. This stress is added to the stress tensor determined from the strain energy functional.

If we wish to include only simple rate effects, the relaxation function is represented by six terms from the Prony series:

$$g\left(t\right) = \alpha_0 + \sum_{m=1}^{N}\alpha_m e^{-\beta t}$$

given by,

$$g\left(t\right) = \sum_{i=1}^{n} G_i e^{-\beta_i t}$$

This model is effectively a Maxwell fluid which consists of a dampers and springs in series. We characterize this in the input by shear moduli, G_i, and decay constants, β_i. The viscoelastic behavior is optional and an arbitrary number of terms may be used.

*MAT_HEART_TISSUE

This is Material Type 128. This material model provides a heart tissue model described in the paper by Walker *et al* [2005] as interpreted by Kay Sun. It is backward compatible with an earlier heart tissue model described in the paper by Guccione, McCulloch, and Waldman [1991]. Both models are transversely isotropic.

Card 1

Variable	MID	RO	C	B1	B2	B3	P	B
Type	A8	F	F	F	F	F	F	F

Card 2 Omit this card for the earlier model.

Variable	L0	CA0MAX	LR	M	BB	CA0	TMAX	TACT
Type	F	I						

Card 3

Variable	AOPT	MACF						
Type	F	I						

Card 4

Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card 5

Variable	V1	V2	V3	D1	D2	D3	BETA	
Type	F	F	F	F	F	F	F	

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
C	Diastolic material coefficient.
B1	b_1 , diastolic material coefficient.
B2	b_2 , diastolic material coefficient.
B3	b_3 , diastolic material coefficient.
P	Pressure in the muscle tissue
B	Systolic material coefficient. Omit for the earlier model.
L0	l_0, sacromere length at which no active tension develops. Omit for the earlier model.
CA0MAX	$(Ca_0)_{max}$, maximum peak intracellular calcium concentrate. Omit for the earlier model.
LR	l_R , Stress-free sacromere length. Omit for the earlier model.
M	Systolic material coefficient. Omit for the earlier model.
BB	Systolic material coefficient. Omit for the earlier model.
CA0	Ca_0 , peak intracellular calcium concentration. Omit for the earlier model.
TMAX	T_{max} , maximum isometric tension achieved at the longest sacromere length. Omit for the earlier model.
TACT	t_{act} , time at which active contraction initiates. Omit for the earlier model.

VARIABLE	DESCRIPTION
AOPT	Material axes option (see MAT_*OPTION* TROPIC_ELASTIC for a more complete description): EQ.0.0: locally orthotropic with material axes determined by element nodes 1, 2, and 4, as with *DEFINE_COORDINATE_NODES. EQ.1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center; this is the a-direction. EQ.2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR. EQ.3.0: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle, BETA, from a line in the plane of the element defined by the cross product of the vector **v** with the element normal. EQ.4.0: locally orthotropic in cylindrical coordinate system with the material axes determined by a vector **v**, and an originating point, P, which define the centerline axis. LT.0.0: the absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_NODES, *DEFINE_COORDINATE_SYSTEM or *DEFINE_COORDINATE_VECTOR). Available in R3 version of 971 and later.
MACF	Material axes change flag for brick elements: EQ.1: No change, default, EQ.2: switch material axes a and b, EQ.3: switch material axes a and c, EQ.4: switch material axes b and c.
XP,YP,ZP	x_p y_p z_p, define coordinates of point **p** for AOPT = 1 and 4.
A1,A2,A3	a_1 a_2 a_3, define components of vector **a** for AOPT = 2.
D1,D2,D3	d_1 d_2 d_3, define components of vector **d** for AOPT = 2.
V1,V2,V3	v_1 v_2 v_3, define components of vector **v** for AOPT = 3 and 4.
BETA	Material angle in degrees for AOPT = 3, may be overridden on the element card, see *ELEMENT_SOLID_ORTHO.

Remarks:

1) The tissue model is described in terms of the energy functional that is transversely isotropic with respect to the local fiber direction,

$$W = \tfrac{C}{2}(e^Q - 1)$$

$$Q = b_f E_{11}^2 + b_t (E_{22}^2 + E_{33}^2 + E_{23}^2 + E_{32}^2) + b_{fs} (E_{12}^2 + E_{21}^2 + E_{13}^2 + E_{31}^2)$$

with C, b_f, b_t, and b_{fs} material parameters and E the Lagranga-Green strains.

The systolic contraction was modeled as the sum of the passive stress derived from the strain energy function and an active fiber directional component, T_0, which is a function of time, t,

$$\underset{\sim}{S} = \frac{\partial W}{\partial \underset{\sim}{E}} - pJ\underset{\sim}{C}^{-1} + T_0\{t, Ca_0, l\}$$

$$\underset{\sim}{\sigma} = \frac{1}{J}\underset{\sim}{F}\underset{\sim}{S}\underset{\sim}{F}^T$$

with $\underset{\sim}{S}$ the second Piola-Kirchoff stress tensor, $\underset{\sim}{C}$ the right Cauchy-Green deformation tensor, J the Jacobian of the deformation gradient tensor $\underset{\sim}{F}$, and $\underset{\sim}{\sigma}$ the Cauchy stress tensor.

The active fiber directional stress component is defined by a time-varying elastance model, which at end-systole, is reduced to

$$T_0 = T_{max} \frac{Ca_0^2}{Ca_0^2 + ECa_{50}^2} C_t$$

with T_{max} the maximum isometric tension achieved at the longest sacromere length and maximum peak intracellular calcium concentration. The length-dependent calcium sensitivity and internal variable is given by,

$$ECa_{50} = \frac{(Ca_0)_{max}}{\sqrt{exp[B(l - l_0] - 1}}$$

$$C_t = 1/2(1 - cos\, w)$$

$$l = l_R \sqrt{2E_{11} + 1}$$

$$w = \pi \frac{0.25 + t_r}{t_r}$$

$$t_r = ml + bb$$

A cross-fiber, in-plane stress equivalent to 40% of that along the myocardial fiber direction is added.

2) The earlier tissue model is described in terms of the energy functional in terms of the Green strain components, E_{ij},

$$W(E) = \tfrac{C}{2}(e^{Q} - 1) + \frac{1}{2}P\left(I_3 - 1\right)$$

$$Q = b_1E_{11}^2 + b_2(E_{22}^2 + E_{33}^2 + E_{23}^2 + E_{32}^2) + b_3(E_{12}^2 + E_{21}^2 + E_{13}^2 + E_{31}^2)$$

The Green components are modified to eliminate any effects of volumetric work following the procedures of Ogden. See the paper by Guccione *et al* [1991] for more detail.

***MAT_LUNG_TISSUE**

This is Material Type 129. This material model provides a hyperelastic model for heart tissue, see [Vawter 1980] combined optionally with linear viscoelasticity as outlined by [Christensen 1980].

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	K	C	DELTA	ALPHA	BETA	
Type	A8	F	F	F	I			

Card 2

Variable	C1	C2	LCID	TRAMP	NT			
Type	F	F	F	F	F			

Card Format for Viscoelastic Constants. Up to 6 cards may be input. A keyword card (with a "*" in column 1) terminates this input if less than 6 cards are used.

Optional Cards	1	2	3	4	5	6	7	8
Variable	GI	BETAI						
Type	F	F						

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
K	Bulk modulus
C	Material coefficient.
DELTA	Δ , material coefficient.

VARIABLE	DESCRIPTION
ALPHA	α, material coefficient.
BETA	β, material coefficient.
C1	Material coefficient.
C2	Material coefficient.
LCID	Optional load curve ID of relaxation curve If constants β_i are determined via a least squares fit. This relaxation curve is shown in Figure 76.1. This model ignores the constant stress.
TRAMP	Optional ramp time for loading.
NT	Number of Prony series terms in optional fit. If zero, the default is 6. Currently, the maximum number is 6. Values less than 6, possibly 3-5 are recommended, since each term used adds significantly to the cost. Caution should be exercised when taking the results from the fit. Always check the results of the fit in the output file. Preferably, all generated coefficients should be positive. Negative values may lead to unstable results. Once a satisfactory fit has been achieved it is recommended that the coefficients which are written into the output file be input in future runs.
GI	Optional shear relaxation modulus for the ith term
BETAI	Optional decay constant if ith term

Remarks:

The material is described by a strain energy functional expressed in terms of the invariants of the Green Strain:

$$W(I_1, I_2) = \frac{C}{2\Delta} e^{(\alpha I_1^2 + \beta I_2)} + \frac{12C_1}{\Delta(1+C_2)}\left[A^{(1+C_2)} - 1\right]$$

$$A^2 = \tfrac{4}{3}\left(I_1 + I_2\right) - 1$$

where the hydrostatic work term is in terms of the bulk modulus, *K*, and the third invariant, *J*, as:

$$W_H\left(J\right) = \frac{K}{2}(J-1)^2$$

Rate effects are taken into account through linear viscoelasticity by a convolution integral of the form:

$$\sigma_{ij} = \int_0^t g_{ijkl}\left(t-\tau\right)\frac{\partial \varepsilon_{kl}}{\partial \tau}d\tau$$

or in terms of the second Piola-Kirchhoff stress, S_{ij}, and Green's strain tensor, E_{ij},

$$S_{ij} = \int_0^t G_{ijkl}\left(t-\tau\right)\frac{\partial E_{kl}}{\partial \tau}d\tau$$

where $g_{ijkl}(t-\tau)$ and $G_{ijkl}(t-\tau)$ are the relaxation functions for the different stress measures. This stress is added to the stress tensor determined from the strain energy functional.

If we wish to include only simple rate effects, the relaxation function is represented by six terms from the Prony series:

$$g(t) = \alpha_0 + \sum_{m=1}^{N} \alpha_m e^{-\beta t}$$

given by,

$$g(t) = \sum_{i=1}^{n} G_i e^{-\beta_i t}$$

This model is effectively a Maxwell fluid which consists of a dampers and springs in series. We characterize this in the input by shear moduli, G_i, and decay constants, β_i. The viscoelastic behavior is optional and an arbitrary number of terms may be used.

*MAT_SPECIAL_ORTHOTROPIC

This is Material Type 130. This model is available the Belytschko-Tsay and the C0 triangular shell elements and is based on a resultant stress formulation. In-plane behavior is treated separately from bending in order to model perforated materials such as television shadow masks. If other shell formulations are specified, the formulation will be automatically switched to Belytschko-Tsay. As implemented, this material model cannot be used with user defined integration rules.

Card 1

1	2	3	4	5	6	7	8
MID	RO	YS	EP				
A8	F	F	F				

Card 2

E11P	E22P	V12P	V21P	G12P	G23P	G31P	
F	F	F	F	F	F	F	

Card 3

E11B	E22B	V12B	V21B	G12B	AOPT		
F	F	F	F	F	F		

Card 4

			A1	A2	A3		
			F	F	F		

Card 5	1	2	3	4	5	6	7	8
Variable	V1	V2	V3	D1	D2	D3	BETA	
Type	F	F	F	F	F	F	F	

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
YS	Yield stress. This parameter is optional and is approximates the yield condition. Set to zero if the behavior is elastic.
EP	Plastic hardening modulus.
E11P	E_{11p}, for in plane behavior.
E22P	E_{22p}, for in plane behavior.
V12P	ν_{12p}, for in plane behavior.
V11P	ν_{21p}, for in plane behavior.
G12P	G_{12p}, for in plane behavior.
G23P	G_{23p}, for in plane behavior.
G31P	G_{31p}, for in plane behavior.
E11B	E_{11b}, for bending behavior.
E22B	E_{22b}, for bending behavior.
V12B	ν_{12b}, for bending behavior.
V21B	ν_{21b}, for bending behavior.
G12B	G_{12b}, for bending behavior.

VARIABLE	DESCRIPTION
AOPT	Material axes option (see MAT_*OPTION* TROPIC_ELASTIC for a more complete description): EQ.0.0: locally orthotropic with material axes determined by element nodes 1, 2, and 4, as with *DEFINE_COORDINATE_NODES. EQ.2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR. EQ.3.0: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle, BETA, from a line in the plane of the element defined by the cross product of the vector **v** with the element normal. LT.0.0: the absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_NODES, *DEFINE_COORDINATE_SYSTEM or *DEFINE_COORDINATE_VECTOR). Available in R3 version of 971 and later.
A1,A2,A3	a_1 a_2 a_3, define components of vector **a** for AOPT = 2.
D1,D2,D3	d_1 d_2 d_3, define components of vector **d** for AOPT = 2.
V1,V2,V3	v_1 v_2 v_3, define components of vector **v** for AOPT = 3.
BETA	Material angle in degrees for AOPT = 3, may be overridden on the element card, see *ELEMENT_SHELL_BETA.

Remarks:

The in-plane elastic matrix for in-plane, plane stress behavior is given by:

$$C_{in\,plane} = \begin{bmatrix} Q_{11p} & Q_{12p} & 0 & 0 & 0 \\ Q_{12p} & Q_{22p} & 0 & 0 & 0 \\ 0 & 0 & Q_{44p} & 0 & 0 \\ 0 & 0 & 0 & Q_{55p} & 0 \\ 0 & 0 & 0 & 0 & Q_{66p} \end{bmatrix}$$

The terms Q_{ijp} are defined as:

$$Q_{11p} = \frac{E_{11p}}{1-\nu_{12p}\nu_{21p}}$$

$$Q_{22p} = \frac{E_{22p}}{1-\nu_{12p}\nu_{21p}}$$

$$Q_{12p} = \frac{\nu_{12p}E_{11p}}{1-\nu_{12p}\nu_{21p}}$$

$$Q_{44p} = G_{12p}$$

$$Q_{55p} = G_{23p}$$

$$Q_{66p} = G_{31p}$$

The elastic matrix for bending behavior is given by:

$$C_{bending} = \begin{bmatrix} Q_{11b} & Q_{12b} & 0 \\ Q_{12b} & Q_{22b} & 0 \\ 0 & 0 & Q_{44b} \end{bmatrix}$$

The terms Q_{ijp} are similarly defined.

*MAT_ISOTROPIC_SMEARED_CRACK

This is Material Type 131. This model was developed by Lemmen and Meijer [2001] as a smeared crack model for isotropic materials. This model is available of solid elements only and is restricted to cracks in the x-y plane. Users should choose other models unless they have the report by Lemmen and Meijer [2001].

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	ISPL	SIGF	GK	SR
Type	A8	F	F	F	I	F	F	F

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
E	Young's modulus
PR	Poisson's ratio
ISPL	Failure option: EQ.0: Maximum principal stress criterion EQ.5: Smeared crack model EQ.6: Damage model based on modified von Mises strain.
SIGF	Peak stress.
GK	Critical energy release rate.
SR	Strength ratio.

Remarks:

The following documentation is taken nearly verbatim from the documentation of Lemmen and Meijer [2001].

Three methods are offered to model progressive failure. The maximum principal stress criterion detects failure if the maximum (most tensile) principal stress exceeds σ_{max}. Upon failure, the material can no longer carry stress.

The second failure model is the smeared crack model with linear softening stress-strain using equivalent uniaxial strains. Failure is assumed to be perpendicular to the principal strain directions. A rotational crack concept is employed in which the crack directions are related to the current directions of principal strain. Therefore crack directions may rotate in time. Principal stresses are expressed as

$$\begin{pmatrix}\sigma_1\\ \sigma_2\\ \sigma_3\end{pmatrix}=\begin{bmatrix}\bar{\mathrm{E}}_1 & 0 & 0\\ 0 & \bar{\mathrm{E}}_2 & 0\\ 0 & 0 & \bar{\mathrm{E}}_3\end{bmatrix}\begin{pmatrix}\tilde{\varepsilon}_1\\ \tilde{\varepsilon}_2\\ \tilde{\varepsilon}_3\end{pmatrix}=\begin{pmatrix}\bar{\mathrm{E}}_1\tilde{\varepsilon}_1\\ \bar{\mathrm{E}}_2\tilde{\varepsilon}_2\\ \bar{\mathrm{E}}_3\tilde{\varepsilon}_3\end{pmatrix} \tag{131.1}$$

with $\bar{\mathrm{E}}_1$, $\bar{\mathrm{E}}_2$ and $\bar{\mathrm{E}}_3$ secant stiffness in the terms that depend on internal variables.

In the model developed for DYCOSS it has been assumed that there is no interaction between the three directions in which case stresses simply follow from

$$\sigma_j(\tilde{\varepsilon}_j)=\begin{cases}\mathrm{E}\tilde{\varepsilon}_j & \text{if} \quad 0\le\tilde{\varepsilon}_j\le\tilde{\varepsilon}_{j,ini}\\ \bar{\sigma}\left(1-\dfrac{\tilde{\varepsilon}_j-\tilde{\varepsilon}_{j,ini}}{\tilde{\varepsilon}_{j,ult}-\tilde{\varepsilon}_{j,ini}}\right) & \text{if} \quad \tilde{\varepsilon}_{j,ini}<\tilde{\varepsilon}_j\le\tilde{\varepsilon}_{j,ult}\\ 0 & \text{if} \quad \tilde{\varepsilon}_j>\tilde{\varepsilon}_{j,ult}\end{cases} \tag{131.2}$$

with $\bar{\sigma}$ the ultimate stress, $\tilde{\varepsilon}_{j,ini}$ the damage threshold, and $\tilde{\varepsilon}_{j,ult}$ the ultimate strain in *j*-direction. The damage threshold is defined as

$$\tilde{\varepsilon}_{j,ini}=\frac{\bar{\sigma}}{\mathrm{E}} \tag{131.3}$$

The ultimate strain is obtained by relating the crack growth energy and the dissipated energy

$$\int\int\bar{\sigma}d\tilde{\varepsilon}_{j,ult}dV=GA \tag{131.4}$$

with *G* the energy release rate, *V* the element volume and *A* the area perpendicular to the principal strain direction. The one point elements LS-DYNA have a single integration point and the integral over the volume may be replaced by the volume. For linear softening it follows

$$\tilde{\varepsilon}_{j,\,ult}=\frac{2GA}{V\bar{\sigma}} \tag{131.5}$$

The above formulation may be regarded as a damage equivalent to the maximum principle stress criterion.

The third model is a damage model represented by Brekelmans et. al [1991]. Here the Cauchy stress tensor σ is expressed as

$$\sigma = (1-D)\mathrm{E}\varepsilon \tag{131.6}$$

where D represents the current damage and the factor (1-D) is the reduction factor caused by damage. The scalar damage variable is expressed as function of a so-called damage equivalent strain ε_d

$$D = D(\varepsilon_d) = 1 - \frac{\varepsilon_{ini}(\varepsilon_{ult} - \varepsilon_d)}{\varepsilon_d(\varepsilon_{ult} - \varepsilon_{ini})} \tag{131.7}$$

and

$$\varepsilon_d = \frac{k-1}{2k(1-2v)} J_1 + \frac{1}{2k}\sqrt{\left(\frac{k-1}{1-2v} J_1\right)^2 + \frac{6k}{(1+v)^2} J_2} \tag{131.8}$$

where the constant k represents the ratio of the strength in tension over the strength in compression

$$k = \frac{\sigma_{ult,\,tension}}{\sigma_{ult,\,compression}} \tag{131.9}$$

J_1 resp. J_2 are the first and second invariant of the strain tensor representing the volumetric and the deviatoric straining respectively

$$\begin{aligned} J_1 &= tr(\varepsilon) \\ J_2 &= tr(\varepsilon \cdot \varepsilon) - \frac{1}{3} tr^2(\varepsilon) \end{aligned} \tag{131.10}$$

If the compression and tension strength are equal the dependency on the volumetric strain vanishes in (8) and failure is shear dominated. If the compressive strength is much larger than the strength in tension, k becomes small and the J_1 terms in (131.8) dominate the behavior.

*MAT_ORTHOTROPIC_SMEARED_CRACK

This is Material Type 132. This material is a smeared crack model for orthotropic materials.

Card 1

1	2	3	4	5	6	7	8
MID	RO	EA	EB	EC	PRBA	PRCA	PRCB
A8	F	F	F	F	F	F	F

Card 2

UINS	UISS	CERRMI	CERRMII	IND	ISD		
F	F	F	F	I	I		

Card 3

GAB	GBC	GCA	AOPT				
F	F	F	F				

Card 4

XP	YP	ZP	A1	A2	A3	MACF	
F	F	F	F	F	F	I	

Card 5

V1	V2	V3	D1	D2	D3	BETA	REF
F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
EA	E_a, Young's modulus in *a*-direction.
EB	E_b, Young's modulus in *b*-direction.
EC	E_c, Young's modulus in *c*-direction
PRBA	ν_{ba}, Poisson's ratio *ba.*
PRCA	ν_{ca}, Poisson's ratio *ca*.
PRCB	ν_{cb}, Poisson's ratio *cb*.
UINS	Ultimate interlaminar normal stress.
UISS	Ultimate interlaminar shear stress.
CERRMI	Critical energy release rate mode I
CERRMII	Critical energy release rate mode II
IND	Interlaminar normal direction : EQ.1.0: Along local *a* axis EQ.2.0: Along local *b* axis EQ.3.0: Along local *c* axis
ISD	Interlaminar shear direction : EQ.4.0: Along local *ab* axis EQ.5.0: Along local *bc* axis EQ.6.0: Along local *ca* axis
GAB	G_{ab}, shear modulus ab.
GBC	G_{bc}, shear modulus bc.
GCA	G_{ca}, shear modulus ca.
AOPT	Material axes option, see Figure 2.1. EQ.0.0: locally orthotropic with material axes determined by element nodes as shown in Figure 2.1. Nodes 1, 2, and 4 of an element are identical to the nodes used for the definition of a coordinate system as by *DEFINE_COORDINATE_NODES.

VARIABLE	DESCRIPTION
	EQ.1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center; this is the a-direction. This option is for solid elements only. EQ.2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR. EQ.3.0: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle, BETA, from a line in the plane of the element defined by the cross product of the vector **v** with the element normal. The plane of a solid element is the midsurface between the inner surface and outer surface defined by the first four nodes and the last four nodes of the connectivity of the element, respectively. EQ.4.0: locally orthotropic in cylindrical coordinate system with the material axes determined by a vector **v**, and an originating point, P, which define the centerline axis. This option is for solid elements only. LT.0.0: the absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_NODES, *DEFINE_COORDINATE_SYSTEM or *DEFINE_COORDINATE_VECTOR). Available in R3 version of 971 and later.
XP YP ZP	Define coordinates of point **p** for AOPT = 1 and 4.
A1 A2 A3	Define components of vector **a** for AOPT = 2.
MACF	Material axes change flag for brick elements: EQ.1: No change, default, EQ.2: switch material axes a and b, EQ.3: switch material axes a and c, EQ.4: switch material axes b and c.
V1 V2 V3	Define components of vector **v** for AOPT = 3 and 4.
D1 D2 D3	Define components of vector **d** for AOPT = 2:
BETA	Material angle in degrees for AOPT = 3, may be overridden on the element card, see *ELEMENT_SHELL_BETA or *ELEMENT_SOLID_ORTHO.
REF	Use reference geometry to initialize the stress tensor. The reference geometry is defined by the keyword: *INITIAL_FOAM_REFERENCE_GEOMETRY (see there for more details). EQ.0.0: off, EQ.1.0: on.

Remarks:

This is an orthotropic material with optional delamination failure for brittle composites. The elastic formulation is identical to the DYNA3D model that uses total strain formulation. The constitutive matrix **C** that relates to global components of stress to the global components of strain is defined as:

$$C = T^T C_L T$$

where T is the transformation matrix between the local material coordinate system and the global system and C_L is the constitutive matrix defined in terms of the material constants of the local orthogonal material axes *a*, *b*, and *c* (see DYNA3D use manual).

Failure is described using linear softening stress strain curves for interlaminar normal and interlaminar shear direction. The current implementation for failure is essentially 2-D. Damage can occur in interlaminar normal direction and a single interlaminar shear direction. The orientation of these directions w.r.t. the principal material directions have to be specified by the user.

Based on specified values for the ultimate stress and the critical energy release rate bounding surfaces are defined

$$f_n = \sigma_n - \bar{\sigma}_n(\varepsilon_n)$$
$$f_s = \sigma_s - \bar{\sigma}_s(\varepsilon_s)$$

where the subscripts n and s refer to the normal and shear component. If stresses exceed the bounding surfaces inelastic straining occurs. The ultimate strain is obtained by relating the crack growth energy and the dissipated energy. For solid elements with a single integration point it can be derived

$$\varepsilon_{i,\,ult} = \frac{2G_i A}{V\sigma_{i,\,ult}}$$

with G_i the critical energy release rate, V the element volume, A the area perpendicular to the active normal direction and $\sigma_{i,\,ult}$ the ultimate stress. For the normal component failure can only occur under tensile loading. For shear component the behavior is symmetric around zero. The resulting stress bounds are depicted in Figure 132.1. Unloading is modeled with a Secant stiffness.

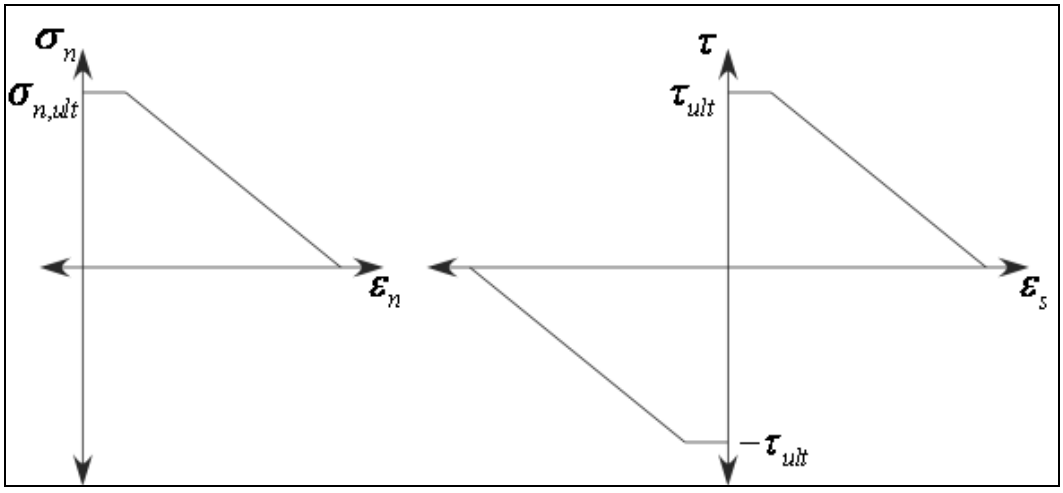


Figure 132.1. Shows stress bounds for the active normal component (left) and the archive shear component (right).

*MAT_BARLAT_YLD2000

This is Material Type 133. This model was developed by Barlat et al. [2003] to overcome some shortcomings of the six parameter Barlat model implemented as material 33 (MAT_ BARLAT_YLD96) in LS-DYNA. This model is available for shell elements only.

Card 1

1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	FIT	BETA	ITER	
Type	A8	F	F	F	F	F	F	

Card 2

Variable	K	E0	N	C	P	HARD	A	
Type	F	F	F	F	F	F	F	

Define the following card if and only if A<0

Card 3

1	2	3	4	5	6	7	8
Variable	CRC1	CRA1	CRC2	CRA2	CRC3	CRA3	CRC4	CRA4
Type	F	F	F	F	F	F	F	F

Define the following card if and only if FIT=0

Card 3

1	2	3	4	5	6	7	8
Variable	ALPHA1	ALPHA2	ALPHA3	ALPHA4	ALPHA5	ALPHA6	ALPHA7	ALPHA8
Type	F	F	F	F	F	F	F	F

Define the following two cards if and only if FIT=1

Card 3

Variable	SIG00	SIG45	SIG90	R00	R45	R90		
Type	F	F	F	F	F	F		

Card 4

Variable	SIGXX	SIGYY	SIGXY	DXX	DYY	DXY		
Type	F	F	F	F	F	F		

Card 4/5

Variable	AOPT	OFFANG						
Type	F	F						

Card 5/6

Variable				A1	A2	A3		
Type				F	F	F		

Card 6/7

Variable	V1	V2	V3	D1	D2	D3		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.

RO	Mass density
E	Young’s modulus
PR	Poisson’s ratio

VARIABLE	DESCRIPTION
FIT	Material parameter fit flag: EQ.0.0: Material parameters are used directly on card 3. EQ.1.0: Material parameters are determined from test data on cards 3 and 4.
BETA	Hardening parameter. Any value ranging from 0 (isotropic hardening) to 1 (kinematic hardening) may be input.
ITER	Plastic iteration flag: EQ.0.0: Plane stress algorithm for stress return EQ.1.0: Secant iteration algorithm for stress return ITER provides an option of using three secant iterations for determining the thickness strain increment as experiments have shown that this leads to a more accurate prediction of shell thickness changes for rapid processes. A significant increase in computation time is incurred with this option so it should be used only for applications associated with high rates of loading and/or for implicit analysis.
K	Material parameter: HARD.EQ.1.0: k, strength coefficient for exponential hardening HARD.EQ.2.0: a in Voce hardening law
E0	Material parameter: HARD.EQ.1.0: ε_0, strain at yield for exponential hardening HARD.EQ.2.0: b in Voce hardening law
N	Material parameter: HARD.EQ.1.0: n, exponent for exponential hardening HARD.EQ.2.0: c in Voce hardening law
C	Cowper-Symonds strain rate parameter, C, see formula below.
P	Cowper-Symonds strain rate parameter, p. $$\sigma_y^v(\varepsilon_p,\dot{\varepsilon}_p)=\sigma_y(\varepsilon_p)\left(1+\left\{\frac{\dot{\varepsilon}_p}{C}\right\}^{1/p}\right)$$
HARD	Hardening law: EQ.1.0: Exponential hardening: $\sigma_y = k\left(\varepsilon_0+\varepsilon_p\right)^n$ EQ.2.0: Voce hardening: $\sigma_y = a - be^{-c\varepsilon^p}$ LT.0.0: Absolute value defines load curve ID
A	Flow potential exponent

VARIABLE	DESCRIPTION
CRCN	Chaboche-Roussilier kinematic hardening parameter, see remarks.
CRCA	Chaboche-Roussilier kinematic hardening parameter, see remarks.
ALPHA1	α_1, see equations below
ALPHA2	α_2, see equations below
ALPHA3	α_3, see equations below
ALPHA4	α_4, see equations below
ALPHA5	α_5, see equations below
ALPHA6	α_6, see equations below
ALPHA7	α_7, see equations below
ALPHA8	α_8, see equations below
SIG00	Yield stress in 00 direction
SIG45	Yield stress in 45 direction
SIG90	Yield stress in 90 direction
R00	R-value in 00 direction
R45	R-value in 45 direction
R90	R-value in 90 direction
SIGXX	xx-component of stress on yield surface (See Remark 2).
SIGYY	yy-component of stress on yield surface (See Remark 2).
SIGXY	xy-component of stress on yield surface (See Remark 2).
DXX	xx-component of tangent to yield surface (See Remark 2).
DYY	yy-component of tangent to yield surface (See Remark 2).
DXY	xy-component of tangent to yield surface (See Remark 2).
AOPT	Material axes option: EQ.0.0: locally orthotropic with material axes determined by element nodes as shown in Figure 20.1. Nodes 1, 2, and 4 of an

element are identical to the nodes used for the definition of a coordinate system as by *DEFINE_COORDINATE_NODES

VARIABLE	DESCRIPTION
	EQ.2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR EQ.3.0: locally orthotropic material axes determined by offsetting the material axes by an angle, OFFANG, from a line determined by taking the cross product of the vector **v** with the normal to the plane of the element. LT.0.0: the absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_NODES, *DEFINE_COORDINATE_SYSTEM or *DEFINE_COORDINATE_VECTOR). Available in R3 version of 971 and later.
OFFANG	Offset angle for AOPT=3
A1 A2 A3	Components of vector **a** for AOPT=2
V1 V2 V3	Components of vector **v** for AOPT=3
D1 D2 D3	Components of vector **d** for AOPT=2

Remarks:

1. Strain rate is accounted for using the Cowper and Symonds model which scales the yield stress with the factor

$$1+\left(\frac{\dot{\varepsilon}}{C}\right)^{1/p}$$

where $\dot{\varepsilon}$ is the strain rate. A fully viscoplastic formulation is optional which incorporates the Cowper and Symonds formulation within the yield surface. An additional cost is incurred but the improvement allows for dramatic results. To ignore strain rate effects set both SRC and SRP to zero.

2. The yield condition for this material can be written

$$f(\boldsymbol{\sigma},\boldsymbol{\alpha},\varepsilon_p)=\sigma_{\text{eff}}(\sigma_{xx}-2\alpha_{xx}-\alpha_{yy},\sigma_{yy}-2\alpha_{yy}-\alpha_{xx},\sigma_{xy}-\alpha_{xy})-\sigma_Y^t(\varepsilon_p,\dot{\varepsilon}_p,\beta)\le 0$$

where

$$\sigma_{\text{eff}}(s_{xx},s_{yy},s_{xy})=\left(\frac{1}{2}(\phi'+\phi'')\right)^{1/a}$$

$$\phi'=\left|X'_1-X'_2\right|^a$$

$$\phi''=\left|2X''_1+X''_2\right|^a+\left|X''_1+2X''_2\right|^a$$

The X'_i and X''_i are eigenvalues of X'_{ij} and X''_{ij} and are given by

$$X'_1 = \frac{1}{2}\left(X'_{11} + X'_{22} + \sqrt{(X'_{11} - X'_{22})^2 + 4X'^2_{12}} \right)$$

$$X'_2 = \frac{1}{2}\left(X'_{11} + X'_{22} - \sqrt{(X'_{11} - X'_{22})^2 + 4X'^2_{12}} \right)$$

and

$$X''_1 = \frac{1}{2}\left(X''_{11} + X''_{22} + \sqrt{(X''_{11} - X''_{22})^2 + 4X''^2_{12}} \right)$$

$$X''_2 = \frac{1}{2}\left(X''_{11} + X''_{22} - \sqrt{(X''_{11} - X''_{22})^2 + 4X''^2_{12}} \right)$$

respectively. The X'_{ij} and X''_{ij} are given by

$$\begin{pmatrix} X'_{11} \\ X'_{22} \\ X'_{12} \end{pmatrix} = \begin{pmatrix} L'_{11} & L'_{12} & 0 \\ L'_{21} & L'_{22} & 0 \\ 0 & 0 & L'_{33} \end{pmatrix} \begin{pmatrix} s_{xx} \\ s_{yy} \\ s_{xy} \end{pmatrix} \qquad \begin{pmatrix} X''_{11} \\ X''_{22} \\ X''_{12} \end{pmatrix} = \begin{pmatrix} L''_{11} & L''_{12} & 0 \\ L''_{21} & L''_{22} & 0 \\ 0 & 0 & L''_{33} \end{pmatrix} \begin{pmatrix} s_{xx} \\ s_{yy} \\ s_{xy} \end{pmatrix}$$

where

$$\begin{pmatrix} L'_{11} \\ L'_{12} \\ L'_{21} \\ L'_{22} \\ L'_{33} \end{pmatrix} = \frac{1}{3} \begin{pmatrix} 2 & 0 & 0 \\ -1 & 0 & 0 \\ 0 & -1 & 0 \\ 0 & 2 & 0 \\ 0 & 0 & 3 \end{pmatrix} \begin{pmatrix} \alpha_1 \\ \alpha_2 \\ \alpha_7 \end{pmatrix} \qquad \begin{pmatrix} L''_{11} \\ L''_{12} \\ L''_{21} \\ L''_{22} \\ L''_{33} \end{pmatrix} = \frac{1}{9} \begin{pmatrix} -2 & 2 & 8 & -2 & 0 \\ 1 & -4 & -4 & 4 & 0 \\ 4 & -4 & -4 & 1 & 0 \\ -2 & 8 & 2 & -2 & 0 \\ 0 & 0 & 0 & 0 & 9 \end{pmatrix} \begin{pmatrix} \alpha_3 \\ \alpha_4 \\ \alpha_5 \\ \alpha_6 \\ \alpha_8 \end{pmatrix}$$

The parameters α_1 to α_8 are the parameters that determines the shape of the yield surface.

The material parameters can be determined from three uniaxial tests and a more general test. From the uniaxial tests the yield stress and R-values are used and from the general test an arbitrary point on the yield surface is used given by the stress components in the material system as

$$\boldsymbol{\sigma} = \begin{pmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{xy} \end{pmatrix}$$

together with a tangent of the yield surface in that particular point. For the latter the tangential direction should be determined so that

$$d_{xx}\varepsilon^p_{xx} + d_{yy}\varepsilon^p_{yy} + 2d_{xy}\varepsilon^p_{xy} = 0$$

The biaxial data can be set to zero in the input deck for LS-DYNA to just fit the uniaxial data.

3. A kinematic hardening model is implemented following the works of Chaboche and Roussilier. A back stress $\boldsymbol{\alpha}$ is introduced such that the effective stress is computed as

$$\sigma_{\text{eff}} = \sigma_{\text{eff}}(\sigma_{11} - 2\alpha_{11} - \alpha_{22}, \sigma_{22} - 2\alpha_{22} - \alpha_{11}, \sigma_{12} - \alpha_{12})$$

The back stress is the sum of up to four terms according to

$$\alpha_{ij} = \sum_{k=1}^{4} \alpha_{ij}^{k}$$

and the evolution of each back stress component is as follows

$$\delta\alpha_{ij}^{k} = C_k \left(a_k \frac{s_{ij}}{\sigma_{\text{eff}}} - \alpha_{ij}^{k} \right) \delta\varepsilon_p$$

where C_k and a_k are material parameters, s_{ij} is the deviatoric stress tensor, σ_{eff} is the effective stress and ε_p is the effective plastic strain.

***MAT_ VISCOELASTIC_FABRIC**

This is Material Type 134. The viscoelastic fabric model is a variation on the general viscoelastic model of material 76. This model is valid for 3 and 4 node membrane elements only and is strongly recommended for modeling isotropic viscoelastic fabrics where wrinkling may be a problem. For thin fabrics, buckling can result in an inability to support compressive stresses; thus, a flag is included for this option. If bending stresses are important use a shell formulation with model 76.

Card 1

1	2	3	4	5	6	7	8
Variable	MID	RO	BULK	(omit)	(omit)	(omit)	CSE	
Type	I	F	F				F	

Insert a blank card here if constants are defined on cards 3,4,... below.

If fitting is done from a relaxation curve, specify fitting parameters on card 2.

Card 2

1	2	3	4	5	6	7	8
Variable	LCID	NT	BSTART	TRAMP	LCIDK	NTK	BSTARTK	TRAMPK
Type	F	I	F	F	F	I	F	F

Card Format for viscoelastic constants. Up to 6 cards may be input. A keyword card (with a "*" in column 1) terminates this input if less than 6 cards are used. These cards are not needed if relaxation data is defined. The number of terms for the shear behavior may differ from that for the bulk behavior: simply insert zero if a term is not included.

Optional Cards 3, . .	1	2	3	4	5	6	7	8
Variable	GI	BETAI	KI	BETAKI				
Type	F	F	F	F				

VARIABLE	DESCRIPTION
MID	Material identification. A unique number must be specified.
RO	Mass density.
BULK	Elastic constant bulk modulus. If the bulk behavior is viscoelastic, then this modulus is used in determining the contact interface stiffness only.
CSE	Compressive stress flag (default = 0.0). EQ.0.0: don't eliminate compressive stresses EQ.1.0: eliminate compressive stresses
LCID	Load curve ID if constants, G_i, and β_i are determined via a least squares fit. This relaxation curve is shown below.
NT	Number of terms in shear fit. If zero the default is 6. Currently, the maximum number is set to 6.
BSTART	In the fit, β_1 is set to zero, β_2 is set to BSTART, β_3 is 10 times β_2, β_4 is 100 times greater than β_3, and so on. If zero, BSTART=0.01.
TRAMP	Optional ramp time for loading.
LCIDK	Load curve ID for bulk behavior if constants, K_i, and $\beta\kappa_i$ are determined via a least squares fit. This relaxation curve is shown below.
NTK	Number of terms desired in bulk fit. If zero the default is 6. Currently, the maximum number is set to 6.

VARIABLE	DESCRIPTION
BSTARTK	In the fit, $\beta\kappa_1$ is set to zero, $\beta\kappa_2$ is set to BSTARTK, $\beta\kappa_3$ is 10 times $\beta\kappa_2$, $\beta\kappa_4$ is 100 times greater than $\beta\kappa_3$, and so on. If zero, BSTARTK=0.01.
TRAMPK	Optional ramp time for bulk loading.
GI	Optional shear relaxation modulus for the ith term
BETAI	Optional shear decay constant for the ith term
KI	Optional bulk relaxation modulus for the ith term
BETAKI	Optional bulk decay constant for the ith term

Remarks:

Rate effects are taken into accounted through linear viscoelasticity by a convolution integral of the form:

$$\sigma_{ij} = \int_0^t g_{ijkl}(t-\tau)\frac{\partial \varepsilon_{kl}}{\partial \tau} d\tau$$

where $g_{ijkl}(t-\tau)$ is the relaxation function.

If we wish to include only simple rate effects for the deviatoric stresses, the relaxation function is represented by six terms from the Prony series:

$$g(t) = \sum_{m=1}^{N} G_m e^{-\beta_m t}$$

We characterize this in the input by shear modulii, G_i, and decay constants, β_i. An arbitrary number of terms, up to 6, may be used when applying the viscoelastic model.

For volumetric relaxation, the relaxation function is also represented by the Prony series in terms of bulk modulii:

$$k(t) = \sum_{m=1}^{N} K_m e^{-\beta_{k_m} t}$$

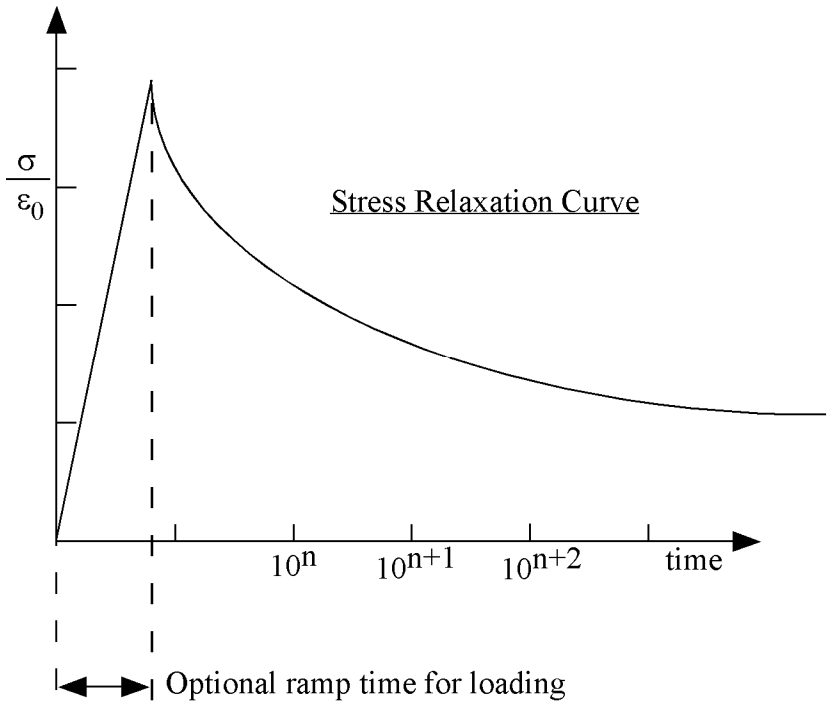


Figure 134.1. Relaxation curve. This curve defines stress versus time where time is defined on a logarithmic scale. For best results, the points defined in the load curve should be equally spaced on the logarithmic scale. Furthermore, the load curve should be smooth and defined in the positive quadrant. If nonphysical values are determined by least squares fit, LS-DYNA will terminate with an error message after the initialization phase is completed. If the ramp time for loading is included, then the relaxation which occurs during the loading phase is taken into account. This effect may or may not be important.

***MAT_WTM_STM**

This is material type 135. This anisotropic-viscoplastic material model adopts two yield criteria for metals with orthotropic anisotropy proposed by Barlat and Lian [1989] (Weak Texture Model) and Aretz [2004] (Strong Texture Model).

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	NUMFI	EPSC	WC	TAUC
Type	A8	F	F	F	F	F	F	F

Card 2

Variable	SIGMA0	QR1	CR1	QR2	CR2	K	LC	FLG
Type	F	F	F	F	F	F	F	F

Describe the following card for FLG = 0

Card 3	1	2	3	4	5	6	7	8
Variable	A1	A2	A3	A4	A5	A6	A7	A8
Type	F	F	F	F	F	F	F	F

Describe the following card for FLG = 1

Card 3	1	2	3	4	5	6	7	8
Variable	S00	S45	S90	SBB	R00	R45	R90	RBB
Type	F	F	F	F	F	F	F	F

Describe the following card for FLG = 2

Card 3

	1	2	3	4	5	6	7	8
Variable	A	C	H	P				
Type	F	F	F	F				

Card 4

Variable	QX1	CX1	QX2	CX2	EDOT	M	EMIN	S100
Type	F	F	F	F	F	F	F	F

Card 5

Variable	AOPT	BETA						
Type	F	F						

Card 6

Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card 7

Variable	V1	V2	V3	D1	D2	D3		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
E	Young's modulus
PR	Poisson's ratio
NUMFI	Number of through thickness integration points that must fail before the element is deleted (remember to change this number if switching between full and reduced integration type of elements).
EPSC	Critical value ε_{tC} of the plastic thickness strain (used in the CTS fracture criterion).
WC	Critical value W_c for the Cockcroft-Latham fracture criterion
TAUC	Critical value τ_c for the Bressan-Williams shear fracture criterion
SIGMA0	Initial mean value of yield stress σ_0
QR1	Isotropic hardening parameter Q_{R1}
CR1	Isotropic hardening parameter C_{R1}
QR2	Isotropic hardening parameter Q_{R2}
CR2	Isotropic hardening parameter C_{R2}
K	k equals half YLD2003 exponent m. Recommended value for FCC materials is $m = 8$, i.e. $k = 4$.
LC	First load curve number for process effects, i.e. the load curve describing the relation between the pre-strain and the yield stress σ_0. Similar curves for Q_{R1}, C_{R1}, Q_{R2}, C_{R2}, and W_c must follow consecutively from this number.
A1	Yld2003 parameter a_1
A2	Yld2003 parameter a_2

VARIABLE	DESCRIPTION
A3	Yld2003 parameter a_3
A4	Yld2003 parameter a_4
A5	Yld2003 parameter a_5
A6	Yld2003 parameter a_6
A7	Yld2003 parameter a_7
A8	Yld2003 parameter a_8
S00	Yield stress in 0° direction
S45	Yield stress in 45° direction
S90	Yield stress in 90° direction
SBB	Balanced biaxial flow stress
R00	R-ratio in 0° direction
R45	R-ratio in 45° direction
R90	R-ratio in 90° direction
RBB	Balance biaxial flow ratio
A	YLD89 parameter a
C	YLD89 parameter c
H	YLD89 parameter h
P	YLD89 parameter p
QX1	Kinematic hardening parameter Q_{x1}
CX1	Kinematic hardening parameter C_{x1}
QX2	Kinematic hardening parameter Q_{x2}

VARIABLE	DESCRIPTION
CX2	Kinematic hardening parameter C_{x2}
EDOT	Strain rate parameter $\dot{\varepsilon}_0$
M	Strain rate parameter m
EMIN	Lower limit of the isotropic hardening rate $\frac{dR}{d\bar{\varepsilon}}$. This feature is included to model a non-zero and linear/exponential isotropic work hardening rate at large values of effective plastic strain. If the isotropic work hardening rate predicted by the utilized Voce-type work hardening rule falls below the specified value it is substituted by the prescribed value or switched to a power-law hardening if S100.NE.0. This option should be considered for problems involving extensive plastic deformations. If process dependent material characteristics are prescribed, i.e. if LC .GT. 0 the same minimum tangent modulus is assumed for all the prescribed work hardening curves. If instead EMIN.LT.0 then –EMIN defines the plastic strain value at which the linear or power-law hardening approximation commences.
S100	Yield stress at 100% strain for using a power-law approximation beyond the strain defined by EMIN.
AOPT	Material axes option (see MAT_*OPTION* TROPIC_ELASTIC for a more complete description): EQ.0.0: locally orthotropic with material axes determined by element nodes 1, 2, and 4, as with *DEFINE_COORDINATE_ NODES. EQ.1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center; this is the a-direction. This option is for solid elements only. EQ.2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_ VECTOR. EQ.3.0: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle, BETA, from a line in the plane of the element defined by the cross product of the vector **v** with the element normal. EQ.4.0: locally orthotropic in cylindrical coordinate system with the material axes determined by a vector **v**, and an originating point, P, which define the centerline axis. This option is for solid elements only. LT.0.0: the absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_NODES, *DEFINE_

	COORDINATE_SYSTEM or *DEFINE_COORDINATE_ VECTOR). Available in R3 version of 971 and later.
BETA	Material angle in degrees for AOPT=3, may be overwritten on the element card, see *ELEMENT_SHELL_BETA or *ELEMENT_ SOLID_ORTHO.

VARIABLE	DESCRIPTION
XP YP ZP	Coordinates of point **p** for AOPT = 1.
A1 A2 A3	Components of vector **a** for AOPT = 2.
V1 V2 V3	Components of vector **v** for AOPT = 3
D1 D2 D3	Components of vector **d** for AOPT = 2.

Remarks:

If FLG=1, i.e. if the yield surface parameters $a_1 \_ a_8$ are identified on the basis of prescribed material data internally in the material routine, files with point data for plotting of the identified yield surface, along with the predicted directional variation of the yield stress and plastic flow are generated in the directory where the LS-DYNA analysis is run. Four different files are generated for each specified material.

These files are named according to the scheme:

1. Contour_1#
2. Contour_2#
3. Contour_3#
4. R_and_S#

Where # is a value starting at 1.

The three first files contain contour data for plotting of the yield surface as shown in Figure 135.1. To generate these plots a suitable plotting program should be adopted and for each file/plot, column A should be plotted vs. columns B. For a more detailed description of these plots it is referred to References. Figure 135.2 further shows a plot generated from the final file named 'R_and_S#' showing the directional dependency of the normalized yield stress (column A vs. B) and plastic strain ratio (column B vs. C).

The yield condition for this material can be written

$$t\left(\boldsymbol{\sigma},\boldsymbol{\alpha},\varepsilon^p,\dot{\varepsilon}^p\right)=\sigma_{eff}\left(\boldsymbol{\sigma},\boldsymbol{\alpha}\right)-\sigma_Y\left(\varepsilon^p,\dot{\varepsilon}^p\right)$$

where

$$\sigma_Y = \left(\sigma_0 + R\left(\varepsilon^p\right)\right)\left(1+\frac{\dot{\varepsilon}^p}{\dot{\varepsilon}_0}\right)^C$$

where the isotropic hardening reads

$$R\left(\dot{\varepsilon}^p\right) = Q_{R1}\left(1-\exp\left(-C_{R1}\varepsilon^p\right)\right) + Q_{R2}\left(1-\exp\left(-C_{R2}\varepsilon^p\right)\right).$$

For the Weak Texture Model the yield function is defined as

$$\sigma_{eff} = \left[\frac{1}{2}\left\{a\left(k_1+k_2\right)^m + a\left(k_1-k_2\right)^m + C\left(2k_2\right)^m\right\}\right]^{\frac{1}{m}}$$

where

$$k_1 = \frac{\sigma_x + h\,\sigma_y}{2}$$

$$k_2 = \sqrt{\left(\frac{\sigma_x + h\,\sigma_y}{2}\right)^2 + \left(r\,\sigma_{xy}\right)^2}\ .$$

For the Strong Texture Model the yield function is defined as

$$\sigma_{eff} = \left[\frac{1}{2}\left\{\left(\sigma_1'\right)^m + \left(\sigma_2'\right)^m + \left(\sigma_1'' - \sigma_2''\right)^m\right\}\right]^{\frac{1}{m}}$$

where

$$\left\{\begin{matrix}\sigma_1' \\ \sigma_2'\end{matrix}\right\} = \frac{a_8\sigma_x + a_1\sigma_y}{2} \pm \sqrt{\left(\frac{a_2\sigma_x - a_3\sigma_y}{2}\right)^2 + a_4^2\sigma_{xy}^2}$$

$$\left\{\begin{matrix}\sigma_1'' \\ \sigma_2''\end{matrix}\right\} \frac{\sigma_x + \sigma_y}{2} \pm \sqrt{\left(\frac{a_5\sigma_x - a_6\sigma_y}{2}\right)^2 + a_7^2\,\sigma_{xy}}\ .$$

Kinematic hardening can be included by

$$\boldsymbol{\alpha} = \sum_{R=1}^{2}\boldsymbol{\alpha}_R$$

where each of the kinematic hardening variables α_R is independent and obeys a nonlinear evolutionary equation in the form

$$\dot{\boldsymbol{\alpha}}_R = C_{\alpha i}\left(Q_{\alpha i}\frac{\boldsymbol{\tau}}{\sigma} - \boldsymbol{\alpha}_R\right)\dot{\varepsilon}^p$$

where the effective stress $\bar{\sigma}$ is defined as $\bar{\sigma} = \sigma_{eff}\left(\boldsymbol{\tau}\right)$

where

$$\boldsymbol{\tau} = \boldsymbol{\sigma} - \boldsymbol{\alpha}\,.$$

Critical thickness strain failure in a layer is assumed to occur when

$$\varepsilon_t \le \varepsilon_{tc}$$

where ε_{tc} is a material parameter. It should be noted that ε_{tc} is a negative number (i.e. failure is assumed to occur only in the case of thinning).

Cockcraft and Latham fracture is assumed to occur when

$$W = \int \max(\sigma_1, 0)\, d\varepsilon^p \ge W_C$$

where σ_1 is the maximum principal stress and W_C is a material parameter.

Table 135.1

History Variable	*Description*
1	Isotropic hardening value R_1
2	Isotropic hardening value R_2
3	Increment in effective plastic strain $\Delta\bar{\varepsilon}$
4	Not defined, for internal use in the material model
5	Not defined, for internal use in the material model
6	Not defined, for internal use in the material model
7	Failure in integration point EQ.0: No failure EQ.1: Failure due to EPSC, i.e. $\varepsilon_t \ge \varepsilon_{tc}$. EQ.2: Failure due to WC, i.e. $W \ge W_c$. EQ.3: Failure due to TAUC, i.e. $\tau \ge \tau_c$
8	Sum of incremental strain in local element x-direction: $\varepsilon_{xx} = \sum \Delta\varepsilon_{xx}$
9	Sum of incremental strain in local element y-direction: $\varepsilon_{yy} = \sum \Delta\varepsilon_{yy}$
10	Value of theh Cockcroft-Latham failure parameter $W = \sum \sigma_1 \Delta p$
11	Plastic strain component in thickness direction ε_t
12	Mean value of increments in plastic strain through the thickness (For use with the non-local instability criterion. Note that constant lamella thickness is assumed and the instability criterion can give unrealistic results if used with a user-defined integration rule with varying

	lamella thickness.)
13	Not defined, for internal use in the material model
14	Nonlocal value $\rho = \frac{\Delta\varepsilon_3}{\Delta\varepsilon_3^{\Omega}}$

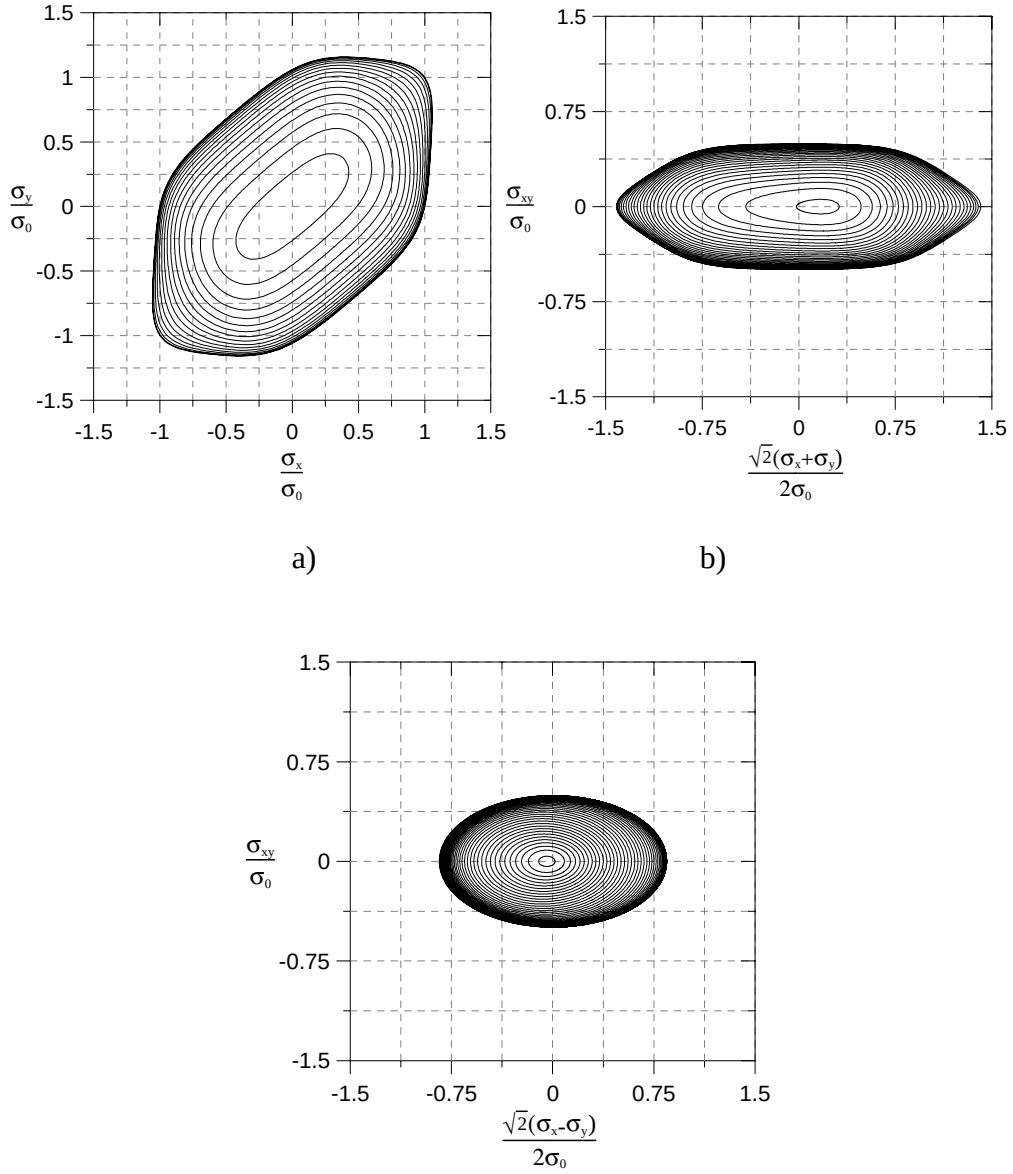


Figure 135.1. Contour plots of the yield surface generated from the files a) 'Contour _1<#>', b) 'Contour_2<#>', and c) 'Contour_3<#>'.

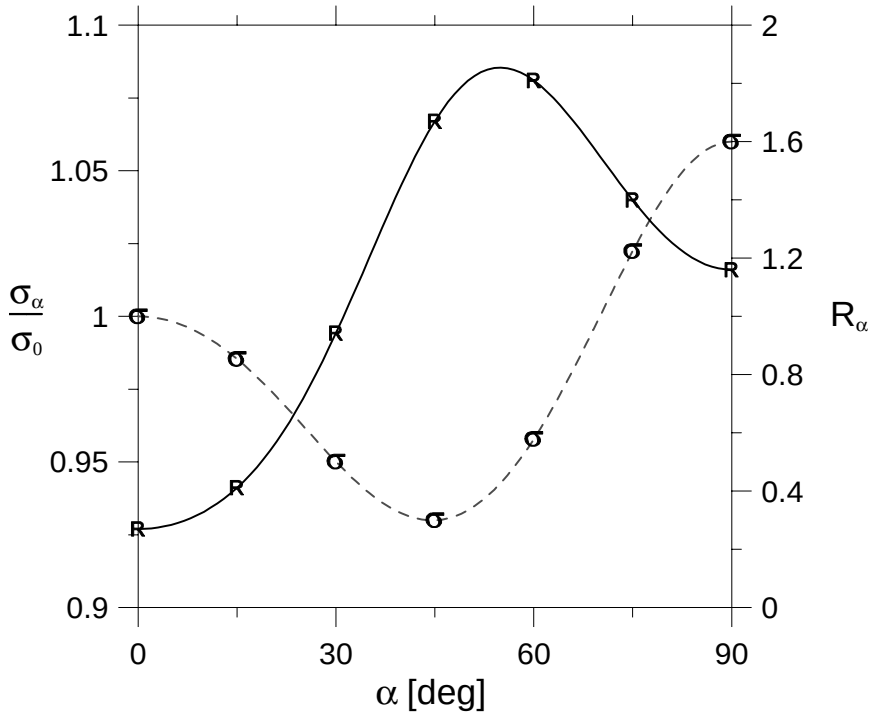


Figure 135.2. Predicted directional variation of the yield stress and plastic flow generated from the file ‘R_and_S<#>’.

*MAT_WTM_STM_PLC

This is Material Type 135. This anisotropic material adopts the yield criteria proposed by Aretz [2004]. The material strength is defined by McCormick's constitutive relation for materials exhibiting negative steady-state Strain Rate Sensitivity (SRS). McCormick [1998] and Zhang, McCormick and Estrin [2001].

Card 1

Variable	MID	RO	E	PR	NUMFI	EPSC	WC	TAUC
Type	A8	F	F	F	F	F	F	F

Card 2

Variable	SIGMA0	QR1	CR1	QR2	CR2	K		
Type	F	F	F	F	F	F		

Card 3

Variable	A1	A2	A3	A4	A5	A6	A7	A8
Type	F	F	F	F	F	F	F	F

Card 4

Variable	S	H	OMEGA	TD	ALPHA	EPS0		
Type	F	F	F	F	F	F		

Card 5	1	2	3	4	5	6	7	8
Variable	AOPT	BETA						
Type	F	F						

Card 6

Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card 7

Variable	V1	V2	V3	D1	D2	D3		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
E	Young's modulus
PR	Poisson's ratio
NUMFI	Number of through thickness integration points that must fail before the element is deleted (remember to change this number if switching between full and reduced integration type of elements).
EPSC	Critical value ε_{tC} of the plastic thickness strain.
WC	Critical value W_c for the Cockcroft-Latham fracture criterion.
TAUC	Critical value τ_c for the shear fracture criterion.

VARIABLE	DESCRIPTION
SIGMA0	Initial yield stress σ_0
QR1	Isotropic hardening parameter, Q_{R1}
CR1	Isotropic hardening parameter, C_{R1}
QR2	Isotropic hardening parameter, Q_{R2}
CR2	Isotropic hardening parameter, C_{R2}
K	*k* equals half the exponent *m* for the yield criterion
A1	Yld2003 parameter, a_1
A2	Yld2003 parameter, a_2
A3	Yld2003 parameter, a_3
A4	Yld2003 parameter, a_4
A5	Yld2003 parameter, a_5
A6	Yld2003 parameter, a_6
A7	Yld2003 parameter, a_7
A8	Yld2003 parameter, a_8
S	Dynamic strain aging parameter, *S*.
H	Dynamic strain aging parameter, *H*.
OMEGA	Dynamic strain aging parameter, Ω.
TD	Dynamic strain aging parameter, t_d.
ALPHA	Dynamic strain aging parameter, α.
EPS0	Dynamic strain aging parameter, $\dot{\varepsilon}_0$.
AOPT	Material axes option (see Mat_*OPTION* TROPIC_ELASTIC for a more complete description)

VARIABLE	DESCRIPTION
	EQ.0.0: Locally orthotropic with material axes determined by element nodes as shown in Figure 2.1. Nodes 1, 2 and 4 of an element are identical to the node used for the definition of a coordinate system as by *DEFINE_COORDINATE_NODES. EQ.2.0: Globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_ VECTOR. EQ.3.0: Locally orthotropic material axes determined by offsetting the material axes by an angle, OFFANG, from a line determined by taking the cross product of the vector v with the normal to the plane of the element. LT.0.0: the absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_NODES, *DEFINE_ COORDINATE_SYSTEM or *DEFINE_COORDINATE_ VECTOR). Available in R3 version of 971 and later.
BETA	Material angle in degrees for AOPT=3, may be overwritten on the element card, see *ELEMENT_SHELL_BETA or *ELEMENT_SHELL_ORTHO.
XP YP ZP	Coordinates of point **p** for AOPT=1.
A1 A2 A3	Components of vector **a** for AOPT=2.
V1 V2 V3	Components of vector **v** for AOPT=3.
D1 D2 D3	Components of vector **d** for AOPT=2.

Remarks:

The yield function is defined as

$$f = \bar{f}(\boldsymbol{\sigma}) - \left[\sigma_Y(t_a) + R(\varepsilon_p) + \sigma_v(\dot{\varepsilon}^p)\right]$$

where the equivalent stress σ_{eq} is defined as by an anisotropic yield criterion

$$\sigma_{eq} = \left[\frac{1}{2}\left(|\sigma_1'|^m + |\sigma_2'|^m + |\sigma_1'' - \sigma_2''|\right)\right]^{\frac{1}{m}}$$

where

$$\begin{Bmatrix}\sigma_1' \\ \sigma_2'\end{Bmatrix} = \frac{a_8\sigma_{xx} + a_1\sigma_{yy}}{2} \pm \sqrt{\left(\frac{a_2\sigma_{xx} - a_3\sigma_{yy}}{2}\right)^2 + a_4^2\sigma_{xy}^2}$$

and

$$\begin{Bmatrix}\sigma_1''\\ \sigma_2''\end{Bmatrix} = \frac{\sigma_{xx}+\sigma_{yy}}{2} \pm \sqrt{\left(\frac{a_5\sigma_{xx}-a_6\sigma_{yy}}{2}\right)^2 + a_7^2\sigma_{xy}^2}$$

The strain hardening function *R* is defined by the extended Voce law

$$R\left(\varepsilon^p\right) = \sum_{i=1}^{2} Q_{Ri}\left(1-\exp\left(-C_{Ri}\varepsilon^p\right)\right)$$

Where ε^p is the effective (or accumulated) plastic strain, and Q_{Ri} and C_{Ri} are strain hardening parameters.

Viscous stress σ_v is given by

$$\sigma_v = \left(\dot{\varepsilon}^p\right) = s\ln\left(1+\frac{\dot{\varepsilon}^p}{\dot{\varepsilon}_0}\right)$$

Where *S* represents the instantaneous strain rate sensitivity (SRS) and $\dot{\varepsilon}_0$ is a reference strain rate. In this model the yield strength, including the contribution from dynamic strain aging (DSA) is defined as

$$\sigma_Y\left(t_a\right) = \sigma_0 + SH\left[1-\exp\left\{-\left(\frac{t_a}{t_d}\right)^{\alpha}\right\}\right]$$

Where σ_0 is the yield strength for vanishing average waiting time, t_a, i.e. at high strain rates, and *H*, α and t_d are material constants linked to dynamic strain aging. It is noteworthy that σ_Y is an increasing function of t_a. The average waiting time is defined by the evolution equation

$$\dot{t}_a = 1-\frac{t_a}{t_{a,ss}}$$

where the quasi-steady waiting time $t_{a,ss}$ is given as

$$t_{a,ss} = \frac{\Omega}{\dot{\varepsilon}^p}$$

where Ω is the strain produced by all mobile dislocations moving to the next obstacle on their path.

***MAT_CORUS_VEGTER**

This is Material Type 136, a plane stress orthotropic material model for metal forming. Yield surface construction is based on the interpolation by second-order Bezier curves, and model parameters are determined directly from a set of mechanical tests conducted for a number of directions. For each direction, four mechanical tests are carried out: a uniaxial, an equi-biaxial, a plane strain tensile test and a shear test. These test results are used to determine the coefficients of the Fourier directional dependency field. For a more detailed description please see Vegter and Boogaard [2006].

Card 1

	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	N	FBI	RBI0	LCID
Type	A8	F	F	F	F	F	F	F

Card 2

Variable	SYS	SIP	SHS	SHL	ESH	E0	ALPHA	LCID2
Type	F	F	F	F	F	F	F	F

Card 3

Variable	AOPT							
Type								

Card 4

Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card 5	1	2	3	4	5	6	7	8
Variable	V1	V2	V3	D1	D2	D3		
Type	F	F	F	F	F	F		

Cards 6 up to N+6 are to define the experimental data obtained from four mechanical tests for a group of equidistantly placed directions $\theta_i = \frac{i\pi}{2N}$ (i = 0, 1, 2, … N)

Card 6	1	2	3	4	5	6	7	8
Variable	FUN-I	RUN-I	FPS1-I	FPS2-I	FSH-I			
Type	F	F	F	F	F			

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Material density
E	Elastic Young's modulus
PR	Poisson's ratio
N	Order of Fourier series (i.e., number of test groups minus one). The minimum number for N is 2, and the maximum is 12.
FBI	Normalized yield stress for equibiaxial test.
RBI0	Initial strain ratio for equibiaxial test.
LCID	Stress-strain curve ID. If defined, SYS, SIP, SHS, and SHL are ignored.
SYS	Static yield stress, σ_0.
SIP	Stress increment parameter, $\Delta\sigma_m$
SHS	Strain hardening parameter for small strain, β.

VARIABLE	DESCRIPTION
SHL	Strain hardening parameter for larger strain, Ω.
ESH	Exponent for strain hardening, *n*.
E0	Initial plastic strain
ALPHA	α distribution of hardening used in the curve-fitting. $\alpha = 0$ pure kinematic hardening and $\alpha = 1$ provides pure isotropic hardening.
LCID2	Curve ID. The curve defines Young's modulus change with respect to the plastic strain. By default it is assumed that Young's modulus remains constant. Effective value is between 0-1.
AOPT	Material axes option (see MAT_*OPTION* TROPIC_ELASTIC for a more complete description): EQ.0.0: locally orthotropic with material axes determined by element nodes 1, 2, and 4, as with *DEFINE_COORDINATE_NODES. EQ.1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center; this is the a-direction. This option is for solid elements only. EQ.2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR. EQ.3.0: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle, BETA, from a line in the plane of the element defined by the cross product of the vector **v** with the element normal. EQ.4.0: locally orthotropic in cylindrical coordinate system with the material axes determined by a vector **v**, and an originating point, P, which define the centerline axis. This option is for solid elements only. LT.0.0: the absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_NODES, *DEFINE_COORDINATE_SYSTEM or *DEFINE_COORDINATE_VECTOR). Available in R3 version of 971 and later.
BETA	Material angle in degrees for AOPT=3, may be overwritten on the element card, see *ELEMENT_SHELL_BETA or *ELEMENT_SOLID_ORTHO.
XP YP ZP	Coordinates of point **p** for AOPT = 1.
A1 A2 A3	Components of vector **a** for AOPT = 2.
V1 V2 V3	Components of vector **v** for AOPT = 3

VARIABLE	DESCRIPTION
D1 D2 D3	Components of vector **d** for AOPT = 2.
FUN-I	Normalized yield stress for uniaxial test for the ith direction.
RUN-I	Strain ratio for uniaxial test for the ith direction.
FPS1-I	First normalized yield stress for plain strain test for the ith direction.
FPS2-I	Second normalized yield stress for plain strain test for the ith direction.
FSH-I	First normalized yield stress for pure shear test for the ith direction.

Remarks:

The yield criterion is chosen as:

$$\sigma_y = \sigma_0 + \Delta\sigma_m \left[\beta\varepsilon_{eq} + \left(1 - e^{-\Omega\varepsilon_{eq}}\right)^n \right]$$

*MAT_COHESIVE_MIXED_MODE

This is Material Type 138. This model is a simplification of *MAT_COHESIVE_GENERAL restricted to linear softening. It includes a bilinear traction-separation law with quadratic mixed mode delamination criterion and a damage formulation. It can be used with solid element types 19 and 20, and is not available for other solid element formulations. See the remarks after *SECTION_SOLID for a description of element types 19 and 20.

Card 1

Variable	MID	RO	ROFLG	INTFAIL	EN	ET	GIC	GIIC
Type	A8	F	F	F	F	F	F	F

Card 2

Variable	XMU	T	S	UND	UTD			
Type	F	F	F	F	F			

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
ROFLG	Flag for whether density is specified per unit area or volume. ROFLG=0 specified density per unit volume (default), and ROFLG=1 specifies the density is per unit area for controlling the mass of cohesive elements with an initial volume of zero.
INTFAIL	The number of integration points required for the cohesive element to be deleted. If it is zero, the element won't be deleted even if it satisfies the failure criterion. The value of INTFAIL may range from 1 to 4, with 1 the recommended value.
EN	The stiffness (units of stress/length) normal to the plane of the cohesive element.

ET	The stiffness (units of stress/length)) in the plane of the cohesive element.

VARIABLE	DESCRIPTION
GIC	Energy release rate for mode I (units of stress*length)
GIIC	Energy release rate for mode II (units of stress*length)
XMU	Exponent of the mixed mode criteria (see remarks below)
T	Peak traction (stress units) in normal direction LT.0.0: Load curve ID = (-T) which defines peak traction in normal direction as a function of element size. See remarks.
S	Peak traction (stress units) in tangential direction LT.0.0: Load curve ID = (-S) which defines peak traction in tangential direction as a function of element size. See remarks.
UND	Ultimate displacement in the normal direction
UTD	Ultimate displacement in the tangential direction

Remarks:

The ultimate displacements in the normal and tangential directions are the displacements at the time when the material has failed completely, i.e., the tractions are zero. The linear stiffness for loading followed by the linear softening during the damage provides an especially simple relationship between the energy release rates, the peak tractions, and the ultimate displacements:

$$GIC = T \cdot UND / 2$$
$$GIIC = S \cdot UTD / 2$$

If the peak tractions aren't specified, they are computed from the ultimate displacements. See Fiolka and Matzenmiller [2005] and Gerlach, Fiolka and Matzenmiller [2005].

In this cohesive material model, the total mixed-mode relative displacement δ_m is defined as $\delta_m = \sqrt{\delta_I^2 + \delta_{II}^2}$, where $\delta_I = \delta_3$ is the separation in normal direction (mode I) and $\delta_{II} = \sqrt{\delta_1^2 + \delta_2^2}$ is the separation in tangential direction (mode II). The mixed-mode damage initiation displacement δ^0 (onset of softening) is given by

$$\delta^0 = \delta_I^0 \delta_{II}^0 \sqrt{\frac{1+\beta^2}{\left(\delta_{II}^0\right)^2 + \left(\beta \delta_I^0\right)^2}}$$

where $\delta_I^0 = T/EN$ and $\delta_{II}^0 = S/ET$ are the single mode damage inititation separations and $\beta = \delta_{II}/\delta_I$ is the "mode mixity" (see Figure 138.1). The ultimate mixed-mode displacement δ^F (total failure) for the power law (XMU>0) is:

$$\delta^F = \frac{2(1+\beta)^2}{\delta^0}\left[\left(\frac{EN}{GIC}\right)^{XMU} + \left(\frac{ET\cdot\beta^2}{GIIC}\right)^{XMU}\right]^{-\frac{1}{XMU}}$$

and alternatively for the Benzeggagh-Kenane law [1996] (XMU<0):

$$\delta^F = \frac{2}{\delta^0\left(\frac{1}{1+\beta^2}EN + \frac{\beta^2}{1+\beta^2}ET\right)}\left[GIC + (GIIC - GIC)\left(\frac{\beta^2\cdot ET}{EN+\beta^2\cdot ET}\right)^{|XMU|}\right]$$

In this model, damage of the interface is considered, i.e. irreversible conditions are enforced with loading/unloading paths coming from/pointing to the origin.

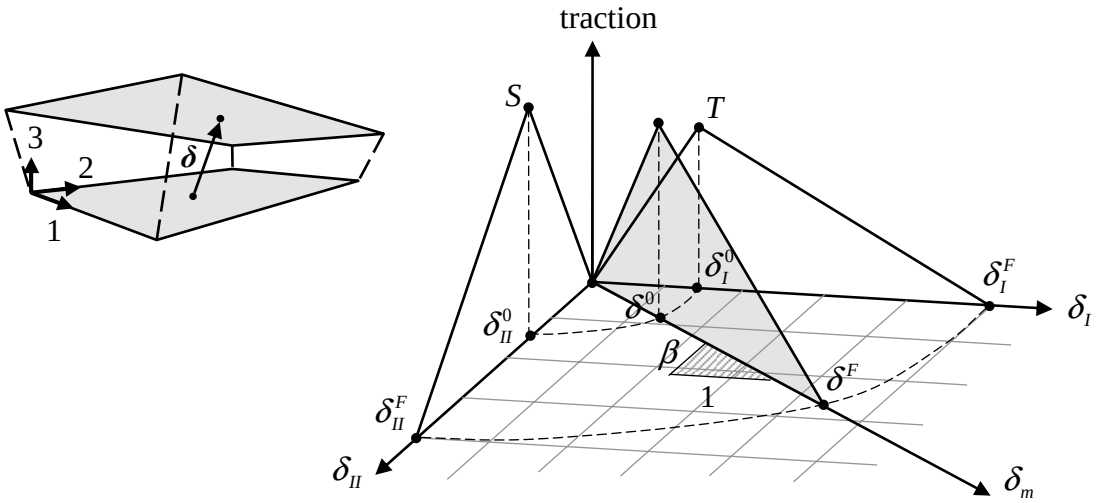


Figure 138.1. Mixed-mode traction-separation law

Peak tractions T and/or S can be defined as functions of characteristic element length (square root of midsurface area) via load curve. This option is useful to get nearly the same global responses (e.g. load-displacement curve) with coarse meshes when compared to a fine mesh solution. In general, lower peak traction values are needed for coarser meshes

Two error checks have been implemented for this material model in order to ensure proper material data. Since the traction versus displacement curve is fairly simple (triangular shaped), equations can be developed to ensure that the displacement (L) at the peak load (QMAX), is smaller than the ultimate distance for failure (u). See Figure 138.2 for the used notation.

One has that

$$GC = \frac{1}{2} u \cdot QMAX \text{ and } L = \frac{QMAX}{E}.$$

To ensure that the peak is not past the failure point, $\frac{u}{L}$ must be larger than 1.

$$u = \frac{2GC}{EL},$$

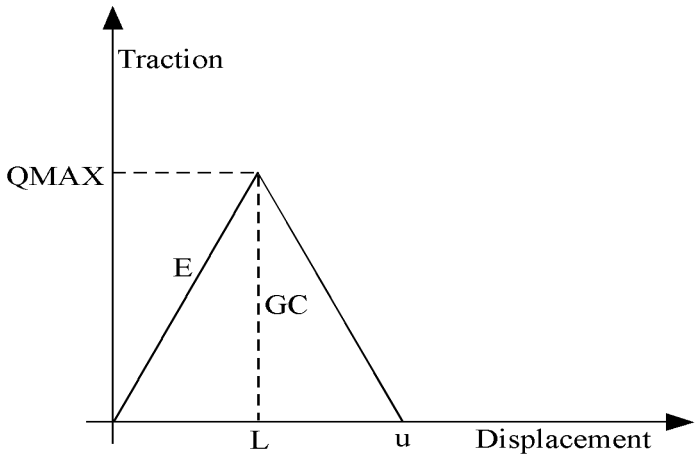


Figure 138.2. Bilinear traction-separation law

where *GC* is the energy release rate. This gives

$$\frac{u}{L} = \frac{2GC}{ELL} = \frac{2GC}{E\left(\frac{QMAX}{E}\right)^2} > 1.$$

The error checks are then done for tension and pure shear, respectively,

$$\frac{u}{L} = \frac{(2GIC)}{EN\left(\frac{T}{EN}\right)^2} > 1,$$

$$\frac{u}{L} = \frac{(2GIIC)}{ET\left(\frac{S}{ET}\right)^2} > 1.$$

*MAT_MODIFIED_FORCE_LIMITED

This is Material Type 139. This material for the Belytschko-Schwer resultant beam is an extension of material 29. In addition to the original plastic hinge and collapse mechanisms of material 29, yield moments may be defined as a function of axial force. After a hinge forms, the moment transmitted by the hinge is limited by a moment-plastic rotation relationship.

Card 1

Variable	MID	RO	E	PR	DF	AOPT	YTFLAG	ASOFT
Type	A8	F	F	F	F	F	F	F
Default	none	none	none	none	0.0	0.0	0.0	0.0

Card 2

Variable	M1	M2	M3	M4	M5	M6	M7	M8
Type	F	F	F	F	F	F	F	F
Default	none	0	0	0	0	0	0	0

Card 3

Variable	LC1	LC2	LC3	LC4	LC5	LC6	LC7	LC8
Type	F	F	F	F	F	F	F	F
Default	none	0	0	0	0	0	0	0

Card 4

Variable	LPS1	SFS1	LPS2	SFS2	YMS1	YMS2		
Type	F	F	F	F	F	F		
Default	0	1.0	LPS1	1.0	1.0E+20	YMS1		

Card 5

Variable	LPT1	SFT1	LPT2	SFT2	YMT1	YMT2		
Type	F	F	F	F	F	F		
Default	0	1.0	LPT1	1.0	1.0E+20	YMT1		

Card 6

Variable	LPR	SFR	YMR					
Type	F	F	F					
Default	0	1.0	1.0E+20					

Card 7

Variable	LYS1	SYS1	LYS2	SYS2	LYT1	SYT1	LYT2	SYT2
Type	F	F	F	F	F	F	F	F
Default	0	1.0	0	1.0	0	1.0	0	1.0

Card 8	1	2	3	4	5	6	7	8
Variable	LYR	SYR						
Type	F	F						
Default	0	1.0						

Card 9

Variable	HMS1_1	HMS1_2	HMS1_3	HMS1_4	HMS1_5	HMS1_6	HMS1_7	HMS1_8
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

Card 10

Variable	LPMS1_1	LPMS1_2	LPMS1_3	LPMS1_4	LPMS1_5	LPMS1_6	LPMS1_7	LPMS1_8
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

Card 11

Variable	HMS2_1	HMS2_2	HMS2_3	HMS2_4	HMS2_5	HMS2_6	HMS2_7	HMS2_8
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

Card 12

Variable	LPMS2_1	LPMS2_2	LPMS2_3	LPMS2_4	LPMS2_5	LPMS2_6	LPMS2_7	LPMS2_8
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

Card 13

Variable	HMT1_1	HMT1_2	HMT1_3	HMT1_4	HMT1_5	HMT1_6	HMT1_7	HMT1_8
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

Card 14

Variable	LPMT1_1	LPMT1_2	LPMT1_3	LPMT1_4	LPMT1_5	LPMT1_6	LPMT1_7	LPMT1_8
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

Card 15

Variable	HMT2_1	HMT2_2	HMT2_3	HMT2_4	HMT2_5	HMT2_6	HMT2_7	HMT2_8
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

Card 16

Variable	1	2	3	4	5	6	7	8
	LPMT2_1	LPMT2_2	LPMT2_3	LPMT2_4	LPMT2_5	LPMT2_6	LPMT2_7	LPMT2_8
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

Card 17

Variable	1	2	3	4	5	6	7	8
	HMR_1	HMR_2	HMR_3	HMR_4	HMR_5	HMR_6	HMR_7	HMR_8
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

Card 18

Variable	1	2	3	4	5	6	7	8
	LPMR_1	LPMR_2	LPMR_3	LPMR_4	LPMR_5	LPMR_6	LPMR_7	LPMR_8
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
E	Young's modulus
PR	Poisson's ratio
DF	Damping factor, see definition in notes below. A proper control for the timestep has to be maintained by the user!

VARIABLE	DESCRIPTION
AOPT	Axial load curve option: EQ.0.0: axial load curves are force versus strain, EQ.1.0: axial load curves are force versus change in length. LT.0.0: the absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_NODES, *DEFINE_COORDINATE_SYSTEM or *DEFINE_COORDINATE_VECTOR). Available in R3 version of 971 and later.
YTFLAG	Flag to allow beam to yield in tension: EQ.0.0: beam does not yield in tension, EQ.1.0: beam can yield in tension.
ASOFT	Axial elastic softening factor applied once hinge has formed. When a hinge has formed the stiffness is reduced by this factor. If zero, this factor is ignored.
M1, M2,...,M8	Applied end moment for force versus (strain/change in length) curve. At least one must be defined. A maximum of 8 moments can be defined. The values should be in ascending order.
LC1, LC2,...,LC8	Load curve ID (see *DEFINE_CURVE) defining axial force versus strain/change in length (see AOPT) for the corresponding applied end moment. Define the same number as end moments. Each curve must contain the same number of points.
LPS1	Load curve ID for plastic moment versus rotation about s-axis at node 1. If zero, this load curve is ignored.
SFS1	Scale factor for plastic moment versus rotation curve about s-axis at node 1. Default = 1.0.
LPS2	Load curve ID for plastic moment versus rotation about s-axis at node 2. Default: is same as at node 1.
SFS2	Scale factor for plastic moment versus rotation curve about s-axis at node 2. Default: is same as at node 1.
YMS1	Yield moment about s-axis at node 1 for interaction calculations (default set to 1.0E+20 to prevent interaction).
YMS2	Yield moment about s-axis at node 2 for interaction calculations (default set to YMS1).
LPT1	Load curve ID for plastic moment versus rotation about t-axis at node 1. If zero, this load curve is ignored.
SFT1	Scale factor for plastic moment versus rotation curve about t-axis at node 1. Default = 1.0.

VARIABLE	DESCRIPTION
LPT2	Load curve ID for plastic moment versus rotation about t-axis at node 2. Default: is the same as at node 1.
SFT2	Scale factor for plastic moment versus rotation curve about t-axis at node 2. Default: is the same as at node 1.
YMT1	Yield moment about t-axis at node 1 for interaction calculations (default set to 1.0E+20 to prevent interactions)
YMT2	Yield moment about t-axis at node 2 for interaction calculations (default set to YMT1)
LPR	Load curve ID for plastic torsional moment versus rotation. If zero, this load curve is ignored.
SFR	Scale factor for plastic torsional moment versus rotation (default = 1.0).
YMR	Torsional yield moment for interaction calculations (default set to 1.0E+20 to prevent interaction)
LYS1	ID of curve defining yield moment as a function of axial force for the s-axis at node 1.
SYS1	Scale factor applied to load curve LYS1.
LYS2	ID of curve defining yield moment as a function of axial force for the s-axis at node 2.
SYS2	Scale factor applied to load curve LYS2.
LYT1	ID of curve defining yield moment as a function of axial force for the t-axis at node 1.
SYT1	Scale factor applied to load curve LYT1.
LYT2	ID of curve defining yield moment as a function of axial force for the t-axis at node 2.
SYT2	Scale factor applied to load curve LYT2.
LYR	ID of curve defining yield moment as a function of axial force for the torsional axis.
SYR	Scale factor applied to load curve LYR.
HMS1_n	Hinge moment for s-axis at node 1.

VARIABLE	DESCRIPTION
LPMS1_n	ID of curve defining plastic moment as a function of plastic rotation for the s-axis at node 1 for hinge moment HMS1_n
HMS2_n	Hinge moment for s-axis at node 2.
LPMS2_n	ID of curve defining plastic moment as a function of plastic rotation for the s-axis at node 2 for hinge moment HMS2_n
HMT1_n	Hinge moment for t-axis at node 1.
LPMT1_n	ID of curve defining plastic moment as a function of plastic rotation for the t-axis at node 1 for hinge moment HMT1_n
HMT2_n	Hinge moment for t-axis at node 2.
LPMT2_n	ID of curve defining plastic moment as a function of plastic rotation for the t-axis at node 2 for hinge moment HMT2_n
HMR_n	Hinge moment for the torsional axis.
LPMR_n	ID of curve defining plastic moment as a function of plastic rotation for the torsional axis for hinge moment HMR_n

Remarks:

This material model is available for the Belytschko resultant beam element only. Plastic hinges form at the ends of the beam when the moment reaches the plastic moment. The plastic moment versus rotation relationship is specified by the user in the form of a load curve and scale factor. The points of the load curve are (plastic rotation in radians, plastic moment). Both quantities should be positive for all points, with the first point being (zero, initial plastic moment). Within this constraint any form of characteristic may be used, including flat or falling curves. Different load curves and scale factors may be specified at each node and about each of the local s and t axes.

Axial collapse occurs when the compressive axial load reaches the collapse load. Collapse load versus collapse deflection is specified in the form of a load curve. The points of the load curve are either (true strain, collapse force) or (change in length, collapse force). Both quantities should be entered as positive for all points, and will be interpreted as compressive. The first point should be (zero, initial collapse load).

The collapse load may vary with end moment as well as with deflections. In this case several load-deflection curves are defined, each corresponding to a different end moment. Each load curve should have the same number of points and the same deflection values. The end moment is defined as the average of the absolute moments at each end of the beam and is always positive.

Stiffness-proportional damping may be added using the damping factor λ. This is defined as follows:

$$\lambda = \frac{2 * \xi}{\omega}$$

where ξ is the damping factor at the reference frequency ω (in radians per second). For example if 1% damping at 2Hz is required

$$\lambda = \frac{2 * 0.01}{2\pi * 2} = 0.001592$$

If damping is used, a small time step may be required. LS-DYNA does not check this so to avoid instability it may be necessary to control the time step via a load curve. As a guide, the time step required for any given element is multiplied by $0.3L/c\lambda$ when damping is present (L = element length, c = sound speed).

Moment Interaction:

Plastic hinges can form due to the combined action of moments about the three axes. This facility is activated only when yield moments are defined in the material input. A hinge forms when the following condition is first satisfied.

$$\left(\frac{M_r}{M_{ryield}}\right)^2 + \left(\frac{M_s}{M_{syield}}\right)^2 + \left(\frac{M_t}{M_{tyield}}\right)^2 \geq 1$$

where,

M_r, M_S, M_t = current moment

M_{ryield}, M_{syield}, M_{tyield} = yield moment

Note that scale factors for hinge behavior defined in the input will also be applied to the yield moments: for example, M_{syield} in the above formula is given by the input yield moment about the local axis times the input scale factor for the local s axis. For strain-softening characteristics, the yield moment should generally be set equal to the initial peak of the moment-rotation load curve.

On forming a hinge, upper limit moments are set. These are given by

$$M_{r_{upper}} = MAX\left(M_r, \frac{M_{r_{yield}}}{2}\right)$$

and similar for M_S and M_t.

Thereafter the plastic moments will be given by

M_{rp}, = min (M_{rupper}, M_{rcurve}) and similar for s and t

where

M_{rp} = current plastic moment

M_{rcurve} = moment taken from load curve at the current rotation scaled according to the scale factor.

The effect of this is to provide an upper limit to the moment that can be generated; it represents the softening effect of local buckling at a hinge site. Thus if a member is bent about is local s-axis it will then be weaker in torsion and about its local t-axis. For moments-softening curves, the effect is to trim off the initial peak (although if the curves subsequently harden, the final hardening will also be trimmed off).

It is not possible to make the plastic moment vary with the current axial load, but it is possible to make hinge formation a function of axial load and subsequent plastic moment a function of the moment at the time the hinge formed. This is discussed in the next section.

Independent plastic hinge formation:

In addition to the moment interaction equation, Cards 7 through 18 allow plastic hinges to form independently for the s-axis and t-axis at each end of the beam and also for the torsional axis. A plastic hinge is assumed to form if any component of the current moment exceeds the yield moment as defined by the yield moment vs. axial force curves input on cards 7 and 8. If any of the 5 curves is omitted, a hinge will not form for that component. The curves can be defined for both compressive and tensile axial forces. If the axial force falls outside the range of the curve, the first or last point in the curve will be used. A hinge forming for one component of moment does not effect the other components.

Upon forming a hinge, the magnitude of that component of moment will not be permitted to exceed the current plastic moment.. The current plastic moment is obtained by interpolating between the plastic moment vs. plastic rotation curves input on cards 10, 12, 14, 16, or 18. Curves may be input for up to 8 hinge moments, where the hinge moment is defined as the yield moment at the time that the hinge formed. Curves must be input in order of increasing hinge moment and each curve should have the same plastic rotation values. The first or last curve will be used if the hinge moment falls outside the range of the curves. If no curves are defined, the plastic moment is obtain from the curves on cards 4 through 6. The plastic moment is scaled by the scale factors on lines 4 to 6.

A hinge will form if either the independent yield moment is exceeded or if the moment interaction equation is satisfied. If both are true, the plastic moment will be set to the minimum of the interpolated value and M_{rp}.

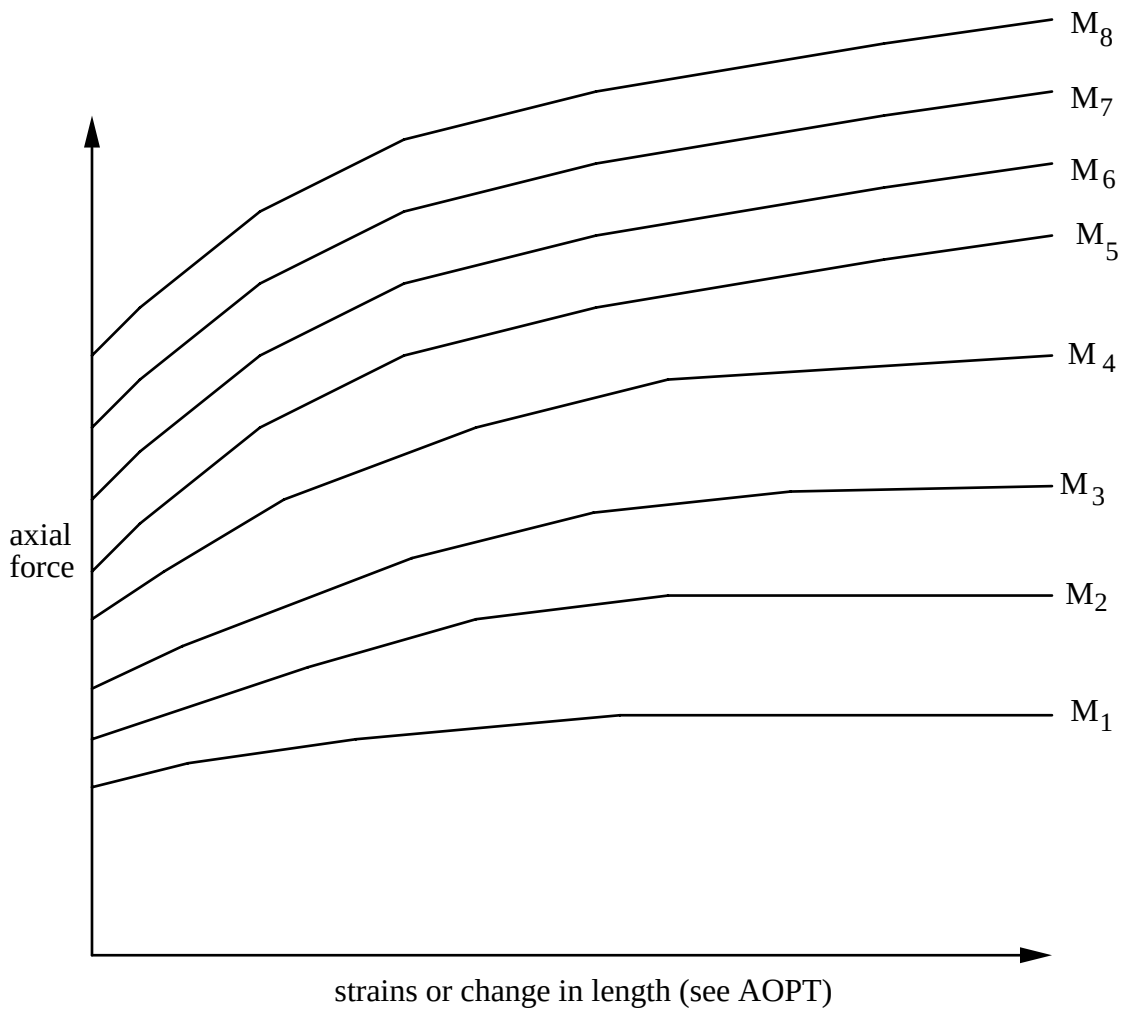


Figure 139.1. The force magnitude is limited by the applied end moment. For an intermediate value of the end moment LS-DYNA interpolates between the curves to determine the allowable force value.

***MAT_VACUUM**

This is Material Type 140. This model is a dummy material representing a vacuum in a multi-material Euler/ALE model. Instead of using ELFORM=12 (under *SECTION_SOLID), it is better to use ELFORM=11 with the void material defined as vacuum material instead.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RHO						
Type	A8	F						

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RHO	Estimated material density. This is used only as stability check.

Remarks:

1. The vacuum density is estimated. It should be small relative to air in the model (possibly at least 10^3 to 10^6 lighter than air).

*MAT_RATE_SENSITIVE_POLYMER

This is Material Type 141. This model is for the simulation of an isotropic ductile polymer with strain rate effects [Stouffer and Dame 1996]. Uniaxial test data has to be used.

Card 1

1	2	3	4	5	6	7	8
MID	RO	E	PR	Do	N	Zo	q
A8	F	F	F	F	F	F	F

Card 2

1	2	3	4	5	6	7	8
Omega							
F							

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
E	Elastic modulus.
PR	Poisson's ratio
Do	Reference strain rate (=1000*max strain rate used in the test).
N	Exponent (see inelastic strain rate equation below)
Zo	Initial hardness of material
q	(see equations below).
Omega	Maximum internal stress.

Remarks:

$$\varepsilon_{ij} = D_o \exp\left[-0.5\left(\frac{Z_o^2}{3K_2}\right)\right]\left(\frac{S_{ij}-\Omega_{ij}}{\sqrt{K_2}}\right)$$

where the K_2 term is defined as follows:

$$K_2 = 0.5(S_{ij}-\Omega_{ij})(S_{ij}-\Omega_{ij})$$

and represent the second invariant of the overstress tensor. The elastic components of the strain are added to the inelastic strain to obtain the total strain. The following relationship defines the internal stress variable rate:

$$\Omega_{ij} = \frac{2}{3} q\Omega_m \dot{\varepsilon}_{ij}^I - q\Omega_{ij}\dot{\varepsilon}_e^I$$

where q is a material constant, Ω_m is a material constant that represents the maximum value of the internal stress, and $\dot{\varepsilon}_e^I$ is the effective inelastic strain.

*MAT_TRANSVERSELY_ISOTROPIC_CRUSHABLE_FOAM

This is Material Type 142. This model is for an extruded foam material that is transversely isotropic, crushable, and of low density with no significant Poisson effect. This material is used in energy-absorbing structures to enhance automotive safety in low velocity (bumper impact) and medium high velocity (interior head impact and pedestrian safety) applications. The formulation of this foam is due to Hirth, Du Bois, and Weimar and is documented by Du Bois [2001]. This model behaves in a more physical way for off axis loading the material, *MAT_HONEYCOMB, which can exhibit nonphysical stiffening for loading conditions that are off axis. The load curves are used to define a yield surface that bounds the deviatoric stress tensor.

Card 1

1	2	3	4	5	6	7	8
MID	RO	E11	E22	E12	E23	G	K
A8	F	F	F	F	F	F	

Card 2

1	2	3	4	5	6	7	8
I11	I22	I12	I23	IAA	NY	ANG	MU
I	I	I	I	I	I	F	F

Card 3

1	2	3	4	5	6	7	8
AOPT	ISCL	MACF					
F	I	I					

Card 4

1	2	3	4	5	6	7	8
XP	YP	ZP	A1	A2	A3		
F	F	F	F	F	F		

Card 5	1	2	3	4	5	6	7	8
Variable	D1	D2	D3	V1	V2	V3		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
E11	Elastic modulus in axial direction.
E22	Elastic modulus in transverse direction (E22=E33).
E12	Elastic shear modulus (E12=E31).
E23	Elastic shear modulus in transverse plane.
G	Shear modulus.
K	Bulk modulus for contact stiffness.
I11	Load curve for nominal axial stress versus volumetric strain.
I22	Load curve ID for nominal transverse stresses versus volumetric strain (I22=I33).
I12	Load curve ID for shear stress component 12 and 31 versus volumetric strain (I12=I31).
I23	Load curve ID for shear stress component 23 versus volumetric strain.
IAA	Load curve ID (optional) for nominal stress versus volumetric strain for load at angle, ANG, relative to the material axis.
NY	Set to unity for a symmetric yield surface.
ANG	Angle corresponding to load curve ID, IAA.

VARIABLE	DESCRIPTION
MU	Damping coefficient for tensor viscosity which acts in both tension and compression. Recommended values vary between 0.05 to 0.10. If zero, tensor viscosity is not used, but bulk viscosity is used instead. Bulk viscosity creates a pressure as the element compresses that is added to the normal stresses, which can have the effect of creating transverse deformations when none are expected.
AOPT	Material axes option (see MAT_*OPTION* TROPIC_ELASTIC for a more complete description): EQ.0.0: locally orthotropic with material axes determined by element nodes as shown in Figure 2.1. Nodes 1, 2, and 4 of an element are identical to the nodes used for the definition of a coordinate system as by *DEFINE_COORDINATE_NODES. EQ.1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center; this is the a-direction. This option is for solid elements only. EQ.2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR. EQ.3.0: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle, BETA, from a line in the plane of the element defined by the cross product of the vector **v** with the element normal. The plane of a solid element is the midsurface between the inner surface and outer surface defined by the first four nodes and the last four nodes of the connectivity of the element, respectively. EQ.4.0: locally orthotropic in cylindrical coordinate system with the material axes determined by a vector **v**, and an originating point, P, which define the centerline axis. This option is for solid elements only. LT.0.0: the absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_NODES, *DEFINE_COORDINATE_SYSTEM or *DEFINE_COORDINATE_VECTOR). Available in R3 version of 971 and later.
ISCL	Load curve ID for the strain rate scale factor versus the volumetric strain rate. The yield stress is scaled by the value specified by the load curve.
MACF	Material axes change flag: EQ.1: No change, default, EQ.2: switch material axes a and b, EQ.3: switch material axes a and c, EQ.4: switch material axes b and c.
XP YP ZP	Coordinates of point **p** for AOPT = 1 and 4.

VARIABLE	DESCRIPTION
A1 A2 A3	Components of vector **a** for AOPT = 2.
D1 D2 D3	Components of vector **d** for AOPT = 2.
V1 V2 V3	Define components of vector **v** for AOPT = 3 and 4.

Remarks:

Tensor viscosity, which is activated by a nonzero value for MU, is generally more stable than bulk viscosity. A damping coefficient less than 0.01 has little effect, and a value greater than 0.10 may cause numerical instabilities.

***MAT_WOOD_**{*OPTION*}

This is Material Type 143. This is a transversely isotropic material and is available for solid elements. The user has the option of inputting his or her own material properties (**<BLANK>**), or requesting default material properties for Southern yellow pine (**PINE**) or Douglas fir (**FIR**). This model was developed by Murray [2002] under a contract from the FHWA.

Available options include:

<BLANK>

PINE

FIR

Card 1

1	2	3	4	5	6	7	8
Variable	MID	RO	NPLOT	ITERS	IRATE	GHARD	IFAIL	IVOL
Type	A8	F	I	I	I	F	I	I

Define the following card for the PINE and FIR options.

Card 2

1	2	3	4	5	6	7	8
Variable	MOIS	TEMP	QUAL_T	QUAL_C	UNITS	IQUAL		
Type	F	F	F	F	I	I		

Define the following cards, 2-6, for the <BLANK> option.

Card 2

1	2	3	4	5	6	7	8
Variable	EL	ET	GLT	GTR	PR			
Type	F	F	F	F	F			

Card 3	1	2	3	4	5	6	7	8
Variable	XT	XC	YT	YC	SXY	SYZ		
Type	F	F	F	F	F	F		

Card 4

Variable	$GF1_{\parallel}$	$GF2_{\parallel}$	BFIT	$DMAX_{\parallel}$	$GF1_{\perp}$	$GF2_{\perp}$	DFIT	$DMAX_{\perp}$
Type	F	F	F	F	F	F	F	F

Card 5

Variable	FLPAR	FLPARC	POWPAR	FLPER	FLPERC	POWPER		
Type	F	F	F	F	F	F		

Card 6

Variable	NPAR	CPAR	NPER	CPER				
Type	F	F	F	F				

Define the following three cards for all options.

Card 3/7	1	2	3	4	5	6	7	8
Variable	AOPT	MACF	BETA					
Type	F	I	F					

Card 4/8

	1	2	3	4	5	6	7	8
Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card 5/9

Variable	D1	D2	D3	V1	V2	V3		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
NPLOT	Controls what is written as component 7 to the d3plot database. LS-PrePost always blindly labels this component as effective plastic strain.: EQ.1: Parallel damage (default). EQ.2: Perpendicular damage.
ITERS	Number of plasticity algorithm iterations. The default is one iteration.
IRATE	Rate effects option: EQ.0: Rate effects model turned off (default). EQ.1: Rate effects model turned on.
GHARD	Perfect plasticity override. Values greater than or equal to zero are allowed. Positive values model late time hardening in compression (an increase in strength with increasing strain). A zero value models perfect plasticity (no increase in strength with increasing strain). The default is zero.
IFAIL	Erosion perpendicular to the grain. EQ.0: No (default). EQ.1: Yes (not recommended except for debugging).
IVOL	Erode on negative volume or strain increments greater than 0.01. EQ.0: No, do not apply erosion criteria. EQ.1: Yes, apply volume and strain erosion criteria.

VARIABLE	DESCRIPTION
MOIS	Percent moisture content. If left blank, moisture content defaults to saturated at 30%.
TEMP	Temperature in ˚C. If left blank, temperature defaults to room temperature at 20 ˚C
QUAL_T	Quality factor options. These quality factors reduce the clear wood tension, shear, and compression strengths as a function of grade. EQ.0: Grade 1, 1D, 2, 2D. Predefined strength reduction factors are: Pine: Qual_T=0.47 in tension/shear. Qual_C=0.63 in compression. Fir: Qual_T=0.40 in tension/shear Qual_C=0.73 in compression. EQ.-1: DS-65 or SEl STR (pine and fir). Predefined strength reduction factors are: Qual_T=0.80 in tension/shear. Qual_C=0.93 in compression. EQ.-2: Clear wood. No strength reduction factors are applied: Qual_T=1.0. Qual_C=1.0. GT.0: User defined quality factor in tension. Values between 0 and 1 are expected. Values greater than one are allowed, but may not be realistic.
QUAL_C	User defined quality factor in compression. This input value is used if Qual_T>0. Values between 0 and 1 are expected. Values greater than one are allowed, but may not be realistic. If left blank, a default value of Qual_C=Qual_T is used.
UNITS	Units options: EQ.0: GPa, mm, msec, Kg/mm^3, kN. EQ.1: MPa, mm, msec, g/mm^3, Nt. EQ.2: MPa, mm, sec, Mg/mm^3, Nt. EQ.3: Psi, inch, sec, lb-s^2/inch^4, lb
IQUAL	Apply quality factors perpendicular to the grain: EQ.0: Yes (default). EQ 1: No.
EL	Parallel normal modulus
ET	Perpendicular normal modulus.
GLT	Parallel shear modulus (GLT=GLR).

VARIABLE	DESCRIPTION
GTR	Perpendicular shear modulus.
PR	Parallel major Poisson's ratio.
XT	Parallel tensile strength.
XC	Parallel compressive strength.
YT	Perpendicular tensile strength.
YC	Perpendicular compressive strength.
SXY	Parallel shear strength.
SYZ	Perpendicular shear strength.
$GF1_{\parallel}$	Parallel fracture energy in tension.
$GF2_{\parallel}$	Parallel fracture energy in shear.
BFIT	Parallel softening parameter.
$DMAX_{\parallel}$	Parallel maximum damage.
$GF1_{\perp}$	Perpendicular fracture energy in tension.
$GF2_{\perp}$	Perpendicular fracture energy in shear.
DFIT	Perpendicular softening parameter.
$DMAX_{\perp}$	Perpendicular maximum damage.
FLPAR	Parallel fluidity parameter for tension and shear.
FLPARC	Parallel fluidity parameter for compression.
POWPAR	Parallel power.
FLPER	Perpendicular fluidity parameter for tension and shear.
FLPERC	Perpendicular fluidity parameter for compression.
POWPER	Perpendicular power.
NPAR	Parallel hardening initiation.
CPAR	Parallel hardening rate

VARIABLE	DESCRIPTION
NPER	Perpendicular hardening initiation.
CPER	Perpendicular hardening rate.
AOPT	Material axes option (see MAT_*OPTION* TROPIC_ELASTIC for a more complete description): EQ.0.0: locally orthotropic with material axes determined by element nodes as shown in Figure 2.1. Nodes 1, 2, and 4 of an element are identical to the nodes used for the definition of a coordinate system as by *DEFINE_COORDINATE_NODES. EQ.1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center; this is the a-direction. This option is for solid elements only. EQ.2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_ VECTOR. EQ.3.0: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle, BETA, from a line in the plane of the element defined by the cross product of the vector **v** with the element normal. The plane of a solid element is the midsurface between the inner surface and outer surface defined by the first four nodes and the last four nodes of the connectivity of the element, respectively. EQ.4.0: locally orthotropic in cylindrical coordinate system with the material axes determined by a vector **v**, and an originating point, P, which define the centerline axis. This option is for solid elements only. LT.0.0: the absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_NODES, *DEFINE_ COORDINATE_SYSTEM or *DEFINE_COORDINATE_ VECTOR). Available in R3 version of 971 and later.
MACF	Material axes change flag: EQ.1: No change, default, EQ.2: switch material axes a and b, EQ.3: switch material axes a and c, EQ.4: switch material axes b and c.
BETA	Material angle in degrees for AOPT = 3, may be overridden on the element card, see *ELEMENT_ SOLID_ORTHO.
XP YP ZP	Coordinates of point **p** for AOPT = 1 and 4.

VARIABLE	DESCRIPTION
A1 A2 A3	Components of vector **a** for AOPT = 2.
D1 D2 D3	Components of vector **d** for AOPT = 2.
V1 V2 V3	Define components of vector **v** for AOPT = 3 and 4.

Remarks:

Material property data is for clear wood (small samples without defects like knots), whereas real structures are composed of graded wood. Clear wood is stronger than graded wood. Quality factors (strength reduction factors) are applied to the clear wood strengths to account for reductions in strength as a function of grade. One quality factor (Qual_T) is applied to the tensile and shear strengths. A second quality factor (Qual_C) is applied to the compressive strengths. As a option, predefined quality factors are provided based on correlations between LS-DYNA calculations and test data for pine and fir posts impacted by bogie vehicles. By default, quality factors are applied to both the parallel and perpendicular to the grain strengths. An option is available (IQUAL) to eliminate application perpendicular to the grain.

*MAT_PITZER_CRUSHABLE_FOAM

This is Material Type 144. This model is for the simulation of isotropic crushable forms with strain rate effects. Uniaxial and triaxial test data have to be used. For the elastic response, the Poisson ratio is set to zero.

Card 1

Variable	MID	RO	K	G	PR	TY	SRTV	
Type	A8	F	F	F	F	F	F	

Card 2

Variable	LCPY	LCUYS	LCSR	VC	DFLG			
Type	I	I	I	F	F			

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
K	Bulk modulus.
G	Shear modulus
PR	Poisson's ratio
TY	Tension yield.
SRTV	Young's modulus (E)
LCPY	Load curve ID giving pressure versus volumetric strain, see Figure 75.1.
LCUYS	Load curve ID giving uniaxial stress versus volumetric strain, see Figure 75.1.
LCSR	Load curve ID giving strain rate scale factor versus volumetric strain rate.

VARIABLE	DESCRIPTION
VC	Viscous damping coefficient (.05 <recommended value < .50).
DFLG	Density flag: EQ.0.0: use initial density EQ.1.0: use current density (larger step size with less mass scaling).

Remarks:

The logarithmic volumetric strain is defined in terms of the relative volume, V, as:

$$\gamma = -\ln(V)$$

In defining the curves the stress and strain pairs should be positive values starting with a volumetric strain value of zero.

*MAT_SCHWER_MURRAY_CAP_MODEL

This is Material Type 145. The Schwer & Murray Cap Model, known as the Continuous Surface Cap Model, is a three invariant extension of the Geological Cap Model (Material Type 25) that also includes viscoplasticity for rate effects and damage mechanics to model strain softening. The primary references are Schwer and Murray [1994], Schwer [1994], and Murray and Lewis [1994]. The model is appropriate for geomaterials including soils, concrete, and rocks.

Warning: no default input parameter values are assumed, but recommendations for the more obscure parameters are provided in the descriptions that follow.

Card 1

Variable	MID	RO	SHEAR	BULK	GRUN	SHOCK	PORE	
Type	A8	F	F	F	F	F	F	

Card 2

Variable	ALPHA	THETA	GAMMA	BETA	EFIT	FFIT	ALPHAN	CALPHA
Type	F	F	F	F	F	F	F	F

Card 3

Variable	RO	XO	IROCK	SECP	AFIT	BFIT	RDAMO	
Type	F	F	F	F	F	F	F	

Card 4	1	2	3	4	5	6	7	8
Variable	W	D1	D2	NPLOT	EPSMAX	CFIT	DFIT	TFAIL
Type	F	F	F	F	F	F	F	F

Card 5

Variable	FAILFL	DBETA	DDELTA	VPTAU				
Type	F	F	F	F				

Card 6

Variable	ALPHA1	THETA1	GAMMA1	BETA1	ALPHA2	THETA2	GAMMA2	BETA2
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
SHEAR	Shear modulus, G
BULK	Bulk modulus, K
GRUN	Gruneisen ratio (typically = 0), Γ
SHOCK	Shock velocity parameter (typically 0), S_l
PORE	Flag for pore collapse EQ.0.0: for Pore collapse EQ.1.0: for Constant bulk modulus (typical)
ALPHA	Shear failure parameter, α
THETA	Shear failure parameter, θ

VARIABLE	DESCRIPTION
GAMMA	Shear failure parameter, γ
BETA	Shear failure parameter, β $\sqrt{J_2'} = F_e(J_1) = \alpha - \gamma \exp(-\beta J_1) + \theta J_1$
EFIT	Dilitation damage mechanics parameter (no damage = 1)
FFIT	Dilitation damage mechanics parameter (no damage = 0)
ALPHAN	Kinematic strain hardening parameter, N^α
CALPHAN	Kinematic strain hardening parameter, c^α
R0	Initial cap surface ellipticity, R
X0	Initial cap surface J_1 (mean stress) axis intercept, $X(\kappa_0)$
IROCK	EQ.0: soils (cap can contract) EQ.1: rock/concrete
SECP	Shear enhanced compaction
AFIT	Ductile damage mechanics parameter (=1 no damage)
BFIT	Ductile damage mechanics parameter (=0 no damage)
RDAM0	Ductile damage mechanics parameter
W	Plastic Volume Strain parameter, W
D1	Plastic Volume Strain parameter, D_1
D2	Plastic Volume Strain parameter, D_2 $\varepsilon_V^P = W\left(1 - \exp\left[-D_1\left(X(\kappa) - X(\kappa_0)\right) - D_2\left(X(\kappa) - X(\kappa_0)\right)^2\right]\right)$
NPLOT	History variable post-processed as effective plastic strain (See Table 145.1 for history variables available for plotting)
EPSMAX	Maximum permitted strain increment (default = 0) $\Delta\varepsilon_{\max} = 0.05(\alpha - N^\alpha - \gamma)\min\left(\frac{1}{G}, \frac{R}{9K}\right)$ (calculated default)
CFIT	Brittle damage mechanics parameter (=1 no damage)

VARIABLE	DESCRIPTION
DFIT	Brittle damage mechanics parameter (=0 no damage)
TFAIL	Tensile failure stress
FAILFL	Flag controlling element deletion and effect of damage on stress (see Remarks 1 and 2): EQ.1: σ_{ij} reduces with increasing damage; element is deleted when fully damaged (default) EQ.-1: σ_{ij} reduces with increasing damage; element is not deleted EQ.2: S_{ij} reduces with increasing damage; element is deleted when fully damaged EQ.-2: S_{ij} reduces with increasing damage; element is not deleted
DBETA	Rounded vertices parameter, $\Delta\beta_0$
DDELTA	Rounded vertices parameter, δ
VPTAU	Viscoplasticity relaxation time parameter, τ
ALPHA1	Torsion scaling parameter, α_1 $\alpha_1 < 0 \rightarrow \lvert\alpha_1\rvert =$ Friction Angle (degrees)
THETA1	Torsion scaling parameter, θ_1
GAMMA1	Torsion scaling parameter, γ_1
BETA1	Torsion scaling parameter, β_1 $Q_1 = \alpha_1 - \gamma_1 \exp(-\beta_1 J_1) + \theta_1 J_1$
ALPHA2	Tri-axial extension scaling parameter, α_2
THETA2	Tri-axial extension scaling parameter, θ_2
GAMMA2	Tri-axial extension scaling parameter, γ_2
BETA2	Tri-axial extension scaling parameter, β_2 $Q_2 = \alpha_2 - \gamma_2 \exp(-\beta_2 J_1) + \theta_2 J_1$

Remarks:

1. FAILFL controls whether the damage accumulation applies to either the total stress tensor σ_{ij} or the deviatoric stress tensor S_{ij}. When FAILFL = 2, damage does not diminish the ability of the material to support hydrostatic stress.

2. FAILFL also serves as a flag to control element deletion. Fully damaged elements are deleted only if FAILFL is a positive value. When MAT_145 is used with the ALE or EFG solvers, failed elements should not be eroded and so a negative value of FAILFL should be used.

Output History Variables

All the output parameters listed in Table 145.1 is available for post-processing using LS-PrePost and its displayed list of History Variables. The LS-DYNA input parameter NEIPH should be set to 26; see for example the keyword input for *DATABASE_EXTENT_BINARY.

Table 145.1. Output variables for post-processing using NPLOT parameter.

NPLOT	Function	Description
1	$X(\kappa)$	J_1 intercept of cap surface
2	$L(\kappa)$	J_1 value at cap-shear surface intercept
3	R	Cap surface ellipticity
4	$\tilde{R}$	Rubin function
5	ε_v^p	Plastic volume strain
6		Yield Flag (= 0 elastic)
7		Number of strain sub-increments
8	G^α	Kinematic hardening parameter
9	J_2^α	Kinematic hardening back stress
10		Effective strain rate
11		Ductile damage
12		Ductile damage threshold
13		Strain energy
14		Brittle damage
15		Brittle damage threshold
16		Brittle energy norm
17		J_1 (w/o visco-damage/plastic)
18		J_2' (w/o visco-damage/plastic)
19		J_3' (w/o visco-damage/plastic)
20		$\hat{J}_3$(w/o visco-damage/plastic)
21	β	Lode Angle
22	d	Maximum damage parameter
23		future variable
24		future variable
25		future variable
26		future variable

Sample Input for Concrete

Gran and Senseny [1996] report the axial stress versus strain response for twelve unconfined compression tests of concrete, used in scale-model reinforced-concrete wall tests. The Schwer & Murray Cap Model parameters provided below were used, see Schwer [2001], to model the unconfined compression test stress-strain response for the nominal 40 MPa strength concrete reported by Gran and Senseny. The basic units for the provided parameters are length in millimeters (mm), time in milliseconds (msec), and mass in grams (g). This base unit set yields units of force in Newtons (N) and pressure in Mega-Pascals (MPa).

Keyword:

Card 1

Variable	MID	RO	SHEAR	BULK	GRUN	SHOCK	PORE	
Type	A8	2.3E-3	1.048E4	1.168E4	0.0	0.0	1.	

Card 2

Variable	ALPHA	THETA	GAMMA	BETA	EFIT	FFIT	ALPHAN	CALPHA
Type	190.0	0.0	184.2	2.5E-3	0.999	0.7	2.5	2.5E3

Card 3

Variable	R0	X0	IROCK	SECP	AFIT	BFIT	RDAM0	
Type	5.0	100.0	1.0	0.0	0.999	0.3	0.94	

Card 4

Variable	W	D1	D2	NPLOT	EPSMAX	CFIT	DFIT	TFAIL
Type	5.0E-2	2.5E-4	3.5E-7	23.0	0.0	1.0	300.0	7.0

Card 5

Variable	FAILFG	DBETA	DDELTA	VPTAU				
Type	1.0	0.0	0.0	0.0				

Card 6

Variable	ALPHA1	THETA1	GAMMA1	BETA1	ALPHA2	THETA2	GAMMA2	BETA2
Type	0.747	3.3E-4	0.17	5.0E-2	0.66	4.0E-4	0.16	5.0E-2

User Input Parameters and System of Units

Consider the following basic units:

Length - L (e.g. millimeters - mm)
Mass - M (e.g. grams - g)
Time - T (e.g. milliseconds - ms)

The following consistent unit systems can then be derived using Newton's Law, i.e. $F = Ma$.

Force - $F = ML/T^2$ [g-mm/ms^2 = Kg-m/s^2 = Newton - N]

and

Stress - $\sigma = F/L^2$ [N/mm^2 = 10^6 N/m^2 = 10^6 Pascals = MPa]

and

Density - $\rho = M/L^3$ [g/mm^3 = 10^6 Kg/m^3]

User Inputs and Units:

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	SHEAR	BULK	GRUN	SHOCK	PORE	
Units	I	Density M/L^3	Stress F/L^2	Stress F/L^2				

Card 2	1	2	3	4	5	6	7	8
Variable	ALPHA	THETA	GAMMA	BETA	EFIT	FFIT	ALPHAN	CALPHA
Units	Stress F/L^2		Stress F/L^2	$\left(\frac{\text{Stress}}{F/L^2}\right)^{-1}$		$\left(\frac{\text{Stress}}{F/L^2}\right)^{-1/2}$	Stress F/L^2	Stress F/L^2

Card 3	1	2	3	4	5	6	7	8
Variable	R0	X0	IROCK	SECP	AFIT	BFIT	RDAM0	
Units		Stress F/L^2				$\left(\frac{\text{Stress}}{F/L^2}\right)^{-1/2}$	$\left(\frac{\text{Stress}}{F/L^2}\right)^{1/2}$	

Card 4	1	2	3	4	5	6	7	8
Variable	W	D1	D2	NPLOT	MAXEPS	CFIT	DFIT	TFAIL
Units		$\left(\frac{\text{Stress}}{F/L^2}\right)^{-1}$	$\left(\frac{\text{Stress}}{F/L^2}\right)^{-2}$				$\left(\frac{\text{Stress}}{F/L^2}\right)^{-1/2}$	Stress F/L^2

Card 5	1	2	3	4	5	6	7	8
Variable	FAILFG	DBETA	DDELTA	VPTAU				
Units		Angle degrees		Time T				

Card 6	1	2	3	4	5	6	7	8
Variable	ALPHA1	THETA1	GAMMA1	BETA1	ALPHA2	THETA2	GAMMA2	BETA2
Units	Stress F/L^2		Stress F/L^2	$\left(\frac{\text{Stress}}{F/L^2}\right)^{-1}$	Stress F/L^2		Stress F/L^2	$\left(\frac{\text{Stress}}{F/L^2}\right)^{-1}$

*MAT_1DOF_GENERALIZED_SPRING

This is Material Type 146. This is a linear spring or damper that allows different degrees-of-freedom at two nodes to be coupled.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	K	C	SCLN1	SCLN2	DOFN1	DOFN2
Type	A8	F	F	F	F	F	I	I

Card 2

Variable	CID1	CID2						
Type	I	I						

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density, see also volume in *SECTION_BEAM definition.
K	Spring stiffness.
C	Damping constant.
SCLN1	Scale factor on force at node 1. Default=1.0.
SCLN2	Scale factor on force at node 2. Default=1.0.
DOFN1	Active degree-of-freedom at node 1, a number between 1 to 6 where 1 in x-translation and 4 is x-rotation. If this parameter is defined in the SECTION_BEAM definition or on the ELEMENT_BEAM_SCALAR card, then the value here, if defined, is ignored.
DOFN2	Active degree-of-freedom at node 2, a number between 1 to 6. If this parameter is defined in the SECTION_BEAM definition or on the ELEMENT_BEAM_SCALAR card, then the value here, if defined, is ignored.

VARIABLE	DESCRIPTION
CID1	Local coordinate system at node 1. This coordinate system can be overwritten by a local system specified on the *ELEMENT_BEAM _SCALAR or *SECTION_BEAM keyword input. If no coordinate system is specified, the global system is used.
CID2	Local coordinate system at node 2. If CID2=0, CID2=CID1.

*MAT_FHWA_SOIL

This is Material Type 147. This is an isotropic material with damage and is available for solid elements. The model has a modified Mohr-Coulomb surface to determine the pressure dependent peak shear strength. It was developed for applications involving roadbase soils by Lewis [1999] for the FHWA, who extended the work of Abbo and Sloan [1995] to include excess pore water effects.

Card 1

Variable	MID	RO	NPLOT	SPGRAV	RHOWAT	VN	GAMMAR	INTRMX
Type	A8	F	I	F	F	F	F	I
Default	none	none	1	none	1.0	0.0	0.0	1

Card 2

Variable	K	G	PHIMAX	AHYP	COH	ECCEN	AN	ET
Type	F	F	F	F	F	F		
Default	none	none	none	none	none	none	none	none

Card 3

Variable	MCONT	PWD1	PWKSK	PWD2	PHIRES	DINT	VDFM	DAMLEV
Type	F	F	F	F	F	F	F	F
Default	none	none	none	none	0.0	none	none	none

Card 4	1	2	3	4	5	6	7	8
Variable	EPSMAX							
Type	F							
Default	none							

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
NPLOT	Controls what is written as component 7 to the d3plot database. LS-Prepost always blindly labels this component as effective plastic strain. EQ.1: Effective Strain EQ.2: Damage Criterion Threshold EQ.3: Damage (diso) EQ.4: Current Damage Criterion EQ.5: Pore Water Pressure EQ.6: Current Friction Angle (phi)
SPGRAV	Specific Gravity of Soil used to get porosity.
RHOWAT	Density of water in model units - used to determine air void strain (saturation)
VN	Viscoplasticity parameter (strain-rate enhanced strength)
GAMMAR	Viscoplasticity parameter (strain-rate enhanced strength)
ITERMAX	Maximum number of plasticity iterations (default 1)
K	Bulk Modulus (non-zero)
G	Shear modulus (non-zero)
PHIMAX	Peak Shear Strength Angle (friction angle) (radians)
AHYP	Coefficient A for modified Drucker-Prager Surface
COH	Cohesion ñ Shear Strength at zero confinement (overburden)

VARIABLE	DESCRIPTION
ECCEN	Eccentricity parameter for third invariant effects
AN	Strain hardening percent of phi max where non-linear effects start
ET	Strain Hardening Amount of non-linear effects
MCONT	Moisture Content of Soil (Determines amount of air voids) (0-1.00)
PWD1	Parameter for pore water effects on bulk modulus
PWKSK	Skeleton bulk modulus- Pore water parameter ñ set to zero to eliminate effects
PWD2	Parameter for pore water effects on the effective pressure (confinement)
PHIRES	The minimum internal friction angle, radians (residual shear strength)
DINT	Volumetric Strain at Initial damage threshold, EMBED Equation.3
VDFM	Void formation energy (like fracture energy)
DAMLEV	Level of damage that will cause element deletion (0.0-1.0)
EPSMAX	Maximum principle failure strain

*MAT_FHWA_SOIL_NEBRASKA

This is an option to use the default properties determined for soils used at the University of Nebraska (Lincoln). The default units used for this material are millimeter, millisecond, and kilograms. If different units are desired, the conversion factors must be input.

This is Material Type 147. This is an isotropic material with damage and is available for solid elements. The model has a modified Mohr-Coulomb surface to determine the pressure dependent peak shear strength. It was developed for applications involving road base soils.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	FCTIM	FCTMAS	FCTLEN				
Type	A8	F	I	F	F	F	F	I
Default	none	none	1	none	1.0	0.0	0.0	1

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
FCTIM	Factor to multiply milliseconds by to get desired time units
FCTMAS	Factor to multiply kilograms by to get desired mass units
FCTLEN	Factor to multiply millimeters by to get desired length units

*MAT_GAS_MIXTURE

This is Material Type 148. This model is for the simulation of thermally equilibrated ideal gas mixtures. This only works with the multi-material ALE formulation (ELFORM=11 in *SECTION_SOLID). This keyword needs to be used together with *INITIAL_GAS_MIXTURE for the initialization of gas densities and temperatures. When applied in the context of ALE airbag modeling, the injection of inflator gas is done with a *SECTION_POINT_SOURCE_ MIXTURE command which controls the injection process. This material model type also has its name start with *MAT_ALE_. For example, an identical material model to this is *MAT_ALE_GAS_MIXTURE (or also, *MAT_ALE_03).

Card 1

Card 1	1	2	3	4	5	6	7	8
Variable	MID	IADIAB	RUNIV					
Type	A8	I	F					
Default	none	0	0.0					
Remark		5	1					

Card 2: Method (A) RUNIV=BLANK or 0.0 ➔ Per-mass unit is used

Card 2	1	2	3	4	5	6	7	8
Variable	CVmass1	CVmass2	CVmass3	CVmass4	CVmass5	CVmass6	CVmass7	CVmass8
Type	F	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	none	none
Remark								

Card 3: Method (A) RUNIV=BLANK or 0.0 ➔ Per-mass unit is used

Card 3	1	2	3	4	5	6	7	8
Variable	CPmass1	CPmass 2	CPmass 3	CPmass 4	CPmass 5	CPmass6	CPmass 7	CPmass 8
Type	F	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	none	none
Remark								

Card 2: Method (B) RUNIV is nonzero

Card 2	1	2	3	4	5	6	7	8
Variable	MOLWT1	MOLWT 2	MOLWT 3	MOLWT 4	MOLWT 5	MOLWT 6	MOLWT 7	MOLWT 8
Type	F	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	none	none
Remark	2							

Card 3: Method (B) RUNIV is nonzero ➔ Per-mole unit is used

Card 3	1	2	3	4	5	6	7	8
Variable	CPmole1	CPmole2	CPmole3	CPmole4	CPmole5	CPmole6	CPmole7	CPmole8
Type	F	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	none	none
Remark	2							

Card 4: Method (B) RUNIV is nonzero

Card 4	1	2	3	4	5	6	7	8
Variable	B1	B2	B3	B4	B5	B6	B7	B8
Type	F	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	none	none
Remark	2							

Card 5: Method (B) RUNIV is nonzero

Card 5	1	2	3	4	5	6	7	8
Variable	C1	C2	C3	C4	C5	C6	C7	C8
Type	F	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	none	none
Remark	2							

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
IADIAB	This flag (default=0) is used to turn ON/OFF adiabatic compression logics for an ideal gas (remark 5). EQ.0: OFF (default) EQ.1: ON
RUNIV	Universal gas constant in per-mole unit (8.31447 J/(mole*K)).
CVmass1-CVmass8	If RUNIV is BLANK or zero (method A): Heat capacity at constant volume for up to eight different gases in per-mass unit.
CPmass1-CPmass8	If RUNIV is BLANK or zero (method A): Heat capacity at constant pressure for up to eight different gases in per-mass unit.

VARIABLE	DESCRIPTION
MOLWT1-MOLWT8	If RUNIV is nonzero (method B): Molecular weight of each ideal gas in the mixture (mass-unit/mole).
CPmole1-CPmole8	If RUNIV is nonzero (method B): Heat capacity at constant pressure for up to eight different gases in per-mole unit. These are nominal heat capacity values typically at STP. These are denoted by the variable "A" in the equation in remark 2.
B1-B8	If RUNIV is nonzero (method B): First order coefficient for a temperature dependent heat capacity at constant pressure for up to eight different gases. These are denoted by the variable "B" in the equation in remark 2.
C1-C8	If RUNIV is nonzero (method B): Second order coefficient for a temperature dependent heat capacity at constant pressure for up to eight different gases. These are denoted by the variable "C" in the equation in remark 2.

Remarks:

1. There are 2 methods of defining the gas properties for the mixture. If RUNIV is BLANK or ZERO ➔ Method (A) is used to define constant heat capacities where per-mass unit values of C_v and C_p are input. Only cards 2 and 3 are required for this method. Method (B) is used to define constant or temperature dependent heat capacities where per-mole unit values of C_p are input. Cards 2-5 are required for this method.

2. The per-mass-unit, temperature-dependent, constant-pressure heat capacity is

$$C_p(T) = \frac{[A + B*T + C*T^2]}{MW} \sim \frac{J}{kg*K} \qquad B \sim J/(mole*K^2)$$

$$A = \tilde{C}_{P0} \sim J/(mole*K) \qquad C \sim J/(mole*K^3)$$

   The units shown are only for demonstration of the equation.

3. The initial temperature and the density of the gas species present in a mesh or part at time zero is specified by the keyword *INITIAL_GAS_MIXTURE.

4. The ideal gas mixture is assumed to be thermal equilibrium, that is, all species are at the same temperature (T). The gases in the mixture are also assumed to follow Dalton's Partial Pressure Law, $P = \sum_i^{ngas} P_i$. The partial pressure of each gas is then $P_i = \rho_i R_{gas_i} T$ where $R_{gasi} = \frac{R_{univ}}{MW}$. The individual gas species temperature equals the mixture temperature. The temperature is computed from the internal energy where the *mixture internal energy per unit volume* is used,

$$e_V = \sum_i^{ngas} \rho_i C_{V_i} T_i = \sum_i^{ngas} \rho_i C_{V_i} T$$

$$T = T_i = \frac{e_V}{\sum_i^{ngas} \rho_i C_{V_i}} \quad .$$

In general, the advection step conserves momentum and internal energy, but not kinetic energy. This can result in energy lost in the system and lead to a pressure drop. In *MAT_GAS_MIXTURE the dissipated kinetic energy is automatically converted into heat (internal energy). Thus in effect the total energy is conserved instead of conserving just the internal energy. This numerical scheme has been shown to improve accuracy in some cases. However, the user should always be vigilant and check the physics of the problem closely.

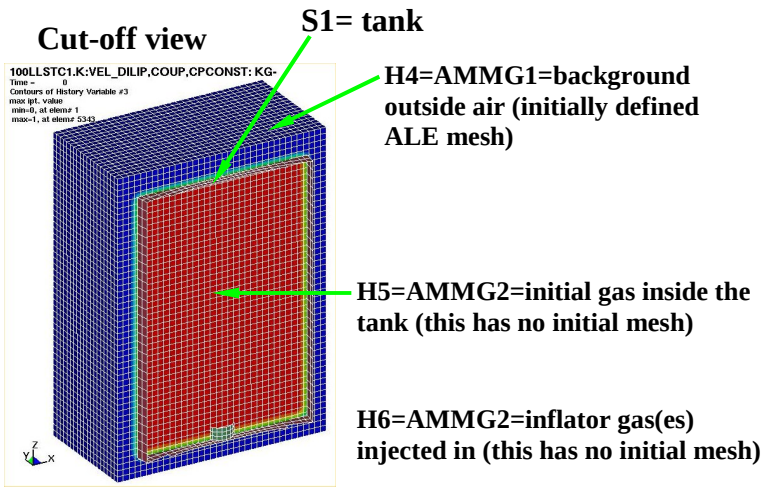
5. As an example consider an airbag surrounded by ambient air. As the inflator gas flows into the bag, the ALE elements cut by the airbag fabric shell elements will contain some inflator gas inside and some ambient air outside. The multi-material element treatment is not perfect. Consequently the temperature of the outside air may be made artificially high after the multi-material element treatment. To prevent the outside ambient air from getting artificially high T, set IDIAB=1 for the ambient air outside. Simple adiabatic compression equation is then assumed for the outside air. The use of this flag may be needed, but only when that air is modeled by the *MAT_GAS_MIXTURE card.

Example:

Consider a tank test model where the Lagrangian tank (Part S1) is surrounded by an ALE air mesh (Part H4=AMMGID 1). There are 2 ALE parts which are defined but initially have no corresponding mesh: part 5 (H5=AMMGID 2) is the resident gas inside the tank at t = 0, and part 6 (H6=AMMGID 2) is the inflator gas(es) which is injected into the tank when t > 0. AMMGID stands for ALE Multi-Material Group ID. Please see figure and input below. The *MAT_GAS_MIXTURE (MGM) card defines the gas properties of ALE parts H5 & H6. The MGM card input for both method (A) and (B) are shown.

The *INITIAL_GAS_MIXTURE card is also shown. It basically specifies that "AMMGID 2 may be present in part or mesh H4 at t=0, and the initial density of this gas is defined in the rho1 position which corresponds to the 1st material in the mixture (or H5, the resident gas)."

Example configuration:



Sample input:

```
$---------------------------------------------------------------------------
*PART
H5 = initial gas inside the tank
$      PID     SECID       MID      EOSID      HGID      GRAV    ADPOPT      TMID
         5         5         5          0         5         0         0
*SECTION_SOLID
         5        11         0
$---------------------------------------------------------------------------
$ Example 1:  Constant heat capacities using per-mass unit.
$*MAT_GAS_MIXTURE
$      MID    IADIAB     R_univ
$        5         0          0
$  Cv1_mas   Cv2_mas   Cv3_mas   Cv4_mas   Cv5_mas   Cv6_mas   Cv7_mas   Cv8_mas
$718.7828911237.56228
$  Cp1_mas   Cp2_mas   Cp3_mas   Cp4_mas   Cp5_mas   Cp6_mas   Cp7_mas   Cp8_mas
$1007.00058 1606.1117
$---------------------------------------------------------------------------
$ Example 2:  Variable heat capacities using per-mole unit.
*MAT_GAS_MIXTURE
$      MID    IADIAB     R_univ
         5         0  8.314470
$      MW1       MW2       MW3       MW4       MW5       MW6       MW7       MW8
 0.0288479   0.02256
$  Cp1_mol   Cp2_mol   Cp3_mol   Cp4_mol   Cp5_mol   Cp6_mol   Cp7_mol   Cp8_mol
 29.049852  36.23388
$       B1        B2        B3        B4        B5        B6        B7        B8
  7.056E-3  0.132E-1
$       C1        C2        C3        C4        C5        C6        C7        C8
 -1.225E-6 -0.190E-5
$---------------------------------------------------------------------------
$ One card is defined for each AMMG that will occupy some elements of a mesh set
*INITIAL_GAS_MIXTURE
$      SID     STYPE     MMGID        T0
         4         1         1    298.15
$     RHO1      RHO2      RHO3      RHO4      RHO5      RHO6      RHO7      RHO8
1.17913E-9
*INITIAL_GAS_MIXTURE
$      SID     STYPE     MMGID        T0
         4         1         2    298.15
$     RHO1      RHO2      RHO3      RHO4      RHO5      RHO6      RHO7      RHO8
1.17913E-9
$---------------------------------------------------------------------------
```

*MAT_EMMI

This is Material Type 151. The Evolving Microstructural Model of Inelasticity (EMMI) is a temperature and rate-dependent state variable model developed to represent the large deformation of metals under diverse loading conditions [Marin 2005].

Card 1

Variable	MID	RHO	E	PR				
Type	A8	F	F	F				
Default	-	-	-	-	-	-	-	

Card 2

Variable	RGAS	BVECT	D0	QD	CV	ADRAG	BDRAG	DMTHETA
Type	F	F	F	F	F	F	F	F
Default	-	-	-	-	-	-	-	-

Card 3

Variable	DMPHI	DNTHETA	DNPHI	THETA0	THETAM	BETA0	BTHETA	DMR
Type	F	F	F	F	F	F	F	F
Default	-	-	-	-	-	-	-	-

Card 4

Variable	DM1	DM2	DM3	DM4	DM5	Q1ND	Q2ND	Q3ND
Type	I	F	F	F	F	F	F	
Default	-	-	-	-	-	-	-	

Card 5

Variable	Q4ND	CALPHA	CKAPPA	C1	C2ND	C3	C4	C5
Type	F	F	F	F	F	F	F	F
Default	-	-	-	-	-	-	-	-

Card 6

Variable	C6	C7ND	C8ND	C9ND	C10	A1	A2	A3
Type	F	F	F	F	F	F	F	F
Default	-	-	-	-	-	-	-	-

Card 7

Variable	A4	A_XX	A_YY	A_ZZ	A_XY	A_YZ	A_XZ	ALPHXX
Type	F	F	F	F	F	F	F	F
Default	-	-	-	-	-	-	-	

Card 8

Variable	ALPHYY	ALPHZZ	ALPHXY	ALPHYZ	ALPHXZ	DKAPPA	PHI0	DLBDAG
Type	F	F	F	F	F	F	F	F
Default	-	-	-	-	-	-	-	-

Card 9

Variable	FACTOR	DMGOPT	DELASO	DIMPLO	ATOL	RTOL	DNITER	XTRA1
Type	F	F	F	F	F	F	F	F
Default	-	-	-	-	-	-	-	-

Card 10	1	2	3	4	5	6	7	8
Variable	XTRA2	XTRA3	XTRA4	XTRA5	XTRA6	XTRA7	XTRA8	XTRA9
Type	I	F	F	F	F	F	F	
Default	-	-	-	-	-	-	-	

Card 11

Variable	XTRA10	XTRA11	XTRA12	XTRA13	XTRA14	XTRA15	XTRA16	XTRA17
Type	F	F	F	F	F	F	F	F
Default	-	-	-	-	-	-	-	-

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RHO	Material density.
E	Young's modulus
PR	Poisson's ratio
RGAS	universal gas constant.
BVECT	Burger's vector
D0	pre-exponential diffusivity coefficient
QD	activation energy
CV	specific heat at constant volume
ADRAG	drag intercept
BDRAG	drag coefficient

VARIABLE	DESCRIPTION
DMTHETA	shear modulus temperature coefficient
DMPHI	shear modulus damage coefficient
DNTHETA	bulk modulus temperature coefficient
DNPHI	bulk modulus damage coefficient
THETA0	reference temperature
THETAM	melt temperature
BETA0	coefficient of thermal expansion at reference temperature
BTHETA	thermal expansion temperature coefficient
DMR	damage rate sensitivity parameter
DM1	coefficient of yield temperature dependence
DM2	coefficient of yield temperature dependence
DM3	coefficient of yield temperature dependence
DM4	coefficient of yield temperature dependence
DM5	coefficient of yield temperature dependence
Q1ND	dimensionless activation energy for f
Q2ND	dimensionless activation energy for rd
Q3ND	dimensionless activation energy for Rd
Q4ND	dimensionless activation energy Rs
CALPHA	coefficient for backstress alpha
CKAPPA	coefficient for internal stress kappa
C1	parameter for flow rule exponent n
C2ND	parameter for transition rate f
C3	parameter for alpha dynamic recovery rd
C4	parameter for alpha hardening h
C5	parameter for kappa dynamic recovery Rd

VARIABLE	DESCRIPTION
C6	parameter for kappa hardening H
C7ND	parameter kappa static recovery Rs
C8ND	parameter for yield
C9ND	parameter for temperature dependence of flow rule exponent n
C10	parameter for static recovery (set=1)
A1	plastic anisotropy parameter
A2	plastic anisotropy parameter
A3	plastic anisotropy parameter
A4	plastic anisotropy parameter
A_XX	initial structure tensor component
A_YY	initial structure tensor component
A_ZZ	initial structure tensor component
A_XY	initial structure tensor component
A_YZ	initial structure tensor component
A_XZ	initial structure tensor component
ALPHXX	initial backstress component
ALPHYY	initial backstress component
ALPHZZ	initial backstress component
ALPHXY	initial backstress component
ALPHYZ	initial backstress component
ALPHXZ	initial backstress component
DKAPPA	initial isotropic internal stress
PHI0	initial isotropic porosity
DLBDAG	slip system geometry parameter
FACTOR	fraction of plastic work converted to heat, adiabatic

VARIABLE	DESCRIPTION
DMGOPT	Damage model option parameter EQ.1.0: pressure independent Cocks/Ashby 1980 EQ.2.0: pressure dependent Cocks/Ashby 1980 EQ.3.0: pressure dependent Cocks 1989
DELASO	
DIMPLO	Implementation option flag EQ.1.0: combined viscous drag and thermally activated dislocation motion EQ.2.0: separate viscous drag and thermally activated dislocation motion
ATOL	absolute error tolerance for local Newton iteration
RTOL	relative error tolerance for local Newton iteration
DNITER	maximum number of iterations for local Newton iteration
XTRA1	
XTRA2	
XTRA3	
XTRA4	
XTRA5	
XTRA6	
XTRA7	
XTRA8	
XTRA9	
XTRA10	
XTRA11	
XTRA12	
XTRA13	
XTRA14	
XTRA15	
XTRA16	
XTRA17	

Remarks:

$$\overset{\nabla}{\boldsymbol{\alpha}} = h\,\boldsymbol{d}^p - r_d\,\dot{\bar{\varepsilon}}^p\,\bar{\alpha}\,\boldsymbol{\alpha}$$

$$\dot{\kappa} = \left(H - R_d\kappa\right)\dot{\bar{\varepsilon}}^p - R_s\kappa\sinh\left(Q_s\kappa\right)$$

$$\boldsymbol{d}^p = \sqrt{\frac{3}{2}}\,\dot{\bar{\varepsilon}}^p\boldsymbol{n},\;\; \dot{\bar{\varepsilon}}^p = f\sinh^n\left[\left\langle\frac{\bar{\sigma}}{\kappa+Y}-1\right\rangle\right]$$

Table 151.1. Plasticity Material Functions of EMMI Model.

$\dot{\bar{\varepsilon}}^p$ – equation	$\boldsymbol{\alpha}$ – equation	κ – equation
$f = c_2\exp\left(\frac{Q_1}{\theta}\right)$	$r_d = c_3\exp\left(\frac{-Q_2}{\theta}\right)$	$R_d = c_5\exp\left(\frac{-Q_3}{\theta}\right)$
$n = \frac{c_9}{\theta} - c_1$	$h = c_4\hat{\mu}(\theta)$	$H = c_6\hat{\mu}(\theta)$
$Y = c_8\hat{Y}(\theta)$		$R_s = c_7\exp\left(\frac{-Q_4}{\theta}\right)$
		$Q_s = c_{10}\exp\left(\frac{-Q_5}{\theta}\right)$

Void growth:

$$\dot{\varphi} = \frac{3}{\sqrt{2}}(1-\varphi)\hat{G}\left(\bar{\sigma}_{eq},\bar{p},\varphi\right)\dot{\bar{\varepsilon}}^p$$

$$\hat{G}\left(\bar{\sigma}_{eq},\bar{p}_\tau,\varphi\right) = \frac{3}{\sqrt{3}}\left[\frac{1}{(1-\varphi)m+1}-1\right]\sinh\left[\frac{2(2m-1)}{2m+1}\frac{\langle\bar{p}\rangle}{\bar{\sigma}_{eq}}\right]$$

*MAT_DAMAGE_3

This is Material Type 153. This model has two back stress terms for kinematic hardening combined with isotropic hardening and a damage model for modeling low cycle fatigue and failure. Huang [2006] programmed this model and provided it as a user subroutine with the documentation that follows. It is available for beam, shell and solid elements. This material model is available starting with the R3 release of Version 971.

Card 1

Variable	MID	RO	E	PR	SIGY	HARDI	BETA	LCSS
Type	A8	F	F	F	F	F	F	I

Card 2

Variable	HARDK1	GAMMA1	HARDK2	GAMMA2	SRC	SRP		
Type	F	F	F	F	F	F		

Card 3

Variable	IDAMAGE	IDS	IDEP	EPSD	S	T	DC	
Type	I	I	I	F	F	F	F	

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density, ρ
E	Young's modulus, E
PR	Poisson's ratio, ν
SIGY	Initial yield stress, σ_{y0}
HARDI	Isotropic hardening modulus, H

VARIABLE	DESCRIPTION
BETA	Isotropic hardening parameter, β. Set $\beta = 0$ for linear isotropic hardening.
LCSS	Load curve ID defining effective stress vs. effective plastic strain
HARDK1	Kinematic hardening modulus C_1
GAMMA1	Kinematic hardening parameter γ_1
HARDK2	Kinematic hardening modulus C_2
GAMMA2	Kinematic hardening parameter γ_2
SRC	Strain rate parameter, C, for Cowper Symonds strain rate model, see below. If zero, rate effects are not considered.
SRP	Strain rate parameter, P, for Cowper Symonds strain rate model, see below. If zero, rate effects are not considered.
IDAMAGE	Isotropic damage flag EQ. 0: damage is inactivated EQ. 1: damage is activated
IDS	Output stress flag EQ. 0: undamaged stress is $\tilde{\sigma}$ output EQ. 1: damaged stress is $\tilde{\sigma}(1-D)$ output
IDEP	Damaged plastic strain EQ. 0: plastic strain is accumulated $r = \int \dot{\bar{\varepsilon}}^{pl}$ EQ. 1: damaged plastic strain is accumulated $r = \int (1-D)\dot{\bar{\varepsilon}}^{pl}$
EPSD	Damage threshold r_d. Damage accumulation begins when $r > r_d$
S	Damage material constant *S*. Default = $\sigma_{y0}/200$
T	Damage material constant *t*. Default = 1
DC	Critical damage value D_c. When damage value reaches critical, the element is deleted from calculation. Default = 0.5

Remarks:

This model is based on the work of Lemaitre [1992], and Dufailly and Lemaitre [1995]. It is a pressure-independent plasticity model with the yield surface defined by the function

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

where σ_v is uniaxial yield stress

$$\sigma_y = \sigma_{y0} + \frac{H}{\beta}\left[1 - \exp\left(-\beta r\right)\right]$$

By setting $\beta = 0$, a linear isotropic hardening is obtained

$$\sigma_y = \sigma_{y0} + Hr$$

where σ_{v0} s the initial yield stress. And $\bar{\sigma}$ is the equivalent von Mises stress, with respect to the deviatoric effective stress

$$\boldsymbol{s}_e = dev\left[\tilde{\boldsymbol{\sigma}}\right] - \boldsymbol{\alpha} = \boldsymbol{s} - \boldsymbol{\alpha}$$

where $\boldsymbol{s}$ is deviatoric stress and $\boldsymbol{\alpha}$ is the back stress, which is decomposed into several components

$$\boldsymbol{\alpha} = \sum_j \boldsymbol{\alpha}_j$$

and $\tilde{\boldsymbol{\sigma}}$ is effective stress (undamaged stress), based on Continuum Damage Mechanics model [Lemaitre 1992]

$$\tilde{\boldsymbol{\sigma}} = \frac{\boldsymbol{\sigma}}{1 - D}$$

where *D* is the isotropic damage scalar, which is bounded by 0 and 1

$$0 \le D \le 1$$

D = 0 represents a damage-free material RVE (representative Volume Element), while *D* = 1 represents a fully broken material RVE in two parts. In fact, fracture occurs when $D = D_c < 1$, modeled as element removal. The evolution of the isotropic damage value related to ductile damage and fracture (the case where the plastic strain or dissipation is much larger than the elastic one, [Lemaitre 1992]) is defined as

$$\dot{D} = \begin{cases} \left(\dfrac{Y}{S}\right)^t \dot{\bar{\varepsilon}}^{pl} & r > r_d \;\&\; \dfrac{\sigma_m}{\sigma_{eq}} > -\dfrac{1}{3} \\ 0 & otherwise \end{cases}$$

where $\frac{\sigma_m}{\sigma_{eq}}$ is the stress triaxiality, r_d is damage threshold, *S* is a material constant, and *Y* is strain energy release rate.

$$Y = \boldsymbol{\varepsilon}^{el} : \mathbf{D}^{el} : \boldsymbol{\varepsilon}^{el}$$

Where $\mathbf{D}^{el}$ represents the fourth-order elasticity tensor, $\boldsymbol{\varepsilon}^{el}$ is elastic strain. And t is a material constant, introduced by Dufailly and Lemaitre [1995], to provide additional degree of freedom for modeling low-cycle fatigue ($t = 1$ in Lemaitre [1992]). Dufailly and Lemaitre [1995] also proposed a simplified method to fit experimental results and get S and t.

The equivalent Mises stress is defined as

$$\bar{\sigma}(\boldsymbol{s}_e) = \sqrt{\frac{3}{2}\boldsymbol{s}_e : \boldsymbol{s}_e} = \sqrt{\frac{3}{2}}\|\boldsymbol{s}_e\|$$

The model assumes associated plastic flow

$$\dot{\boldsymbol{\varepsilon}}^{pl} = \frac{\partial F}{\partial \boldsymbol{\sigma}} d\lambda = \frac{3}{2}\frac{\boldsymbol{s}_e}{\bar{\sigma}} d\lambda$$

Where $d\lambda$ is the plastic consistency parameter. The evolution of the kinematic component of the model is defined as [Armstrong and Frederick 1966]:

$$\begin{cases} \dot{\boldsymbol{\alpha}}_j = \frac{2}{3} C_j \dot{\boldsymbol{\varepsilon}}^{pl} - \gamma_j \boldsymbol{\alpha}_j \dot{\bar{\varepsilon}}^{pl} & \text{IDEP=0} \\ \dot{\boldsymbol{\alpha}}_j = (1-D)\left(\frac{2}{3} C_j \dot{\boldsymbol{\varepsilon}}^{pl} - \gamma_j \boldsymbol{\alpha}_j \dot{\bar{\varepsilon}}^{pl}\right) & \text{IDEP=1} \end{cases}$$

The damaged plastic strain is accumulated as

$$\begin{cases} r = \int \dot{\bar{\varepsilon}}^{pl} & \text{IDEP=0} \\ r = \int (1-D)\,\dot{\bar{\varepsilon}}^{pl} & \text{IDEP=1} \end{cases}$$

where $\dot{\bar{\varepsilon}}^{pl}$ is the equivalent plastic strain rate

$$\dot{\bar{\varepsilon}}^{pl} = \sqrt{\frac{2}{3}\dot{\boldsymbol{\varepsilon}}^{pl} : \dot{\boldsymbol{\varepsilon}}^{pl}}$$

where $\dot{\boldsymbol{\varepsilon}}^{pl}$ represents the rate of plastic flow.

Strain rate is accounted for using the Cowper and Symonds model which scales the yield stress with the factor

$$1 + \left(\frac{\dot{\varepsilon}}{C}\right)^{1/p}$$

where $\dot{\varepsilon}$ is the strain rate.

Table 153.1 shows the difference between MAT 153 and MAT 104/105. MAT 153 is less computationally expensive than MAT 104/105. Kinematic hardening, which already exists in MAT 103, is included in MAT 153, but not in MAT 104/105.

Table 153.1 Difference between MAT 153 and MAT 104/105

	MAT 153	**MAT 104**	**MAT 105**
Computational cost	1.0	3.0	3.0
Isotropic hardening	One component	Two components	One component
Kinematic hardening	Two components	N/A	N/A
Output stress	$\tilde{\sigma}$ IDS=0 $\tilde{\sigma}(1-D)$ IDS=1	$\tilde{\sigma}(1-D)$	$\tilde{\sigma}(1-D)$
Damaged plastic strain	$r=\int \dot{\bar{\varepsilon}}^{pl}$ IDEP=0 $r=\int (1-D)\dot{\bar{\varepsilon}}^{pl}$ IDEP=1	$r=\int (1-D)\dot{\bar{\varepsilon}}^{pl}$	$r=\int (1-D)\dot{\bar{\varepsilon}}^{pl}$
Accumulation when	$\frac{\sigma_m}{\sigma_{eq}} > -\frac{1}{3}$	$\sigma_1 > 0$	$\sigma_1 > 0$
Isotropic plasticity	Yes	Yes	Yes
Anisotropic plasticity	No	Yes	No
Isotropic damage	Yes	Yes	Yes
Anisotropic damage	No	Yes	No

*MAT_DESHPANDE_FLECK_FOAM

This is material type 154 for solid elements. This material is for modeling aluminum foam used as a filler material in aluminum extrusions to enhance the energy absorbing capability of the extrusion. Such energy absorbers are used in vehicles to dissipate energy during impact. This model was developed by Reyes, Hopperstad, Berstad, and Langseth [2002] and is based on the foam model by Deshpande and Fleck [2000].

Card 1

Variable	MID	RHO	E	PR	ALPHA	GAMMA		
Type	A8	F	F	F	F	F		
Default	-	-	-	-	-	-		

Card 2

Variable	EPSD	ALPHA2	BETA	SIGP	DERFI	CFAIL		
Type	F	F	F	F	F	F		
Default	-	-	-	-	-	-		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RHO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.
ALPHA	Controls shape of yield surface.
GAMMA	See remarks.
EPSD	Densification strain.

VARIABLE	DESCRIPTION
ALPHA2	See remarks.
BETA	See remarks.
SIGP	See remarks.
DERFI	Type of derivation used in material subroutine EQ.0: Numerical derivation EQ.1: Analytical derivation
CFAIL	Failure strain.

The yield stress function Φ is defined by:

$$\Phi = \hat{\sigma} - \sigma_y$$

The equivalent stress $\hat{\sigma}$ is given by:

$$\hat{\sigma}^2 = \frac{\sigma_{VM}^2 + \alpha^2 \sigma_m^2}{1 + \left(\frac{\alpha}{3}\right)^2}$$

where, σ_{VM}, is the von Mises effective stress:

$$\sigma_{VM} = \sqrt{\frac{2}{3}\sigma^{dev} : \sigma^{dev}}$$

In this equation σ_m and σ^{dev} are the mean and deviatoric stress:

$$\sigma^{dev} = \sigma - \sigma_m I$$

The yield stress σ_y can be expressed as:

$$\sigma_y = \sigma_p + \gamma \frac{\hat{\varepsilon}}{\varepsilon_D} + \alpha_2 \ln\left(\frac{1}{1 - \left(\frac{\hat{\varepsilon}}{\varepsilon_D}\right)^\beta}\right)$$

Here, σ_p, α_2, γ and β are material parameters. The densification strain ε_D is defined as:

$$\varepsilon_D = -\ln\left(\frac{\rho_f}{\rho_{f0}}\right)$$

where ρ_f is the foam density and ρ_{f0} is the density of the virgin material.

*MAT_PLASTICITY_COMPRESSION_TENSION_EOS

This is Material Type 155. An isotropic elastic-plastic material where unique yield stress versus plastic strain curves can be defined for compression and tension. Also, failure can occur based on a plastic strain or a minimum time step size. Rate effects on the yield stress are modeled either by using the Cowper-Symonds strain rate model or by using two load curves that scale the yield stress values in compression and tension, respectively. Material rate effects, which are independent of the plasticity model, are based on a 6-term Prony series Maxwell mode that generates an additional stress tensor. The viscous stress tensor is superimposed on the stress tensor generated by the plasticity. Pressure is defined by an equation of state, which is required to utilize this model. This model is for solid elements only.

Card 1

Variable	MID	RO	E	PR	C	P	FAIL	TDEL
Type	A8	F	F	F	F	F	F	F
Default	none	none	none	none	0	0	10.E+20	0

Card 2

Variable	LCIDC	LCIDT	LCSRC	LCSRT	SRFLAG			
Type	I	I	I	I	F			
Default	0	0	0	0	0			

Card 3

Variable	PC	PT	PCUTC	PCUTT	PCUTF			
Type	F	F	F	F	F			
Default	0	0	0	0	0			

Card 4	1	2	3	4	5	6	7	8
Variable	K							
Type	F							

Card Format for viscoelastic constants. Up to 6 cards may be input. A keyword card (with a "*" in column 1) terminates this input if less than 6 cards are used.

Optional Cards	1	2	3	4	5	6	7	8
Variable	GI	BETAI						
Type	F	F						

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.
C	Strain rate parameter, C, see formula below.
P	Strain rate parameter, P, see formula below.
FAIL	Failure flag. LT.0.0: User defined failure subroutine is called to determine failure EQ.0.0: Failure is not considered. This option is recommended if failure is not of interest since many calculations will be saved. GT.0.0: Plastic strain to failure. When the plastic strain reaches this value, the element is deleted from the calculation.
TDEL	Minimum time step size for automatic element deletion.
LCIDC	Load curve ID defining yield stress versus effective plastic strain in compression.

VARIABLE	DESCRIPTION
LCIDT	Load curve ID defining yield stress versus effective plastic strain in tension.
LCSRC	Optional load curve ID defining strain rate scaling effect on yield stress when the material is in compression.
LCSRT	Optional load curve ID defining strain rate scaling effect on yield stress when the material is in tension.
SRFLAG	Formulation for rate effects: EQ.0.0: Total strain rate, EQ.1.0: Deviatoric strain rate.
PC	Compressive mean stress (pressure) at which the yield stress follows load curve ID, LCIDC. If the pressure falls between PC and PT a weighted average of the two load curves is used.
PT	Tensile mean stress at which the yield stress follows load curve ID, LCIDT.
PCUTC	Pressure cut-off in compression.
PCUTT	Pressure cut-off in tension.
PCUTF	Pressure cut-off flag. EQ.0.0: Inactive, EQ.1.0: Active.
K	Optional bulk modulus for the viscoelastic material. If nonzero a Kelvin type behavior will be obtained. Generally, K is set to zero.
GI	Optional shear relaxation modulus for the ith term
BETAI	Optional shear decay constant for the ith term

Remarks:

The stress strain behavior follows a different curve in compression than it does in tension. Tension is determined by the sign of the mean stress where a positive mean stress (i.e., a negative pressure) is indicative of tension. Two curves must be defined giving the yield stress versus effective plastic strain for both the tension and compression regimes.

*MAT_MUSCLE

This is material type 156 for truss elements. This material is a Hill-type muscle model with activation and a parallel damper. Also, see *MAT_SPRING_MUSCLE where a description of the theory is available.

Card 1

Variable	MID	RO	SNO	SRM	PIS	SSM	CER	DMP
Type	A8	F	F	F	F	F	F	F
Default								

Card 2

Variable	ALM	SFR	SVS	SVR	SSP			
Type	F	F	F	F	F			
Default	0.0	1.0	1.0	1.0	0.0			

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Material density in the initial undeformed configuration.
SNO	Initial stretch ratio, $\frac{l}{l_0}$, i.e., the current length as defined by the nodal points at t=0 divided by the initial length. The density for the nodal mass calculation is RO/SNO, or $\frac{l_0\rho}{l}$.
SRM	Maximum strain rate.
PIS	Peak isometric stress corresponding to the dimensionless value of unity in the dimensionless stress versus strain function, see *SSP* below.

VARIABLE	DESCRIPTION
SSM	Strain when the dimensionless stress versus strain function, *SSP* below, reaches its maximum stress value.
CER	Constant, governing the exponential rise of *SSP*. Required if *SSP*=0.
DMP	Damping constant.
ALM	Activation level vs. time. LT.0: absolute value gives load curve ID GE.0: constant value of *ALM* is used
SFR	Scale factor for strain rate maximum vs. the stretch ratio, $\frac{l}{l_0}$. LT.0: absolute value gives load curve ID GE.0: constant value of 1.0 is used
SVS	Active dimensionless tensile stress vs. the stretch ratio, $\frac{l}{l_0}$. LT.0: absolute value gives load curve ID GE.0: constant value of 1.0 is used
TV	Active dimensionless tensile stress vs. the normalized strain rate, $\frac{\dot{l}}{l_0}$. LT.0: absolute value gives load curve ID GE.0: constant value of 1.0 is used
SSP	Isometric dimensionless stress vs. the stretch ratio, $\frac{l}{l_0}$ for the parallel elastic element. LT.0: absolute value gives load curve ID EQ.0: exponential function is used (see below) GT.0: constant value of 0.0 is used

Remarks:

The material behavior of the muscle model is adapted from material_S15, the spring muscle model and treated here as a standard material. The initial length of muscle is calculated automatically. The force, relative length and shortening velocity are replaced by stress, strain and strain rate. A new parallel damping element is added.

The strain and normalized strain rate are defined respectively as

$$\varepsilon = \frac{l}{l_o} - 1 = L - 1$$

$$\dot{\varepsilon} = \frac{\dot{l}}{l_o\ \dot{\varepsilon}_{\max}} = \frac{V^M}{l_o * (SRM * SFR)} = \frac{V^M}{(l_o * SRM) * SFR} = \frac{V^M}{V_{\max} * SFR} = V$$

where $l_o =$, is the original muscle length.

From the relation above, it is known:

$$l_o = \frac{l_0}{1+\varepsilon_0}$$

where $\varepsilon_0 = SNO$; $l_0 =$ muscle length at time 0.

Stress of Contractile Element is:

$$\sigma_1 = \sigma_{\max}\, a(t)\, f(\varepsilon)\, g(\dot{\varepsilon})$$

where $\sigma_{\max} = PIS$; $a(t) = ALM$; $f(\varepsilon) = SVS$; $g(\dot{\varepsilon}) = SVR$.

Stress of Passive Element is:

$$\sigma_2 = \sigma_{\max} h(\varepsilon)$$

For exponential relationship: $h(\varepsilon) = \begin{cases} 0 & \varepsilon \le 0 \\ \dfrac{1}{\exp(c)-1}\left[\exp\left(\dfrac{c\varepsilon}{L_{\max}}\right)-1\right] & \varepsilon > 0 \quad c \ne 0 \\ \varepsilon / L_{\max} & \varepsilon > 0 \quad c = 0 \end{cases}$

where $L_{\max} = 1 + SSM$; and $c = CER$.

Stress of Damping Element is:

$$\boldsymbol{\sigma_3 = D\varepsilon\dot{\varepsilon}}$$

Total Stress is:

$$\sigma = \sigma_1 + \sigma_2 + \sigma_3$$

*MAT_ANISOTROPIC_ELASTIC_PLASTIC

This is Material Type 157. This material model is a combination of the anisotropic elastic material model (MAT_002) and the anisotropic plastic material model (MAT_103_P).

Card 1

1	2	3	4	5	6	7	8
MID	RO	SIGY	LCSS	QR1	CR1	QR2	CR2
A8	F	F	F	F	F	F	F

Card 2

C11	C12	C13	C14	C15	C16	C22	C23
F	F	F	F	F	F	F	F

Card 3

C24	C25	C26	C33	C34	C35	C36	C44
F	F	F	F	F	F	F	F

Card 4

C45	C46	C55	C56	C66	R00	R45	R90
F	F	F	F	F	F	F	F

Card 5

S11	S22	S33	S12	AOPT			
F	F	F	F	F			

Card 6

Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card 7

Variable	V1	V2	V3	D1	D2	D3	BETA	
Type	F	F	F	F	F	F	F	

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
SIGY	Initial yield stress
LCSS	Load curve ID. The load curve ID defines effective stress versus effective plastic strain. QR1, CR1, QR2, and CR2 are ignored with this option.
CIJ	The I, J term in the 6×6 anisotropic constitutive matrix. Note that 1 corresponds to the *a* material direction, 2 to the *b* material direction, and 3 to the *c* material direction.
QR1	Isotropic hardening parameter $Qr1$
CR1	Isotropic hardening parameter $Cr1$
QR2	Isotropic hardening parameter $Qr2$
CR2	Isotropic hardening parameter $Cr2$
S11	Yield stress in local-x direction. This input is ignored if (R00, R45, R90) > 0.
S22	Yield stress in local-y direction. This input is ignored if (R00, R45, R90) > 0.
S33	Yield stress in local-z direction. This input is ignored if (R00, R45, R90) > 0.

VARIABLE	DESCRIPTION
S12	Yield stress in local-xy direction. This input is ignored if (R00, R45, R90) > 0.
AOPT	Material axes option (see MAT_*OPTION* TROPIC_ELASTIC for a more complete description. EQ.0.0: locally orthotropic with material axes determined by element nodes as shown in Figure 2.1. Nodes 1, 2, and 4 of an element are identical to the nodes used for the definition of a coordinate system as by *DEFINE_COORDINATE_NODES. EQ.2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_ VECTOR. EQ.3.0: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle, BETA, from a line in the plane of the element defined by the cross product of the vector **v** with the element normal. The plane of a solid element is the mid surface between the inner surface and outer surface defined by the first four nodes and the last four nodes of the connectivity of the element, respectively. LT.0.0: the absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_NODES, *DEFINE_ COORDINATE_SYSTEM or *DEFINE_COORDINATE_ VECTOR). Available in R3 version of 971 and later.
XP, YP, ZP	XP, YP, ZP define coordinates of point **p** for AOPT=1 and 4.
A1, A2, A3	a1, a2, a3 define components of vector **a** for AOPT=2.
D1, D2, D3	d1, d2, d3 define components of vector **d** for AOPT=2.
V1, V2, V3	v1, v2, v3 define components of vector **v** for AOPT=3 and 4.
BETA	Material angle in degrees for AOPT=3, may be overwritten on the element card, see *ELEMENT_SHELL_BETA or *ELEMENT_SOLID_ORTHO.

*MAT_RATE_SENSITIVE_COMPOSITE_FABRIC

This is Material Type 158. Depending on the type of failure surface, this model may be used to model rate sensitive composite materials with unidirectional layers, complete laminates, and woven fabrics. A viscous stress tensor, based on an isotropic Maxwell model with up to six terms in the Prony series expansion, is superimposed on the rate independent stress tensor of the composite fabric. The viscous stress tensor approach should work reasonably well if the stress increases due to rate affects are up to 15% of the total stress. This model is implemented for both shell and thick shell elements. The viscous stress tensor is effective at eliminating spurious stress oscillations.

Card 1

Variable	MID	RO	EA	EB	(EC)	PRBA	TAU1	GAMMA1
Type	A8	F	F	F	F	F	F	F

Card 2

Variable	GAB	GBC	GCA	SLIMT1	SLIMC1	SLIMT2	SLIMC2	SLIMS
Type	F	F	F	F	F	F	F	F

Card 3

Variable	AOPT	TSIZE	ERODS	SOFT	FS			
Type	F	F	F	F	F			

Card 4

Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card 5	1	2	3	4	5	6	7	8
Variable	V1	V2	V3	D1	D2	D3	BETA	
Type	F	F	F	F	F	F	F	

Card 6

Variable	E11C	E11T	E22C	E22T	GMS			
Type	F	F	F	F	F			

Card 7

Variable	XC	XT	YC	YT	SC			
Type	F	F	F	F	F			

Card 8

Variable	K							
Type	F							

Card Format for viscoelastic constants. Up to 6 cards may be input. A keyword card (with a "*" in column 1) terminates this input if less than 6 cards are used.

Optional Cards	1	2	3	4	5	6	7	8
Variable	GI	BETAI						
Type	F	F						

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
EA	E_a, Young's modulus - longitudinal direction
EB	E_b, Young's modulus - transverse direction
(EC)	E_c, Young's modulus - normal direction (not used)
PRBA	ν_{ba}, Poisson's ratio ba
TAU1	$\tau 1$, stress limit of the first slightly nonlinear part of the shear stress versus shear strain curve. The values $\tau 1$ and $\gamma 1$ are used to define a curve of shear stress versus shear strain. These values are input if FS, defined below, is set to a value of -1.
GAMMA1	$\gamma 1$, strain limit of the first slightly nonlinear part of the shear stress versus shear strain curve.
GAB	G_{ab}, shear modulus ab
GBC	G_{bc}, shear modulus bc
GCA	G_{ca}, shear modulus ca
SLIMT1	Factor to determine the minimum stress limit after stress maximum (fiber tension).
SLIMC1	Factor to determine the minimum stress limit after stress maximum (fiber compression).
SLIMT2	Factor to determine the minimum stress limit after stress maximum (matrix tension).
SLIMC2	Factor to determine the minimum stress limit after stress maximum (matrix compression).
SLIMS	Factor to determine the minimum stress limit after stress maximum (shear).
AOPT	Material axes option (see MAT_*OPTION* TROPIC_ELASTIC for a more complete description): EQ.0.0: locally orthotropic with material axes determined by element nodes 1, 2, and 4, as with *DEFINE_COORDINATE_NODES.

VARIABLE	DESCRIPTION
	EQ.2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR. EQ.3.0: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle (BETA) from a line in the plane of the element defined by the cross product of the vector **v** with the element normal. LT.0.0: the absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_NODES, *DEFINE_COORDINATE_SYSTEM or *DEFINE_COORDINATE_VECTOR). Available in R3 version of 971 and later.
TSIZE	Time step for automatic element deletion.
ERODS	Maximum effective strain for element layer failure. A value of unity would equal 100% strain.
SOFT	Softening reduction factor for strength in the crashfront.
FS	Failure surface type: EQ.1.0: smooth failure surface with a quadratic criterion for both the fiber (a) and transverse (b) directions. This option can be used with complete laminates and fabrics. EQ.0.0: smooth failure surface in the transverse (b) direction with a limiting value in the fiber (a) direction. This model is appropriate for unidirectional (UD) layered composites only. EQ.-1: faceted failure surface. When the strength values are reached then damage evolves in tension and compression for both the fiber and transverse direction. Shear behavior is also considered. This option can be used with complete laminates and fabrics.
XP YP ZP	Define coordinates of point **p** for AOPT = 1.
A1 A2 A3	Define components of vector **a** for AOPT = 2.
V1 V2 V3	Define components of vector **v** for AOPT = 3.
D1 D2 D3	Define components of vector **d** for AOPT = 2.
BETA	Material angle in degrees for AOPT = 3, may be overridden on the element card, see *ELEMENT_SHELL_BETA.
E11C	Strain at longitudinal compressive strength, a-axis.
E11T	Strain at longitudinal tensile strength, a-axis.

VARIABLE	DESCRIPTION
E22C	Strain at transverse compressive strength, b-axis.
E22T	Strain at transverse tensile strength, b-axis.
GMS	Strain at shear strength, ab plane.
XC	Longitudinal compressive strength
XT	Longitudinal tensile strength, see below.
YC	Transverse compressive strength, b-axis, see below.
YT	Transverse tensile strength, b-axis, see below.
SC	Shear strength, ab plane.
K	Optional bulk modulus for the viscoelastic material. If nonzero a Kelvin type behavior will be obtained. Generally, K is set to zero.
GI	Optional shear relaxation modulus for the ith term
BETAI	Optional shear decay constant for the ith term

Remarks:

See the remark for material type 58, *MAT_LAMINATED_COMPOSITE_FABRIC, for the treatment of the composite material.

Rate effects are taken into account through a Maxwell model using linear viscoelasticity by a convolution integral of the form:

$$\sigma_{ij} = \int_0^t g_{ijkl}\left(t-\tau\right)\frac{\partial \varepsilon_{kl}}{\partial \tau}d\tau$$

where $g_{ijkl(t-\tau)}$ is the relaxation functions for the different stress measures. This stress is added to the stress tensor determined from the strain energy functional. Since we wish to include only simple rate effects, the relaxation function is represented by six terms from the Prony series:

$$g\left(t\right) = \sum_{m=1}^{N} G_m e^{-\beta_m t}$$

We characterize this in the input by the shear moduli, G_i, and decay constants, β_i. An arbitrary number of terms, not exceeding 6, may be used when applying the viscoelastic model. The composite failure is not directly affected by the presence of the viscous stress tensor.

***MAT_CSCM** _{*OPTION*}

This is material type 159. This is a smooth or continuous surface cap model and is available for solid elements in LS-DYNA. The user has the option of inputting his own material properties (<BLANK> option), or requesting default material properties for normal strength concrete (CONCRETE).

Available options include:

<**BLANK**>

CONCRETE

such that the keyword cards appear as:

***MAT_CSCM**
***MAT_CSCM _CONCRETE**

Define the next two cards for all options:

Card 1

Variable	MID	RO	NPLOT	INCRE	IRATE	ERODE	RECOV	ITRETRC
Type	A8	F	I	F	I	F	F	I

Card 2

Variable	PRED							
Type	F							

Define the following card for the CONCRETE option. Do not define for the <BLANK> option.

Card 3

Variable	FPC	DAGG	UNITS					
Type	F	F	I					

Define the following cards for the <BLANK> option. Do not define for CONCRETE.

Card 3

Variable	G	K	ALPHA	THETA	LAMDA	BETA	NH	CH
Type	F	F	F	F	F	F	F	F

Card 4

Variable	ALPHA1	THETA1	LAMDA1	BETA1	ALPHA2	THETA2	LAMDA2	BETA2
Type	F	F	F	F	F	F	F	F

Card 5

Variable	R	X0	W	D1	D2			
Type	F	F	F	F	F			

Card 6

Variable	B	GFC	D	GFT	GFS	PWRC	PWRT	PMOD
Type	F	F	F	F	F	F	F	F

Card 7

Variable	ETA0C	NC	ETA0T	NT	OVERC	OVERT	SRATE	REPOW
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
NPLOT	Controls what is written as component 7 to the d3plot database. LS-Prepost always blindly labels this component as effective plastic strain: EQ.1: Maximum of brittle and ductile damage (default). EQ.2: Maximum of brittle and ductile damage, with recovery of brittle damage. EQ.3: Brittle damage. EQ.4: Ductile damage. EQ.5: κ (intersection of cap with shear surface). EQ.6: X_0 (intersection of cap with pressure axis). EQ.7: ε_v^p (plastic volume strain).
INCRE	Maximum strain increment for subincrementation. If left blank, a default value is set during initialization based upon the shear strength and stiffness.
IRATE	Rate effects options: EQ.0: Rate effects model turned off (default). EQ.1: Rate effects model turned on.
ERODE	Elements erode when damage exceeds 0.99 and the maximum principal strain exceeds ERODE-1.0. For erosion that is independent of strain, set ERODE equal to 1.0. Erosion does not occur if ERODE is less than 1.0.
RECOV	The modulus is recovered in compression when RECOV is equal to 0 (default). The modulus remains at the brittle damage level when RECOV is equal to 1. Partial recovery is modeled for values of RECOV between 0 and 1. Two options are available: EQ.1: Input a value between 0 and 1. Recovery is based upon the sign of the pressure invariant only. EQ.2: Input a value between 10 and 11. Recovery is based upon the sign of both the pressure and volumetric strain. In this case, RECOV=RECOV-10, and a flag is set to request the volumetric strain check.
IRETRC	Cap retraction option: EQ.0: Cap does not retract (default). EQ.1: Cap retracts.
PRED	Pre-existing damage ($0 \le$ PreD < 1). If left blank, the default is zero (no pre-existing damage).

Define for the CONCRETE option. Note that the default concrete input parameters are for normal strength concrete with unconfined compression strengths between about 28 and 58 MPa.

VARIABLE	DESCRIPTION
FPC	Unconfined compression strength, f'_C. If left blank, default is 30 MPa.
DAGG	Maximum aggregate size, Dagg. If left blank, default is 19 mm (3/4 inch).
UNITS	Units options: EQ.0: GPa, mm, msec, Kg/mm^3, kN EQ.1: MPa, mm, msec, g/mm^3, N EQ.2: MPa, mm, sec, Mg/mm^3, N EQ.3: Psi, inch, sec, lbf-s^2/inch^4, lbf EQ.4: Pa, m, sec, kg/m^3, N

Define for <BLANK> option only.

VARIABLE	DESCRIPTION
G	Shear modulus.
K	Bulk modulus.
ALPHA	Tri-axial compression surface constant term, α.
THETA	Tri-axial compression surface linear term, θ.
LAMDA	Tri-axial compression surface nonlinear term, λ.
BETA	Tri-axial compression surface exponent, β.
ALPHA1	Torsion surface constant term, α_1.
THETA1	Torsion surface linear term, θ_1.
LAMDA1	Torsion surface nonlinear term, λ_1.
BETA1	Torsion surface exponent, β_1.
ALPHA2	Tri-axial extension surface constant term, α_2.
THETA2	Tri-axial extension surface linear term, θ_2.
LAMDA2	Tri-axial extension surface nonlinear term, λ_2.

VARIABLE	DESCRIPTION
BETA2	Tri-axial extension surface exponent, β_2.
NH	Hardening initiation, N_H.
CH	Hardening rate, C_H.
R	Cap aspect ratio, R.
X0	Cap initial location, X_0.
W	Maximum plastic volume compaction, W.
D1	Linear shape parameter, D_1.
D2	Quadratic shape parameter, D_2.
B	Ductile shape softening parameter, B.
GFC	Fracture energy in uniaxial stress G_{fc}.
D	Brittle shape softening parameter, D.
GFT	Fracture energy in uniaxial tension, G_{ft}.
GFS	Fracture energy in pure shear stress, G_{fs}.
PWRC	Shear-to-compression transition parameter.
PWRT	Shear-to-tension transition parameter.
PMOD	Modify moderate pressure softening parameter.
ETA0C	Rate effects parameter for uniaxial compressive stress, η_{0c}.
NC	Rate effects power for uniaxial compressive stress, N_C.
ETA0T	Rate effects parameter for uniaxial tensile stress, η_{0t}.
NT	Rate effects power for uniaxial tensile stress, N_t.
OVERC	Maximum overstress allowed in compression.
OVERT	Maximum overstress allowed in tension.
SRATE	Ratio of effective shear stress to tensile stress fluidity parameters.
*R*EPOW	Power which increases fracture energy with rate effects.

Remarks:

Model Formulation and Input Parameters

This is a cap model with a smooth intersection between the shear yield surface and hardening cap, as shown in Figure 159.1. The initial damage surface coincides with the yield surface. Rate effects are modeled with viscoplasticity. For a complete theoretical description, with references and example problems see [Murray 2007] and [Murray, Abu-Odeh and Bligh 2007].

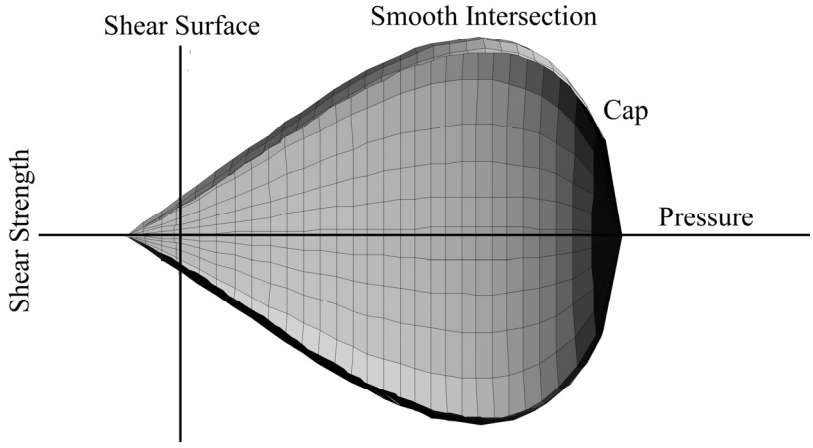


Figure 159.1. General shape of the concrete model yield surface in two-dimensions.

Stress Invariants. The yield surface is formulated in terms of three stress invariants: J_1 is the first invariant of the stress tensor, J_2' is the second invariant of the deviatoric stress tensor, and J_3' is the third invariant of the deviatoric stress tensor. The invariants are defined in terms of the deviatoric stress tensor, S_{ij} and pressure, P, as follows:

$$J_1 = 3P$$

$$J_2' = \frac{1}{2} S_{ij} S_{ij}$$

$$J_3' = \frac{1}{3} S_{ij} S_{jk} S_{ki}$$

Plasticity Surface. The three invariant yield function is based on these three invariants, and the cap hardening parameter, κ, as follows:

$$f(J_1, J_2', J_3', \kappa) = J_2' - \Re^2 F_f^2 F_c$$

Here F_f is the shear failure surface, F_c is the hardening cap, and $\Re$ is the Rubin three-invariant reduction factor. The cap hardening parameter κ is the value of the pressure invariant at the intersection of the cap and shear surfaces.

Trial elastic stress invariants are temporarily updated via the trial elastic stress tensor, σ^T. These are denoted J_1^T, $J_2'^T$, and $J_3'^T$. Elastic stress states are modeled when $f(J_1^T, J_2'^T, J_3'^T, \kappa^T) \le 0$. Elastic-plastic stress states are modeled when $f(J_1^T, J_2'^T, J_3'^T, \kappa^T) > 0$. In this case, the plasticity algorithm returns the stress state to the yield surface such that $f(J_1^P, J_2'^P, J_3'^P, \kappa^P) = 0$. This is accomplished by enforcing the plastic consistency condition with associated flow.

Shear Failure Surface. The strength of concrete is modeled by the shear surface in the tensile and low confining pressure regimes:

$$F_f(J_1) = \alpha - \lambda \exp^{-\beta J_1} + \theta J_1$$

Here the values of $\alpha, \beta, \lambda,$ and θ are selected by fitting the model surface to strength measurements from triaxial compression (TXC) tests conducted on plain concrete cylinders.

Rubin Scaling Function. Concrete fails at lower values of $\sqrt{3J_2'}$ (principal stress difference) for triaxial extension (TXE) and torsion (TOR) tests than it does for TXC tests conducted at the same pressure. The Rubin scaling function $\Re$ determines the strength of concrete for any state of stress relative to the strength for TXC, via $\Re F_f$. Strength in torsion is modeled as $Q_1 F_f$. Strength in TXE is modeled as $Q_2 F_f$, where:

$$Q_1 = \alpha_1 - \lambda_1 exp^{-\beta_1 J_1} + \theta_1 J_1$$
$$Q_2 = \alpha_2 - \lambda_2 exp^{-\beta_2 J_1} + \theta_2 J_1$$

Cap Hardening Surface. The strength of concrete is modeled by a combination of the cap and shear surfaces in the low to high confining pressure regimes. The cap is used to model plastic volume change related to pore collapse (although the pores are not explicitly modeled). The isotropic hardening cap is a two-part function that is either unity or an ellipse:

$$F_c(J_1, \kappa) = 1 - \frac{[J_1 - L(\kappa)]\,[\,|J_1 - L(\kappa)| + J_1 - L(\kappa)\,]}{2\,[X(\kappa) - L(\kappa)]^2}$$

where $L(\kappa)$ is defined as:

$$L(\kappa) = \begin{cases} \kappa & if \quad \kappa > \kappa_0 \\ \kappa_0 & otherwise \end{cases}$$

The equation for F_c is equal to unity for $J_1 \le L(\kappa)$. It describes the ellipse for $J_1 > L(\kappa)$. The intersection of the shear surface and the cap is at $J_1 = \kappa$. κ_0 is the value of J_1 at the *initial* intersection of the cap and shear surfaces before hardening is engaged (before the cap moves). The equation for $L(\kappa)$ restrains the cap from retracting past its initial location at κ_0.

The intersection of the cap with the J_1 axis is at $J_1 = X(\kappa)$. This intersection depends upon the cap ellipticity ratio R, where R is the ratio of its major to minor axes:

$$X(\kappa) = L(\kappa) + RF_f(L(\kappa))$$

The cap moves to simulate plastic volume change. The cap expands ($X(\kappa)$ and κ increase) to simulate plastic volume compaction. The cap contracts ($X(\kappa)$ and κ decrease) to simulate plastic volume expansion, called dilation. The motion (expansion and contraction) of the cap is based upon the hardening rule:

$$\varepsilon_v^p = W\left(1 - \exp^{-D_1(X-X_0)-D_2(X-X_0)^2}\right)$$

Here ε_v^p the plastic volume strain, W is the maximum plastic volume strain, and D_1 and D_2 are model input parameters. X_0 is the initial location of the cap when $\kappa = \kappa_0$.

The five input parameters (X_0, W, D_1, D_2, and R) are obtained from fits to the pressure-volumetric strain curves in isotropic compression and uniaxial strain. X_0 determines the pressure at which compaction initiates in isotropic compression. R, combined with X_0, determines the pressure at which compaction initiates in uniaxial strain. D_1, and D_2 determine the shape of the pressure-volumetric strain curves. W determines the maximum plastic volume compaction.

Shear Hardening Surface. In unconfined compression, the stress-strain behavior of concrete exhibits nonlinearity and dilation prior to the peak. Such behavior is be modeled with an initial shear yield surface, $N_H F_f$, which hardens until it coincides with the ultimate shear yield surface, F_f. Two input parameters are required. One parameter, N_H, initiates hardening by setting the location of the initial yield surface. A second parameter, C_H, determines the rate of hardening (amount of nonlinearity).

Damage. Concrete exhibits softening in the tensile and low to moderate compressive regimes.

$$\sigma_{ij}^d = (1-d)\sigma_{ij}^{vp}$$

A scalar damage parameter, d, transforms the viscoplastic stress tensor without damage, denoted σ^{vp}, into the stress tensor with damage, denoted σ^d. Damage accumulation is based upon two distinct formulations, which we call brittle damage and ductile damage. The initial damage threshold is coincident with the shear plasticity surface, so the threshold does not have to be specified by the user.

Ductile Damage. Ductile damage accumulates when the pressure (*P*) is compressive and an energy-type term, τ_c, exceeds the damage threshold, τ_{0c}. Ductile damage accumulation depends upon the total strain components, ε_{ij}, as follows:

$$\tau_c = \sqrt{\frac{1}{2}\sigma_{ij}\varepsilon_{ij}}$$

The stress components σ_{ij} are the elasto-plastic stresses (with kinematic hardening) calculated before application of damage and rate effects.

Brittle Damage. Brittle damage accumulates when the pressure is tensile and an energy-type term, τ_t, exceeds the damage threshold, τ_{0t}. Brittle damage accumulation depends upon the maximum principal strain, $\varepsilon_{\max}$, as follows:

$$\tau_t = \sqrt{E\,\varepsilon_{\max}^2}$$

Softening Function. As damage accumulates, the damage parameter *d* increases from an initial value of zero, towards a maximum value of one, via the following formulations:

Brittle Damage
$$d(\tau_t) = \frac{0.999}{D}\left[\frac{1+D}{1+D\exp^{-C(\tau_t-\tau_{0t})}}-1\right]$$

Ductile Damage
$$d(\tau_c) = \frac{dmax}{B}\left[\frac{1+B}{1+B\exp^{-A(\tau_c-\tau_{0c})}}-1\right]$$

The damage parameter that is applied to the six stresses is equal to the current maximum of the brittle or ductile damage parameter. The parameters A and B or C and D set the shape of the softening curve plotted as stress-displacement or stress-strain. The parameter *dmax* is the maximum damage level that can be attained. It is calculated internally calculated and is less than one at moderate confining pressures. The compressive softening parameter, A, may also be reduced with confinement, using the input parameter *pmod*, as follows:

$$A = A(dmax + 0.001)^{pmod}$$

Regulating Mesh Size Sensitivity. The concrete model maintains constant fracture energy, regardless of element size. The fracture energy is defined here as the area under the stress-displacement curve from peak strength to zero strength. This is done by internally formulating the softening parameters A and C in terms of the element length, *l* (cube root of the element volume), the fracture energy, G_f, the initial damage threshold, τ_{0t} or τ_{0c}, and the softening shape parameters, D or B.

The fracture energy is calculated from up to five user-specified input parameters (G_{fc}, G_{ft}, G_{fs}, *pwrc*, *pwrc*). The user specifies three distinct fracture energy values. These are the fracture energy in uniaxial tensile stress, G_{ft}, pure shear stress, G_{fs}, and uniaxial compressive stress, G_{fc}.

The model internally selects the fracture energy from equations which interpolate between the three fracture energy values as a function of the stress state (expressed via two stress invariants). The interpolation equations depend upon the user-specified input powers *pwrc* and *pwrt*, as follows.

$$\textit{if the pressure is tensile} \qquad G_f = G_{fs} + trans(G_{ft} - G_{fs}) \;\; \textit{where} \;\; trans = \left(\frac{-J_1}{\sqrt{3J_2'}} \right)^{pwrt}$$

$$\textit{if the pressure is compressive} \quad G_f = G_{fs} + trans(G_{fc} - G_{fs}) \;\; \textit{where} \;\; trans = \left(\frac{J_1}{\sqrt{3J_2'}} \right)^{pwrc}$$

The internal parameter *trans* is limited to range between 0 and 1.

Element Erosion. An element losses all strength and stiffness as $d \rightarrow 1$. To prevent computational difficulties with very low stiffness, element erosion is available as a user option. An element erodes when $d > 0.99$ and the maximum principal strain is greater than a user supplied input value, ERODE-1.0.

Viscoplastic Rate Effects. At each time step, the viscoplastic algorithm interpolates between the elastic trial stress, σ_{ij}^T, and the inviscid stress (without rate effects), σ_{ij}^p, to set the viscoplastic stress (with rate effects), σ_{ij}^{vp}:

$$\sigma_{ij}^{vp} = (1-\gamma)\sigma_{ij}^T + \gamma\sigma_{ij}^p \qquad \text{with} \qquad \gamma = \frac{\Delta t / \eta}{1 + \Delta t / \eta}$$

This interpolation depends upon the effective fluidity coefficient, η, and the time step, Δt. The effective fluidity coefficient is internally calculated from five user-supplied input parameters and interpolation equations:

$$\textit{if the pressure is tensile} \qquad \eta = \eta_s + trans(\eta_t - \eta_s) \qquad trans = \left(\frac{-J_1}{\sqrt{3J_2'}} \right)^{pwrt}$$

$$\textit{if the pressure is compressive} \quad \eta = \eta_s + trans(\eta_c - \eta_s) \qquad trans = \left(\frac{J_1}{\sqrt{3J_2'}} \right)^{pwrc}$$

$$\eta_t = \frac{\eta_{0t}}{\dot{\varepsilon}^{N_t}} \qquad \eta_c = \frac{\eta_{0c}}{\dot{\varepsilon}^{N_c}} \qquad \eta_s = Srate\,\eta_t$$

The input parameters are η_{0t} and N_t for fitting uniaxial tensile stress data, η_{0c} and N_c for fitting the uniaxial compressive stress data, and *Srate* for fitting shear stress data. The effective strain rate is $\dot{\varepsilon}$.

This viscoplastic model may predict substantial rate effects at high strain rates ($\dot{\varepsilon}$ >100). To limit rate effects at high strain rates, the user may input overstress limits in tension (*overt*) and

compression (*overc*). These input parameters limit calculation of the fluidity parameter, as follows:

$$if \quad E\dot{\varepsilon}\eta > over \qquad then \qquad \eta = \frac{over}{E\dot{\varepsilon}}$$

where *over* = *overt* when the pressure is tensile, and *over* = *overc* when the pressure is compressive.

The user has the option of increasing the fracture energy as a function of effective strain rate via the *repow* input parameter, as follows:

$$G_f^{rate} = G_f \left(1 + \frac{E\dot{\varepsilon}\eta}{f'} \right)^{repow}$$

Here G_f^{rate} is the fracture energy enhanced by rate effects, and f' is the yield strength before application of rate effects (which is calculated internally by the model). The term in brackets is greater than, or equal to one, and is the approximate ratio of the dynamic to static strength.

***MAT_COMPOSITE_MSC_**{*OPTION*}

Available options include:

<**BLANK**>

DMG

These are Material Types 161 and 162. These models may be used to model the progressive failure analysis for composite materials consisting of unidirectional and woven fabric layers. The progressive layer failure criteria have been established by adopting the methodology developed by Hashin [1980] with a generalization to include the effect of highly constrained pressure on composite failure. These failure models can be used to effectively simulate fiber failure, matrix damage, and delamination behavior under all conditions - opening, closure, and sliding of failure surfaces. The model with DMG option (material 162) is a generalization of the basic layer failure model of Material 161 by adopting the damage mechanics approach for characterizing the softening behavior after damage initiation. These models require an additional license from Materials Sciences Corporation, which developed and supports these models.

Card 1

Variable	MID	RO	EA	EB	EC	PRBA	PRCA	PRCB
Type	A8	F	F	F	F	F	F	F

Card 2

Variable	GAB	GBC	GCA	AOPT	MACF			
Type	F	F	F	F	I			

Card 3

Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card 4	1	2	3	4	5	6	7	8
Variable	V1	V2	V3	D1	D2	D3	BETA	
Type	F	F	F	F	F	F	F	

Card 5

Variable	SAT	SAC	SBT	SBC	SCT	SFC	SFS	SAB
Type	F	F	F	F	F	F	F	F

Card 6

Variable	SBC	SCA	SFFC	AMODEL	PHIC	E_LIMT	S_DELM	
Type	F	F	F	F	F	F	F	

Card 7

Variable	OMGMX	ECRSH	EEXPN	CERATE1	AM1			
Type	F	F	F	F	F			

Define the following cards if and only if the option DMG is specified

Card 8	1	2	3	4	5	6	7	8
Variable	AM2	AM3	AM4	CERATE2	CERATE3	CERATE4		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
EA	E_a, Young's modulus - longitudinal direction
EB	E_b, Young's modulus - transverse direction
EC	E_c, Young's modulus - through thickness direction
PRBA	ν_{ba}, Poisson's ratio ba
PRCA	ν_{ca}, Poisson's ratio ca
PRCB	ν_{cb}, Poisson's ratio cb
GAB	G_{ab}, shear modulus ab
GBC	G_{bc}, shear modulus bc
GCA	G_{ca}, shear modulus ca
AOPT	Material axes option, see Figure 2.1: EQ.0.0: locally orthotropic with material axes determined by element nodes as shown in Figure 2.1. Nodes 1, 2, and 4 of an element are identical to the Nodes used for the definition of a coordinate system by *DEFINE_COORDINATE_NODES. EQ.1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center, to define the a-direction. EQ.2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_ VECTOR. LT.0.0: the absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_NODES, *DEFINE_ COORDINATE_SYSTEM or *DEFINE_COORDINATE_ VECTOR). Available in R3 version of 971 and later.
MACF	Material axes change flag: EQ.1: No change, default, EQ.2: switch material axes a and b, EQ.3: switch material axes a and c, EQ.4: switch material axes b and c.
XP YP ZP	Define coordinates of point **p** for AOPT = 1.

VARIABLE	DESCRIPTION
A1 A2 A3	Define components of vector **a** for AOPT = 2.
V1 V2 V3	Define components of vector **v** for AOPT = 3.
D1 D2 D3	Define components of vector **d** for AOPT = 2.
BETA	Layer in-plane rotational angle in degrees.
SAT	Longitudinal tensile strength
SAC	Longitudinal compressive strength
SBT	Transverse tensile strength
SBC	Transverse compressive strength
SCT	Through thickness tensile strength
SFC	Crush strength
SFS	Fiber mode shear strength
SAB	Matrix mode shear strength, ab plane, see below.
SBC	Matrix mode shear strength, bc plane, see below.
SCA	Matrix mode shear strength, ca plane, see below.
SFFC	Scale factor for residual compressive strength
AMODEL	Material models: EQ.1: Unidirectional layer model EQ.2: Fabric layer model
PHIC	Coulomb friction angle for matrix and delamination failure, <90
E_LIMT	Element eroding axial strain
S_DELM	Scale factor for delamination criterion
OMGMX	Limit damage parameter for elastic modulus reduction
ECRSH	Limit compressive volume strain for element eroding
EEXPN	Limit tensile volume strain for element eroding
CERATE1	Coefficient for strain rate dependent strength properties

VARIABLE	DESCRIPTION
AM1	Coefficient for strain rate softening property for fiber damage in a direction.
AM2	Coefficient for strain rate softening property for fiber damage in b direction.
AM3	Coefficient for strain rate softening property for fiber crush and punch shear damage.
AM4	Coefficient for strain rate softening property for matrix and delamination damage.
CERATE2	Coefficient for strain rate dependent axial moduli.
CERATE3	Coefficient for strain rate dependent shear moduli.
CERATE4	Coefficient for strain rate dependent transverse moduli.

Material Models:

The unidirectional and fabric layer failure criteria and the associated property degradation models for material 161 are described as follows. All the failure criteria are expressed in terms of stress components based on ply level stresses $(\sigma_a, \sigma_b, \sigma_c, \tau_{ab}, \tau_{bc}, \tau_{ca})$ and the associated elastic moduli are $(E_a, E_b, E_c, G_{ab}, G_{bc}, G_{ca})$. Note that for the unidirectional model, a, b and c denote the fiber, in-plane transverse and out-of-plane directions, respectively, while for the fabric model, a, b and c denote the in-plane fill, in-plane warp and out-of-plane directions, respectively.

Unidirectional lamina model

Three criteria are used for fiber failure, one in tension/shear, one in compression and another one in crush under pressure. They are chosen in terms of quadratic stress forms as follows:

Tensile/shear fiber mode:

$$f_1 = \left(\frac{\langle \sigma_a \rangle}{S_{aT}} \right)^2 + \left(\frac{\tau_{ab}^2 + \tau_{ca}^2}{S_{FS}^2} \right) - 1 = 0$$

Compression fiber mode:

$$f_2 = \left(\frac{\langle \sigma_a' \rangle}{S_{aC}} \right)^2 - 1 = 0, \quad \sigma_a' = -\sigma_a + \left\langle -\frac{\sigma_b + \sigma_c}{2} \right\rangle$$

Crush mode:

$$f_3 = \left(\frac{\langle p \rangle}{S_{FC}} \right)^2 - 1 = 0, \quad p = -\frac{\sigma_a + \sigma_b + \sigma_c}{3}$$

where $\langle\ \rangle$ are Macaulay brackets, S_{aT} and S_{aC} are the tensile and compressive strengths in the fiber direction, and S_{FS} and S_{FC} are the layer strengths associated with the fiber shear and crush failure, respectively.

Matrix mode failures must occur without fiber failure, and hence they will be on planes parallel to fibers. For simplicity, only two failure planes are considered: one is perpendicular to the planes of layering and the other one is parallel to them. The matrix failure criteria for the failure plane perpendicular and parallel to the layering planes, respectively, have the forms:

Perpendicular matrix mode:

$$f_4 = \left(\frac{\langle \sigma_b \rangle}{S_{bT}} \right)^2 + \left(\frac{\tau_{bc}}{S'_{bc}} \right)^2 + \left(\frac{\tau_{ab}}{S_{ab}} \right)^2 - 1 = 0$$

Parallel matrix mode (Delamination):

$$f_5 = S^2 \left\{ \left(\frac{\langle \sigma_c \rangle}{S_{bT}} \right)^2 + \left(\frac{\tau_{bc}}{S''_{bc}} \right)^2 + \left(\frac{\tau_{ca}}{S_{ca}} \right)^2 \right\} - 1 = 0$$

where S_{bT} is the transverse tensile strength. Based on the Coulomb-Mohr theory, the shear strengths for the transverse shear failure and the two axial shear failure modes are assumed to be the forms,

$$S_{ab} = S_{ab}^{(0)} + \tan(\varphi)\langle -\sigma_b \rangle$$

$$S'_{bc} = S_{bc}^{(0)} + \tan(\varphi)\langle -\sigma_b \rangle$$

$$S_{ca} = S_{ca}^{(0)} + \tan(\varphi)\langle -\sigma_c \rangle$$

$$S''_{bc} = S_{bc}^{(0)} + \tan(\varphi)\langle -\sigma_c \rangle$$

where φ is a material constant as $\tan(\varphi)$ is similar to the coefficient of friction, and $S_{ab}^{(0)}$, $S_{ca}^{(0)}$ and $S_{bc}^{(0)}$ are the shear strength values of the corresponding tensile modes.

Failure predicted by the criterion of f_4 can be referred to as transverse matrix failure, while the matrix failure predicted by f_5, which is parallel to the layer, can be referred as the delamination mode when it occurs within the elements that are adjacent to the ply interface. Note that a scale factor S is introduced to provide better correlation of delamination area with experiments. The scale factor S can be determined by fitting the analytical prediction to experimental data for the delamination area.

When fiber failure in tension/shear mode is predicted in a layer by f_1, the load carrying capacity of that layer is completely eliminated. All the stress components are reduced to zero instantaneously (100 time steps to avoid numerical instability). For compressive fiber failure, the layer is assumed to carry a residual axial load, while the transverse load carrying capacity is reduced to zero. When the fiber compressive failure mode is reached due to f_2, the axial layer compressive strength stress is assumed to reduce to a residual value S_{RC} ($= SFFC * S_{AC}$). The axial stress is then assumed to remain constant, i.e., $\sigma_a = -S_{RC}$, for continuous compressive loading, while the subsequent unloading curve follows a reduced axial modulus to zero axial stress and strain state. When the fiber crush failure occurs, the material is assumed to behave elastically for compressive pressure, p > 0, and to carry no load for tensile pressure, p < 0.

When a matrix failure (delamination) in the a-b plane is predicted, the strength values for $S_{ca}^{(0)}$ and $S_{bc}^{(0)}$ are set to zero. This results in reducing the stress components σ_c, τ_{bc} and τ_{ca} to the fractured material strength surface. For tensile mode, $\sigma_c > 0$, these stress components are reduced to zero. For compressive mode, $\sigma_c < 0$, the normal stress σ_c is assumed to deform elastically for the closed matrix crack. Loading on the failure envelop, the shear stresses are assumed to 'slide' on the fractured strength surface (frictional shear stresses) like in an ideal plastic material, while the subsequent unloading shear stress-strain path follows reduced shear moduli to the zero shear stress and strain state for both τ_{bc} and τ_{ca} components.

The post failure behavior for the matrix crack in the a-c plane due to f_4 is modeled in the same fashion as that in the a-b plane as described above. In this case, when failure occurs, $S_{ab}^{(0)}$ and $S_{bc}^{(0)}$ are reduced to zero instantaneously. The post fracture response is then governed by failure criterion of f_5 with $S_{ab}^{(0)} = 0$ and $S_{bc}^{(0)} = 0$. For tensile mode, $\sigma_b > 0$, σ_b, τ_{ab} and τ_{bc} are zero. For compressive mode, $\sigma_b < 0$, σ_b is assumed to be elastic, while τ_{ab} and τ_{bc} 'slide' on the fracture strength surface as in an ideal plastic material, and the unloading path follows reduced shear moduli to the zero shear stress and strain state. It should be noted that τ_{bc} is governed by both the failure functions and should lie within or on each of these two strength surfaces.

Fabric lamina model

The fiber failure criteria of Hashin for a unidirectional layer are generalized to characterize the fiber damage in terms of strain components for a plain weave layer. The fill and warp fiber tensile/shear failure are given by the quadratic interaction between the associated axial and shear stresses, i.e.

$$f_6 = \left(\frac{\langle \sigma_a \rangle}{S_{aT}} \right)^2 + \frac{\left(\tau_{ab}^2 + \tau_{ca}^2 \right)}{S_{aFS}^2} - 1 = 0$$

$$f_7 = \left(\frac{\langle \sigma_b \rangle}{S_{bT}} \right)^2 + \frac{\left(\tau_{ab}^2 + \tau_{bc}^2 \right)}{S_{bFS}^2} - 1 = 0$$

where S_{aT} and S_{bT} are the axial tensile strengths in the fill and warp directions, respectively, and S_{aFS} and S_{bFS} are the layer shear strengths due to fiber shear failure in the fill and warp directions. These failure criteria are applicable when the associated σ_a or σ_b is positive. It is assumed S_{aFS} = SFS, and

$$S_{bFS} = SFS * S_{bT} / S_{aT} .$$

When σ_a or σ_b is compressive, it is assumed that the in-plane compressive failure in both the fill and warp directions are given by the maximum stress criterion, i.e.

$$f_8 = \left[\frac{\langle \sigma_a^{'} \rangle}{S_{aC}} \right]^2 - 1 = 0, \quad \sigma_a^{'} = -\sigma_a + \langle -\sigma_c \rangle$$

$$f_9 = \left[\frac{\langle \sigma_b^{'} \rangle}{S_{bC}} \right]^2 - 1 = 0, \quad \sigma_b^{'} = -\sigma_b + \langle -\sigma_c \rangle$$

where S_{aC} and S_{bC} are the axial compressive strengths in the fill and warp directions, respectively. The crush failure under compressive pressure is

$$f_{10} = \left(\frac{\langle p \rangle}{S_{FC}} \right)^2 - 1 = 0, \quad p = -\frac{\sigma_a + \sigma_b + \sigma_c}{3}$$

A plain weave layer can fail under in-plane shear stress without the occurrence of fiber breakage. This in-plane matrix failure mode is given by

$$f_{11} = \left(\frac{\tau_{ab}}{S_{ab}} \right)^2 - 1 = 0$$

where S_{ab} is the layer shear strength due to matrix shear failure.

Another failure mode, which is due to the quadratic interaction between the thickness stresses, is expected to be mainly a matrix failure. This through the thickness matrix failure criterion is

$$f_{12} = S^2 \left\{ \left(\frac{\langle \sigma_c \rangle}{S_{cT}} \right)^2 + \left(\frac{\tau_{bc}}{S_{bc}} \right)^2 + \left(\frac{\tau_{ca}}{S_{ca}} \right)^2 \right\} - 1 = 0$$

where S_{cT} is the through the thickness tensile strength, and S_{bc}, and S_{ca} are the shear strengths assumed to depend on the compressive normal stress σ_c, i.e.,

$$\begin{Bmatrix} S_{ca} \\ S_{bc} \end{Bmatrix} = \begin{Bmatrix} S_{ca}^{(0)} \\ S_{bc}^{(0)} \end{Bmatrix} + \tan(\varphi)\langle -\sigma_c \rangle$$

When failure predicted by this criterion occurs within elements that are adjacent to the ply interface, the failure plane is expected to be parallel to the layering planes, and, thus, can be referred to as the delamination mode. Note that a scale factor S is introduced to provide better correlation of delamination area with experiments. The scale factor S can be determined by fitting the analytical prediction to experimental data for the delamination area.

Similar to the unidirectional model, when fiber tensile/shear failure is predicted in a layer by f_6 or f_7, the load carrying capacity of that layer in the associated direction is completely eliminated. For compressive fiber failure due to by f_8 or f_9, the layer is assumed to carry a residual axial load in the failed direction, while the load carrying capacity transverse to the failed direction is assumed unchanged. When the compressive axial stress in a layer reaches the compressive axial strength S_{aC} or S_{bC}, the axial layer stress is assumed to be reduced to the residual strength S_{aRC} or S_{bRC} where $S_{aRC} = SFFC * S_{aC}$ and $S_{bRC} = SFFC * S_{bC}$. The axial stress is assumed to remain constant, i.e., $\sigma_a = -S_{aCR}$ or $\sigma_b = -S_{bCR}$, for continuous compressive loading, while the subsequent unloading curve follows a reduced axial modulus. When the fiber crush failure is occurred, the material is assumed to behave elastically for compressive pressure, p > 0, and to carry no load for tensile pressure, p < 0.

When the in-plane matrix shear failure is predicted by f_{11} the axial load carrying capacity within a failed element is assumed unchanged, while the in-plane shear stress is assumed to be reduced to zero.

For through the thickness matrix (delamination) failure given by equations f_{12}, the in-plane load carrying capacity within the element is assumed to be elastic, while the strength values for the tensile mode, $S_{ca}^{(0)}$ and $S_{bc}^{(0)}$, are set to zero. For tensile mode, $\sigma_c > 0$, the through the thickness stress components are reduced to zero. For compressive mode, $\sigma_c < 0$, σ_c is assumed to be elastic, while τ_{bc} and τ_{ca} 'slide' on the fracture strength surface as in an ideal plastic material, and the unloading path follows reduced shear moduli to the zero shear stress and strain state.

The effect of strain-rate on the layer strength values of the fiber failure modes is modeled by the strain-rate dependent functions for the strength values $\{S_{RT}\}$ as

$$\{S_{RT}\} = \{S_0\}\left(1 + C_{rate1} \ln \frac{\{\dot{\bar{\varepsilon}}\}}{\dot{\varepsilon}_0} \right)$$

$$\{S_{RT}\} = \begin{Bmatrix} S_{aT} \\ S_{aC} \\ S_{bT} \\ S_{bC} \\ S_{FC} \\ S_{FS} \end{Bmatrix} \text{ and } \{\dot{\bar{\varepsilon}}\} = \begin{Bmatrix} |\dot{\varepsilon}_a| \\ |\dot{\varepsilon}_a| \\ |\dot{\varepsilon}_b| \\ |\dot{\varepsilon}_b| \\ |\dot{\varepsilon}_c| \\ \left(\dot{\varepsilon}_{ca}^2 + \dot{\varepsilon}_{bc}^2\right)^{1/2} \end{Bmatrix}$$

where C_{rate} is the strain-rate constants, and $\{S_0\}$ are the strength values of $\{S_{RT}\}$ at the reference strain-rate $\dot{\varepsilon}_0$.

Damage model

The damage model is a generalization of the layer failure model of Material 161 by adopting the MLT damage mechanics approach, Matzenmiller et al. [1995], for characterizing the softening behavior after damage initiation. Complete model description is given in Yen [2002]. The damage functions, which are expressed in terms of ply level engineering strains, are converted from the above failure criteria of fiber and matrix failure modes by neglecting the Poisson's effect. Elastic moduli reduction is expressed in terms of the associated damage parameters ϖ_i:

$$E_i^{'} = (1 - \varpi_i) E_i$$

$$\varpi_i = 1 - \exp\left(-r_i^{m_i} / m_i\right) \qquad r_i \geq 0 \quad i = 1,...,6$$

where E_i are the initial elastic moduli, $E_i^{'}$ are the reduced elastic moduli, r_i are the damage thresholds computed from the associated damage functions for fiber damage, matrix damage and delamination, and m_i are material damage parameters, which are currently assumed to be independent of strain-rate. The damage function is formulated to account for the overall nonlinear elastic response of a lamina including the initial 'hardening' and the subsequent softening beyond the ultimate strengths.

In the damage model (material 162), the effect of strain-rate on the nonlinear stress-strain response of a composite layer is modeled by the strain-rate dependent functions for the elastic moduli $\{E_{RT}\}$ as

$$\{E_{RT}\} = \{E_0\}\left(1 + \{C_{rate}\} \ln \frac{\{\dot{\bar{\varepsilon}}\}}{\dot{\varepsilon}_0}\right)$$

$$\{E_{RT}\} = \begin{Bmatrix} E_a \\ E_b \\ E_c \\ G_{ab} \\ G_{bc} \\ G_{ca} \end{Bmatrix}, \{\dot{\bar{\varepsilon}}\} = \begin{Bmatrix} |\dot{\varepsilon}_a| \\ |\dot{\varepsilon}_b| \\ |\dot{\varepsilon}_c| \\ |\dot{\varepsilon}_{ab}| \\ |\dot{\varepsilon}_{bc}| \\ |\dot{\varepsilon}_{ca}| \end{Bmatrix} \text{ and } \{C_{rate}\} = \begin{Bmatrix} C_{rate2} \\ C_{rate2} \\ C_{rate4} \\ C_{rate3} \\ C_{rate3} \\ C_{rate3} \end{Bmatrix}$$

where $\{C_{rate}\}$ are the strain-rate constants. $\{E_0\}$ are the modulus values of $\{E_{RT}\}$ at the reference strain-rate $\dot{\varepsilon}_0$.

Element Erosion:

A failed element is eroded in any of three different ways:

1. If fiber tensile failure in a unidirectional layer is predicted in the element and the axial tensile strain is greater than E_LIMT. For a fabric layer, both in-plane directions are failed and exceed E_LIMT.
2. If compressive relative volume in a failed element is smaller than ECRSH.
3. If tensile relative volume in a failed element is greater than EEXPN.

Damage History Parameters:

Information about the damage history variables for the associated failure modes can be plotted in LS-PrePost. These additional history variables are tabulated below:

History Variable	Description	Value	LS-PrePost History Variable
1. efa(I)	Fiber mode in a		7
2. efb(I)	Fiber mode in b	0-elastic	8
3. efp(I)	Fiber crush mode		9
4. em(I)	Perpendicular matrix mode	≥1-failed	10
5. ed(I)	Parallel matrix/ delamination mode		11
6. delm(I)	delamination mode		12

*MAT_MODIFIED_CRUSHABLE_FOAM

This is Material Type 163 which is dedicated to modeling crushable foam with optional damping, tension cutoff, and strain rate effects. Unloading is fully elastic. Tension is treated as elastic-perfectly-plastic at the tension cut-off value.

Card 1

Variable	MID	RO	E	PR	TID	TSC	DAMP	NCYCLE
Type	A8	F	F	F	F	F	F	F
Default	none	none	none	none	none	0.0	0.10	12.

Card 2

Variable	SRCLMT	SFLAG						
Type	F	I						
Default	1.E+20	0						

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
E	Young's modulus
PR	Poisson's ratio
TID	Table ID defining yield stress versus volumetric strain, γ, at different strain rates.
TSC	Tensile stress cutoff. A nonzero, positive value is strongly recommended for realistic behavior.
DAMP	Rate sensitivity via damping coefficient (.05<recommended value<.50).
NCYCLE	Number of cycles to determine the average volumetric strain rate.
SRCLMT	Strain rate change limit.

SFLAG	The strain rate in the table may be the true strain rate (SFLAG=0) or the engineering strain rate (SFLAG=1).

Remarks:

The volumetric strain is defined in terms of the relative volume, V, as:

$$\gamma = 1.-V$$

The relative volume is defined as the ratio of the current to the initial volume. In place of the effective plastic strain in the D3PLOT database, the integrated volumetric strain is output.

This material is an extension of material 63, *MAT_CRUSHABLE_FOAM. It allows the yield stress to be a function of both volumetric strain rate and volumetric strain. Rate effects are accounted for by defining a table of curves using *DEFINE_TABLE. Each curve defines the yield stress versus volumetric strain for a different strain rate. The yield stress is obtained by interpolating between the two curves that bound the strain rate.

To prevent high frequency oscillations in the strain rate from causing similar high frequency oscillations in the yield stress, a modified volumetric strain rate is used when interpolating to obtain the yield stress. The modified strain rate is obtained as follows. If NYCLE is >1, then the modified strain rate is obtained by a time average of the actual strain rate over NCYCLE solution cycles. For SRCLMT>0, the modified strain rate is capped so that during each cycle, the modified strain rate is not permitted to change more than SRCLMT multiplied by the solution time step.

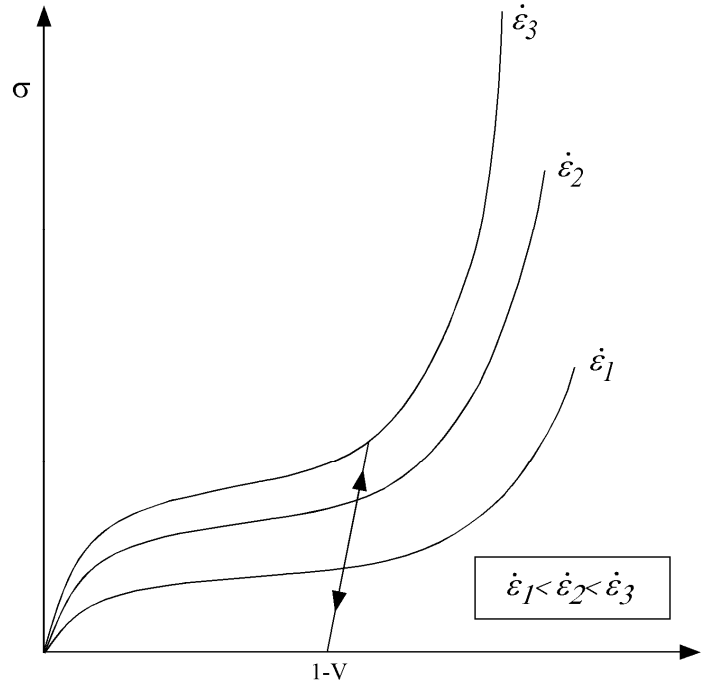


Figure 163.1. Rate effects are defined by a family of curves giving yield stress versus volumetric strain where *V* is the relative volume.

*MAT_BRAIN_LINEAR_VISCOELASTIC

This is Material Type 164. This material is a Kelvin-Maxwell model for modeling brain tissue, which is valid for solid elements only. See Remarks below.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	BULK	G0	GI	DC	FO	SO
Type	A8	F	F	F	F	F	F	F
Default	none	none	none	none	none	0.0	0.0	0.0

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
BULK	Bulk modulus (elastic)
G0	Short-time shear modulus, G_0
GI	Long-time (infinite) shear modulus, G_∞
DC	Maxwell decay constant, β[FO=0.0] or Kelvin relaxation constant, τ [FO=1.0]
FO	Formulation option: EQ.0.0: Maxwell, EQ.1.0: Kelvin.
SO	Strain (logarithmic) output option to control what is written as component 7 to the d3plot database. (LS-Prepost always blindly labels this component as effective plastic strain.) The maximum values are updated for each element each time step: EQ.0.0: maximum principal strain that occurs during the calculation, EQ.1.0: maximum magnitude of the principal strain values that occurs during the calculation, EQ.2.0: maximum effective strain that occurs during the calculation.

Remarks:

The shear relaxation behavior is described for the Maxwell model by:

$$G(t) = G + (G_0 - G_\infty)e^{-\beta t}$$

A Jaumann rate formulation is used

$$\overset{\nabla}{\sigma}'_{ij} = 2\int_0^t G(t-\tau)\, D'_{ij}(\tau)\, dt$$

where the prime denotes the deviatoric part of the stress rate, $\overset{\nabla}{\sigma}_{ij}$, and the strain rate D_{ij}. For the Kelvin model the stress evolution equation is defined as:

$$\dot{s}_{ij} + \frac{1}{\tau} s_{ij} = (1+\delta_{ij}) G_0 \dot{e}_{ij} + (1+\delta_{ij}) \frac{G_\infty}{\tau} \dot{e}_{ij}$$

The strain data as written to the d3plot database may be used to predict damage, see [Bandak 1991].

*MAT_PLASTIC_NONLINEAR_KINEMATIC

This is Material Type 165. This relatively simple model, based on a material model by Lemaitre and Chaboche [1990], is suited to model nonlinear kinematic hardening plasticity. The model accounts for the nonlinear Bauschinger effect, cyclic hardening, and ratcheting. Huang [2006] programmed this model and provided it as a user subroutine. It is a very cost effective model and is available shell and solid elements. This material model is available starting with the R3 release of Version 971.

Card 1

Variable	MID	RO	E	PR	SIGY	H	C	GAMMA
Type	A8	F	F	F	F	F	F	F
Default	none	none	none	none	none	0.0	0.0	0.0

Card 2

Variable	FS							
Type	F							
Default	1.E+16							

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.
SIGY	Initial yield stress, σ_{y0}.
H	Isotropic plastic hardening modulus
C	Kinematic hardening modulus

VARIABLE	DESCRIPTION
GAMMA	Kinematic hardening parameter, γ.
FS	Failure strain for eroding elements.

Remarks:

If the isotropic hardening modulus, *H*, is nonzero, the size of the surface increases as function of the equivalent plastic strain, ε^p:

$$\sigma_y = \sigma_{y0} + H\varepsilon^p$$

The rate of evolution of the kinematic component is a function of the plastic strain rate:

$$\dot{\alpha} = \left[Cn - \gamma\alpha\right]\dot{\varepsilon}^p$$

where, *n*, is the flow direction. The term, $\gamma\alpha\dot{\varepsilon}^p$, introduces the nonlinearity into the evolution law, which becomes linear if the parameter, γ, is set to zero.

*MAT_MOMENT_CURVATURE_BEAM

This is Material Type 166. This material is for performing nonlinear elastic or multi-linear plastic analysis of Belytschko-Schwer beams with user-defined axial force-strain, moment curvature and torque-twist rate curves. If strain, curvature or twist rate is located outside the curves, use extrapolation to determine the corresponding rigidity. For multi-linear plastic analysis, the user-defined curves are used as yield surfaces.

Card 1

Variable	MID	RO	E	ELAF	EPFLG	CTA	CTB	CTT
Type	A8	F	F	F	F	F	F	F
Default	none	none	none	none	0.0	0.0	0.0	0.0

Card 2

Variable	N1	N2	N3	N4	N5	N6	N7	N8
Type	F	F	F	F	F	F	F	F
Default	none	none	0.0/none	0.0	0.0	0.0	0.0	0.0

Card 3

Variable	LCMS1	LCMS2	LCMS3	LCMS4	LCMS5	LCMS6	LCMS7	LCMS8
Type	F	F	F	F	F	F	F	F
Default	none	none	0.0/none	0.0	0.0	0.0	0.0	0.0

Card 4	1	2	3	4	5	6	7	8
Variable	LCMT1	LCMT2	LCMT3	LCMT4	LCMT5	LCMT6	LCMT7	LCMT8
Type	F	F	F	F	F	F	F	F
Default	none	none	0.0/none	0.0	0.0	0.0	0.0	0.0

Card 5

Variable	LCT1	LCT2	LCT3	LCT4	LCT5	LCT6	LCT7	LCT8
Type	F	F	F	F	F	F	F	F
Default	none	none	0.0/none	0.0	0.0	0.0	0.0	0.0

Card 6 is for multi-linear plastic analysis only.

Card 6	1	2	3	4	5	6	7	8
Variable	CFA	CFB	CFT	HRULE	REPS	RBETA	RCAPAY	RCAPAZ
Type	F	F	F	F	F	F	F	F
Default	1.0	1.0	1.0	0.0	1.0E+20	1.0E+20	1.0E+20	1.0E+20

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
E	Young's modulus. This variable controls the time step size and must be chosen carefully. Increasing the value of E will decrease the time step size.
ELAF	Load curve ID for the axial force-strain curve
EPFLG	Function flag EQ.0.0: nonlinear elastic analysis EQ.1.0: multi-linear plastic analysis

VARIABLE	DESCRIPTION
CTA, CTB, CTT	Type of axial force-strain, moment-curvature, and torque-twist rate curves EQ.0.0: curve is symmetric EQ.1.0: curve is asymmetric For symmetric curves, all data point must be in the first quadrant and at least three data points need to be given, starting from the origin, ensued by the yield point. For asymmetric curves, at least five data points are needed and exactly one point must be at the origin. The two points on both sides of the origin record the positive and negative yield points. The last data point(s) has no physical meaning: it serves only as a control point for inter or extrapolation. The curves are input by the user and treated in LS-DYNA as a linearly piecewise function. The curves must be monotonically increasing, while the slopes must be monotonically decreasing.
N1-N8	Axial forces at which moment-curvature curves are given. The axial forces must be ordered monotonically increasing. At least two axial forces must be defined if the curves are symmetric. At least three axial forces must be defined if the curves are asymmetric.
LCMS1-LCMS8	Load curve IDs for the moment-curvature curves about axis S under corresponding axial forces.
LCMT1-LCMT8	Load curve IDs for the moment-curvature curves about axis T under corresponding axial forces.
LCT1-LCT8	Load curve IDs for the torque-twist rate curves under corresponding axial forces.
CFA, CFB, CFT	For multi-linear plastic analysis only. Ratio of axial, bending and torsional elastic rigidities to their initial values, no less than 1.0 in value.
HRULE	Hardening rule, for multi-linear plastic analysis only. EQ.0.0: isotropic hardening EQ.1.0: kinematic hardening In between: mixed hardening
REPS	Rupture effective plastic axial strain
RBETA	Rupture effective plastic twist rate
RCAPAY	Rupture effective plastic curvature about axis S
RCAPAZ	Rupture effective plastic curvature about axis T

*MAT_MCCORMICK

This is Material Type 167. This is a constitute model for finite plastic deformities in which the material's strength is defined by McCormick's constitutive relation for materials exhibiting negative steady-state Strain Rate Sensitivity (SRS). McCormick [1988] and Zhang, McCormick and Estrin [2001].

Card 1

Variable	MID	RO	E	PR	SIGY			
Type	A8	F	F	F	F			

Card 2

Variable	Q1	C1	Q2	C2				
Type	F	F	F	F				

Card 3

Variable	S	H	OMEGA	TD	ALPHA	EPS0		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.
SIGY	Initial yield stress
Q1	Isotropic hardening parameter, Q_1

VARIABLE	DESCRIPTION
C1	Isotropic hardening parameter, C_1
Q2	Isotropic hardening parameter, Q_2
C2	Isotropic hardening parameter, C_2
S	Dynamic strain aging parameter, S
H	Dynamic strain aging parameter, H
OMEGA	Dynamic strain aging parameter, Ω
TD	Dynamic strain aging parameter, t_d
ALPHA	Dynamic strain aging parameter, α
EPS0	Reference strain rate, $\dot{\varepsilon}_0$

Remarks:

The uniaxial stress-strain curve is given in the following form:

$$\sigma\left(\varepsilon^p, \dot{\varepsilon}^p\right) = \sigma_Y\left(t_a\right) + R\left(\varepsilon^p\right) + \sigma_v\left(\dot{\varepsilon}^p\right)$$

Viscous stress σ_v is given by

$$\sigma_v\left(\dot{\varepsilon}^p\right) = S \ln\left(1 + \frac{\dot{\varepsilon}^p}{\dot{\varepsilon}_o}\right)$$

where S represents the instantaneous strain rate sensitivity and $\dot{\varepsilon}_o$ is a reference strain rate.

In the McCormick model the yield strength including the contribution from dynamic strain again (DSA) is defined as

$$\sigma_Y\left(t_a\right) = \sigma_o + SH\left[1 - \exp\left\{-\left(\frac{t_a}{t_d}\right)^{\alpha}\right\}\right]$$

where σ_o is the yield strength for vanishing average waiting time t_a, and H, α, and t_d are material constants linked to dynamic strain aging.

The average waiting time is defined by the evolution equation

$$\dot{t}_a = 1 - \frac{t_a}{t_{a,ss}}$$

where the quasi-steady state waiting time $t_{a,ss}$ is given as

$$t_{a,ss} = \frac{\Omega}{\dot{\varepsilon}^p}.$$

The strain hardening function R is defined by the extended Voce Law

$$R\left(\varepsilon^p\right) = Q_1\left[1 - \exp\left(-C_1\varepsilon^p\right)\right] + Q_2\left[1 - \exp\left(-C_2\varepsilon^p\right)\right].$$

*MAT_POLYMER

This is material type 168. This model is implemented for brick elements.

Card 1

Variable	MID	RO	E	PR	GAMMA0	DG	SC	ST
Type	A8	F	F	F	F	F	F	F

Card 2

Variable	TEMP	K	CR	N	C			
Type	F	F	F	F	F			

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass Density.
E	Young's modulus, E.
PR	Poisson's ratio, ν.
GAMMA0	Pre-exponential factor, $\dot{\gamma}_{0A}$.
DG	Energy barrier to flow, ΔG.
SC	Shear resistance in compression, S_c.
ST	Shear resistance in tension, S_t.
TEMP	Absolute temperature, θ.
K	Boltzmann constant, k.

VARIABLE	DESCRIPTION
CR	Product, $C_r = nk\theta$.
N	Number of 'rigid links' between entanglements, N.
C	Relaxation factor, C.

Remarks:

The polymer is assumed to have two basic resistances to deformation:

1. An inter-molecular barrier to deformation related to relative movement between molecules.
2. An evolving anisotropic resistance related to straightening of the molecule chains.

The model which is implemented and presented in this paper is mainly based on the framework suggested by Boyce et al. [2000]. Going back to the original work by Haward and Thackray [1968], they considered the uniaxial case only. The extension to a full 3D formulation was proposed by Boyce et al. [1988]. Moreover, Boyce and co-workers have during a period of 20 years changed or further developed the parts of the original model. Haward and Thackray [1968] used an Eyring model to represent the dashpot in Fig. 168.1, while Boyce et al. [2000] emplyed the double-kink model of Argon [1973] instead. Part B of the model, describing the resistance associated with straightening of the molecules, contained originally a one-dimensional Langevin spring [Haward and Thackray, 1968], which was generalized to 3D with the eight-chain model by Arruda and Boyce [1993].

The main structure of the model presented by Boyce et al. [2000] is kept for this model. Recognizing the large elastic deformations occurring for polymers, a formulation based on a Neo-Hookean material is here selected for describing the spring in resistance A in Figure 168.1.

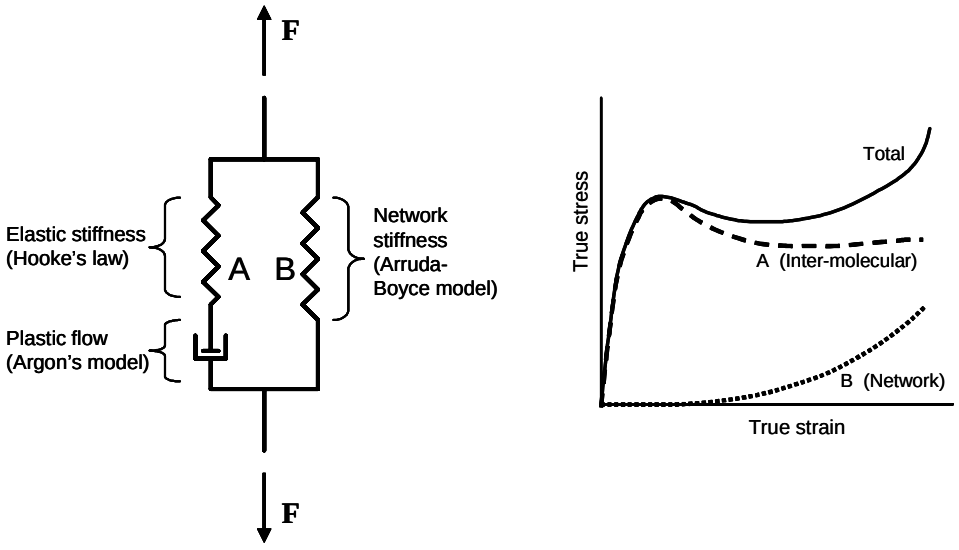


Figure 168.1. Stress decomposition in inter-molecular and network contributions.

Referring to Fig. 1, it is assumed that the deformation gradient tensor is the same for the two resistances (Part A and B)

$$\mathbf{F} = \mathbf{F}_A = \mathbf{F}_B$$

while the Cauchy stress tensor for the system is assumed to be the sum of the Cauchy stress tensors for the two parts

$$\boldsymbol{\sigma} = \boldsymbol{\sigma}_A + \boldsymbol{\sigma}_B$$

Part A: Inter-molecular resistance

The deformation is decomposed into elastic and plastic parts, $\mathbf{F}_A = \mathbf{F}_A^e \cdot \mathbf{F}_A^p$, where it is assumed that the intermediate configuration $\bar{\Omega}_A$ defined by $\mathbf{F}_A^p$ is invariant to rigid body rotations of the current configuration. The velocity gradient in the current configuration Ω is defined by

$$\mathbf{L}_A = \dot{\mathbf{F}}_A \cdot \mathbf{F}_A^{-1} = \mathbf{L}_A^e + \mathbf{L}_A^p$$

Owing to the decomposition, $\mathbf{F}_A = \mathbf{F}_A^e \cdot \mathbf{F}_A^p$, the elastic and plastic rate-of-deformation and spin tensors are defined by

$$\mathbf{L}_A^e = \mathbf{D}_A^e + \mathbf{W}_A^e = \dot{\mathbf{F}}_A^e \cdot (\mathbf{F}_A^e)^{-1}$$

$$\mathbf{L}_A^p = \mathbf{D}_A^p + \mathbf{W}_A^p = \mathbf{F}_A^e \cdot \dot{\mathbf{F}}_A^p \cdot (\mathbf{F}_A^p)^{-1} \cdot (\mathbf{F}_A^e)^{-1} = \mathbf{F}_A^e \cdot \bar{\mathbf{L}}_A^p \cdot (\mathbf{F}_A^e)^{-1}$$

where $\bar{\mathbf{L}}_A^p = \dot{\mathbf{F}}_A^p \cdot (\mathbf{F}_A^p)^{-1}$. The Neo-Hookean material represents an extension of Hooke's law to large elastic deformations and may be chosen for the elastic part of the deformation when the elastic behavior is assumed to be isotropic.

$$\boldsymbol{\tau}_A = \lambda_0 \ln J_A^e \, \mathbf{I} + \mu_0 (\mathbf{B}_A^e - \mathbf{I})$$

where $\boldsymbol{\tau}_A = J_A \boldsymbol{\sigma}_A$ is the Kirchhoff stress tensor of Part A and $J_A^e = \sqrt{\det \mathbf{B}_A^e} = J_A$ is the Jacobian determinant. The elastic left Cauchy-Green deformation tensor is given by $\mathbf{B}_A^e = \mathbf{F}_A^e \cdot \mathbf{F}_A^{eT}$.

The flow rule is defined by

$$\mathbf{L}_A^p = \dot{\gamma}_A^p \mathbf{N}_A$$

Where

$$\mathbf{N}_A = \frac{1}{\sqrt{2}\,\tau_A} \boldsymbol{\tau}_A^{dev}, \quad \tau_A = \sqrt{\frac{1}{2} \mathrm{tr} \left(\boldsymbol{\tau}_A^{dev} \right)^2}$$

and $\boldsymbol{\tau}_A^{dev}$ is the stress deviator. The rate of flow is taken to be a thermally activated process

$$\dot{\gamma}_A^p = \dot{\gamma}_{0A} \exp\left[-\frac{\Delta G\left(1-\tau_A / s\right)}{k\theta} \right]$$

where $\dot{\gamma}_{0A}$ is a pre-exponential factor, ΔG is the energy barrier to flow, s is the shear resistance, k is the Boltzmann constant and θ is the absolute temperature. The shear resistance s is assumed to depend on the stress triaxiality σ^*,

$$s = s\left(\sigma^*\right), \quad \sigma^* = \frac{\operatorname{tr} \boldsymbol{\sigma}_A}{3\sqrt{3}\tau_A}$$

The exact dependence is given by a user-defined load curve, which is linear between the shear resistances in compression and tension. These resistances are denoted s_c and s_t, respectively.

Part B: Network resistance

The network resistance is assumed to be nonlinear elastic with deformation gradient $\mathbf{F}_B = \mathbf{F}_B^N$, i.e. any viscoplastic deformation of the network is neglected. The stress-stretch relation is defined by

$$\boldsymbol{\tau}_B = \frac{nk\theta}{3} \frac{\sqrt{N}}{\bar{\lambda}_N} \mathcal{L}^{-1}\left(\frac{\bar{\lambda}_N}{\sqrt{N}} \right) (\bar{\mathbf{B}}_B^N - \bar{\lambda}_N^2 \mathbf{I})$$

where $\boldsymbol{\tau}_B = J_B \boldsymbol{\sigma}_B$ is the Kirchhoff stress for Part B, n is the chain density and N the number of 'rigid links' between entanglements. In accordance with Boyce et. al [2000], the product, $nk\theta$ is denoted C_R herein. Moreover, $\mathcal{L}^{-1}$ is the inverse Langevin function, $\mathcal{L}(\beta) = \coth\beta - 1/\beta$, and further

$$\bar{\mathbf{B}}_B^N = \bar{\mathbf{F}}_B^N \cdot \bar{\mathbf{F}}_B^{NT}, \quad \bar{\mathbf{F}}_B^N = J_B^{-1/3} \mathbf{F}_B^N, \quad J_B = \det \mathbf{F}_B^N, \quad \bar{\lambda}_N = \left[\frac{1}{3} \operatorname{tr} \bar{\mathbf{B}}_B^N \right]^{\frac{1}{2}}$$

The flow rule defining the rate of molecular relaxation reads

$$\mathbf{L}_B^F = \dot{\gamma}_B^F \mathbf{N}_B$$

Where

$$\mathbf{N}_B = \frac{1}{\sqrt{2}\,\tau_B} \boldsymbol{\tau}_B^{dev}, \quad \tau_B = \sqrt{\frac{1}{2} \boldsymbol{\tau}_B^{dev} : \boldsymbol{\tau}_B^{dev}}$$

The rate of relaxation is taken equal to

$$\dot{\gamma}_B^F = C\left(\frac{1}{\bar{\lambda}_F - 1}\right)\tau_B$$

Where

$$\bar{\lambda}_F = \left[\frac{1}{3}\mathrm{tr}\left(\mathbf{F}_B^F\left\{\mathbf{F}_B^F\right\}^T\right)\right]^{\frac{1}{2}}$$

The model has been implemented into LS-DYNA using a semi-implicit stress-update scheme [Moran et. al 1990], and is available for the explicit solver only.

*MAT_ARUP_ADHESIVE

This is Material Type 169. This material model was written for adhesive bonding in aluminum structures. The plasticity model is not volume-conserving, and hence avoids the spuriously high tensile stresses that can develop if adhesive is modeled using traditional elasto-plastic material models. It is available **only** for solid elements of formulations 1 and 2, and it is assumed that the smallest dimension of the element is the through-thickness dimension of the bond.

Note: This Material Type will be available starting in release 3 of version 971.

Card 1

Variable	MID	RO	E	PR	TENMAX	GCTEN	SHRMAX	GCSHR
Type	A8	F	F	F	F	F	F	F
Default	none	none	none	none	1.e20	1.e20	1.e20	1.e20

Card 2

Variable	PWRT	PWRS	SHRP	SHT_SL	EDOT0	EDOT2	THKDIR	XEDGE
Type	F	F	F	F	F	F	F	F
Default	2.0	2.0	0.0	0.0	1.0	0.0	0.0	0.0

Define Card 3 and 4 only if XEDGE=1, otherwise omit both cards

Card 3

Variable	TMAXE	GCTE	SMAXE	GCSE	PWRTE	PWRSE		
Type	F	F	F	F	F	F		
Default	1.e20	1.e20	1.e20	1.e20	2.0	2.0		

Card 4 Format

Card 4	1	2	3	4	5	6	7	8
Variable	FACET	FACCT	FACES	FACCS	SOFTT	SOFTS		
Type	F	F	F	F	F	F		
Default	1.0	1.0	1.0	1.0	1.0	1.0		

Card 5: Define the following card for rate effects only if EDOT2 is non-zero

Card 5	1	2	3	4	5	6	7	8
Variable	SDFAC	SGFAC	SDEFAC	SGEFAC				
Type	F	F	F	F				
Default	1.0	1.0	1.0	1.0				

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.
TENMAX	Maximum through-thickness tensile stress
GCTEN	Energy per unit area to fail the bond in tension
SHRMAX	Maximum through-thickness shear stress
GCSHR	Energy per unit area to fail the bond in shear
PWRT	Power law term for tension
PWRS	Power law term for shear

VARIABLE	DESCRIPTION
SHRP	Shear plateau ratio (Optional)
SHT_SL	Slope (non-dimensional) of yield surface at zero tension (See Remarks)
EDOT0	Strain rate at which the “static” properties apply
EDOT2	Strain rate at which the “dynamic” properties apply (Card 5)
THKDIR	Through-thickness direction flag (See remarks) EQ.0.0: smallest element dimension (default) EQ.1.0: direction from nodes 1-2-3-4 to nodes 5-6-7-8
Data for interfacial failure based on stress concentrations at edges of joint:	
XEDGE	Flag = 1.0 to input interfacial failure properties (Cards 3 and 4)
TMAXE	Maximum tensile force per unit length on edges of joint
GCTE	Energy per unit length to fail the edge of the bond in tension
SMAXE	Maximum shear force per unit length on edges of joint
GCSE	Energy per unit length to fail the edge of the bond in shear
PWRTE	Power law term for tension
PWRSE	Power law term for shear
FACET	Stiffness scaling factor for edge elements - tension
FACCT	Stiffness scaling factor for interior elements - tension
FACES	Stiffness scaling factor for edge elements - shear
FACCS	Stiffness scaling factor for interior elements - shear
SOFTT	Factor by which the tensile strength is reduced when a neighbor fails
SOFTS	Factor by which the shear strength is reduced when a neighbor fails
Data for rate effects (Card 5)	
SDFAC	Factor on TENMAX and SHRMAX at strain rate EDOT2
SGFAC	Factor on GCTEN and GCSHR at strain rate EDOT2
SDEFAC	Factor on TMAXE and SMAXE at strain rate EDOT2
SDGFAC	Factor on GCTE and GCSE at strain rate EDOT2

Remarks:

The through-thickness direction is identified from the smallest dimension of each element by default (THKDIR=0.0). It is expected that this dimension will be smaller than in-plane dimensions (typically 1-2mm compared with 5-10mm). If this is not the case, one can set the through-thickness direction via element numbering (THKDIR=1.0). Then the thickness direction is expected to point from lower face (nodes 1-2-3-4) to upper face (nodes 5-6-7-8).

In-plane stresses are set to zero: it is assumed that the stiffness and strength of the substrate is large compared with that of the adhesive, given the relative thicknesses

If the substrate is modeled with shell elements, it is expected that these will lie at the mid-surface of the substrate geometry. Therefore the solid elements representing the adhesive will be thicker than the actual bond. If the elastic compliance of the bond is significant, this can be corrected by increasing the elastic stiffness property E.

The yield and failure surfaces are treated as a power-law combination of direct tension and shear across the bond:

$$\left(\frac{\sigma}{\sigma_{\max}}\right)^{PWRT} + \left(\frac{\tau}{\tau_{\max} - SHT\_SL * \sigma}\right)^{PWRS} = 1.0$$

At yield SHT_SL is the slope of the yield surface at $\sigma = 0$.

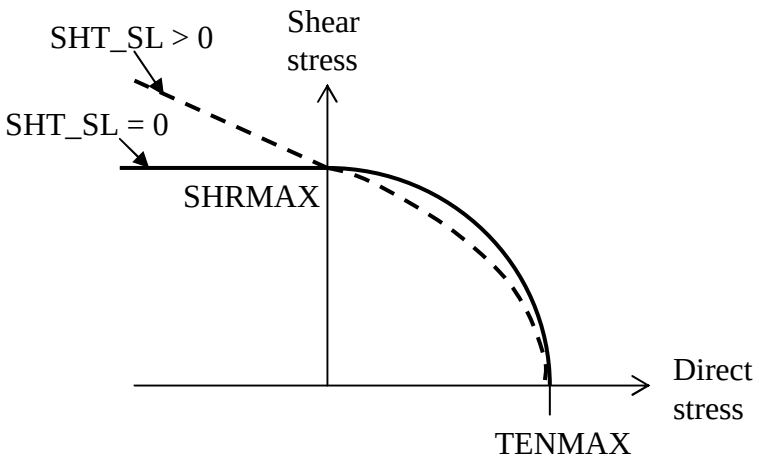


Figure 169.1

The stress-displacement curves for tension and shear are shown in the diagrams below. In both cases, Gc is the area under the curve.

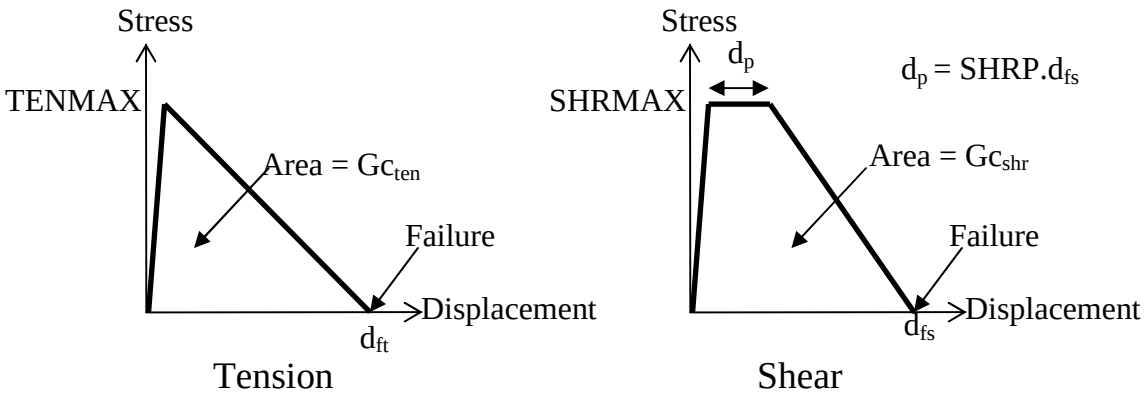


Figure 169.2

Because of the algorithm used, yielding in tension across the bond does not require strains in the plane of the bond – unlike the plasticity models, plastic flow is not treated as volume-conserving.

The Plastic Strain output variable has a special meaning:

$0 < ps < 1$: ps is the maximum value of the yield function experienced since time zero

$1 < ps < 2$: the element has yielded and the strength is reducing towards failure – yields at ps=1, fails at ps=2.

The damage cause by cohesive deformation (0 at first yield to 1 at failure) and by interfacial deformation are stored in the first two extra history variables. These can be plotted if NEIPH on *DATABASE_EXTENT_BINARY is 2 or more. By this means, the reasons for failure may be assessed.

When the plastic strain rate rises above EDOT0, rate effects are assumed to scale with log(plastic strain rate), as in the example below for cohesive tensile strength with dynamic factor SDFAC. The same form of relationship is applied for the other dynamic factors. If EDOT0 is zero or blank, no rate effects are applied.

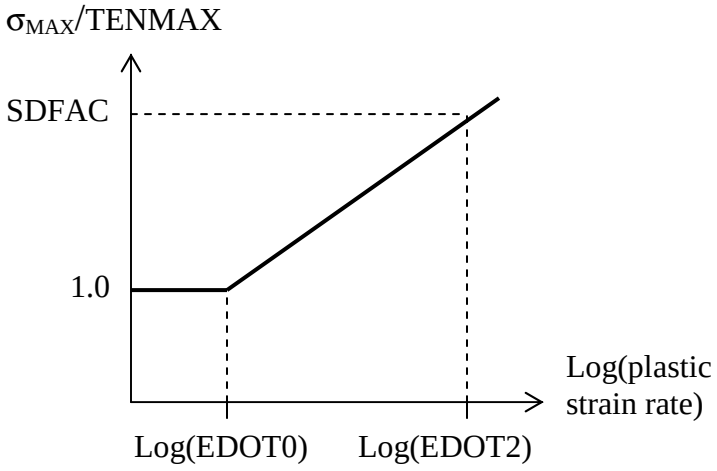


Figure 169.3

Rate effects are applied using the viscoplastic method.

Interfacial failure is assumed to arise from stress concentrations at the edges of the bond – typically the strength of the bond becomes almost independent of bond length. This type of failure is usually more brittle than cohesive failure. To simulate this, LS-DYNA identifies the free edges of the bond (made up of element faces that are not shared by other elements of material type *MAT_ARUP_ADHESIVE, excluding the faces that bond to the substrate). Only these elements can fail initially. The neighbors of failed elements can then develop free edges and fail in turn. In real adhesive bonds, the stresses at the edges can be concentrated over very small areas; in typical finite element models the elements are much too large to capture this. Therefore the concentration of loads onto the edges of the bond is accomplished artificially, by stiffening elements containing free edges (e.g. FACET, FACES >1) and reducing the stiffness of interior elements (e.g. FACCT, FACCS <1). Interior elements are allowed to yield at reduced loads (equivalent to TMAXE*FACET/FACCT and SMAXE*FACES/FACCS) – this is to prevent excessive stresses developing before the edge elements have failed - but cannot be damaged until they become edge elements after the failure of their neighbors.

*MAT_RESULTANT_ANISOTROPIC

This is Material Type 170. This model is available the Belytschko-Tsay and the C0 triangular shell elements and is based on a resultant stress formulation. In-plane behavior is treated separately from bending in order to model perforated materials such as television shadow masks. The plastic behavior of each resultant is specified with a load curve and is completely uncoupled from the other resultants. If other shell formulations are specified, the formulation will be automatically switched to Belytschko-Tsay. As implemented, this material model cannot be used with user defined integration rules.

Card 1

Variable	MID	RO						
Type	A8	F						

Card 2

Variable	E11P	E22P	V12P	V21P	G12P	G23P	G31P	
Type	F	F	F	F	F	F	F	

Card 3

Variable	E11B	E22B	V12B	V21B	G12B	AOPT		
Type	F	F	F	F	F	F		

Card 4

Variable	LN11	LN22	LN12	LQ1	LQ2	LM11	LM22	LM12
Type	F	F	F	F	F	F	F	F

Card 5	1	2	3	4	5	6	7	8
Variable				A1	A2	A3		
Type				F	F	F		

Card 6

Variable	V1	V2	V3	D1	D2	D3	BETA	
Type	F	F	F	F	F	F	F	

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
E11P	E_{11p}, for in plane behavior.
E22P	E_{22p}, for in plane behavior.
V12P	ν_{12p}, for in plane behavior.
V11P	ν_{21p}, for in plane behavior.
G12P	G_{12p}, for in plane behavior.
G23P	G_{23p}, for in plane behavior.
G31P	G_{31p}, for in plane behavior.
E11B	E_{11b}, for bending behavior.
E22B	E_{22b}, for bending behavior.
V12B	ν_{12b}, for bending behavior.
V21B	ν_{21b}, for bending behavior.

VARIABLE	DESCRIPTION
G12B	G_{12b}, for bending behavior.
AOPT	Material axes option (see MAT_*OPTION* TROPIC_ELASTIC for a more complete description): EQ.0.0: locally orthotropic with material axes determined by element nodes 1, 2, and 4, as with *DEFINE_COORDINATE_NODES. EQ.2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR. EQ.3.0: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle, BETA, from a line in the plane of the element defined by the cross product of the vector **v** with the element normal. LT.0.0: the absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_NODES, *DEFINE_COORDINATE_SYSTEM or *DEFINE_COORDINATE_VECTOR). Available in R3 version of 971 and later.
LN11	Yield curve ID for N_{11}.
LN22	Yield curve ID for N_{22}.
LN12	Yield curve ID for N_{12}.
LQ1	Yield curve ID for Q_1.
LQ2	Yield curve ID for Q_2.
LM11	Yield curve ID for M_{11}.
LM22	Yield curve ID for M_{22}.
LM12	Yield curve ID for M_{12}.
A1,A2,A3	a_1 a_2 a_3, define components of vector **a** for AOPT = 2.
V1,V2,V3	v_1 v_2 v_3, define components of vector **v** for AOPT = 3.
D1,D2,D3	d_1 d_2 d_3, define components of vector **d** for AOPT = 2.
BETA	Material angle in degrees for AOPT = 3, may be overridden on the element card, see *ELEMENT_SHELL_BETA.

Remarks:

The in-plane elastic matrix for in-plane, plane stress behavior is given by:

$$C_{in\,plane} = \begin{bmatrix} Q_{11p} & Q_{12p} & 0 & 0 & 0 \\ Q_{12p} & Q_{22p} & 0 & 0 & 0 \\ 0 & 0 & Q_{44p} & 0 & 0 \\ 0 & 0 & 0 & Q_{55p} & 0 \\ 0 & 0 & 0 & 0 & Q_{66p} \end{bmatrix}$$

The terms Q_{ijp} are defined as:

$$Q_{11p} = \frac{E_{11p}}{1-\nu_{12p}\nu_{21p}}$$

$$Q_{22p} = \frac{E_{22p}}{1-\nu_{12p}\nu_{21p}}$$

$$Q_{12p} = \frac{\nu_{12p}E_{11p}}{1-\nu_{12p}\nu_{21p}}$$

$$Q_{44p} = G_{12p}$$

$$Q_{55p} = G_{23p}$$

$$Q_{66p} = G_{31p}$$

The elastic matrix for bending behavior is given by:

$$C_{bending} = \begin{bmatrix} Q_{11b} & Q_{12b} & 0 \\ Q_{12b} & Q_{22b} & 0 \\ 0 & 0 & Q_{44b} \end{bmatrix}$$

The terms Q_{ijp} are similarly defined.

*MAT_STEEL_CONCENTRIC_BRACE

This is Material Type 171. It represents the cyclic buckling and tensile yielding behavior of steel braces and is intended primarily for seismic analysis. Use only for beam elements with ELFORM=2 (Belytschko-Schwer beam).

Note: This Material Type will be available starting in release 3 of version 971.

Card 1

Variable	MID	RO	YM	PR	SIGY	LAMDA	FBUCK	FBUCK2
Type	A8	F	F	F	F	F	F	F
Default	none	none	none	none	none	See Remarks	See Remarks	0.0

Card 2

Variable	CCBRF	BCUR						
Type	F	F						
Default	See Remarks							

Card 3

Variable	TS1	TS2	TS3	TS4	CS1	CS2	CS3	CS4
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	=TS1	=TS2	=TS3	=TS4

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
YM	Young's Modulus
PR	Poisson's Ratio
SIGY	Yield stress
LAMDA	Slenderness ratio (optional – see note)
FBUCK	Initial buckling load (optional – see note. If used, should be positive)
FBUCK2	Optional extra term in initial buckling load – see note
CCBRF	Reduction factor on initial buckling load for cyclic behavior
BCUR	Optional load curve giving compressive buckling load (y-axis) versus compressive strain (x-axis - both positive)
TS1-TS4	Tensile axial strain thresholds 1 to 4
CS1-CS4	Compressive axial strain thresholds 1 to 4

Remarks:

The brace element is intended to represent the buckling, yielding and cyclic behavior of steel elements such as tubes or I-sections that carry only axial loads. Empirical relationships are used to determine the buckling and cyclic load-deflection behavior. A single beam element should be used to represent each structural element.

The cyclic behavior is shown in the graph (compression shown as negative force and displacement).

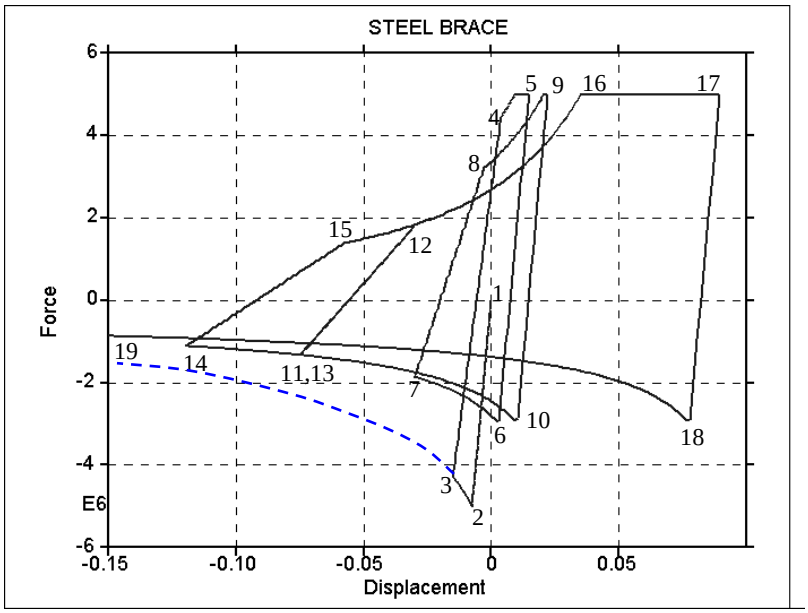


Figure 171.1

The initial buckling load (point 2) is:

$$F_{b\,initial} = FBUCK + FBUCK2 / L^2$$

where FBUCK, FBUCK2 are input parameters and L is the length of the beam element. If neither FBUCK nor FBUCK2 are defined, the default is that the initial buckling load is

$$SIGY * A\left(A = cross - sectional\ area\right).$$

The buckling curve (shown dashed) has the form:

$$F\left(d\right) = F_{b\,initial} / \sqrt{A\delta + B}$$

where δ is abs(strain/yield strain), and *A* and *B* are internally-calculated functions of slenderness ratio (λ) and loading history.

The member slenderness ratio λ is defined as $\frac{kL}{r}$, where *k* depends on end conditions, *L* is the element length, and *r* is the radius of gyration such that $Ar^2 = I$ (and $I = \min\left(I_{yy}, I_{zz}\right)$); λ will by default be calculated from the section properties and element length using *k*=1. Optionally, this may be overridden by input parameter LAMDA to allow for different end conditions.

Optionally, the user may provide a buckling curve (BCUR). The points of the curve give compressive displacement (x-axis) versus force (y-axis); the first point should have zero displacement and the initial buckling force. Displacement and force should both be positive. The initial buckling force must not be greater than the yield force.

The tensile yield force (point 5 and section 16-17) is defined by

$$F_y = SIGY * A,$$

where yield stress SIGY is an input parameter and A is the cross-sectional area.

Following initial buckling and subsequent yield in tension, the member is assumed to be damaged. The initial buckling curve is then scaled by input parameter CCBRF, leading to reduced strength curves such as segments 6-7, 10-14 and 18-19. This reduction factor is typically in the range 0.6 to 1.0 (smaller values for more slender members). By default, CCBRF is calculated using SEAOC 1990:

$$CCBRF = \frac{1}{\left(1 + \frac{0.5\lambda}{C_c}\right)} \quad \text{and} \quad C_c = \pi\sqrt{\frac{E}{0.5\sigma_y}}$$

When tensile loading is applied after buckling, the member must first be straightened before the full tensile yield force can be developed. This is represented by a reduced unloading stiffness (e.g. segment 14-15) and the tensile reloading curve (segments 8-9 and 15-16). Further details can be found in Bruneau, Uang, and Whittaker [1998] and Structural Engineers Association of California [1974, 1990, 1996].

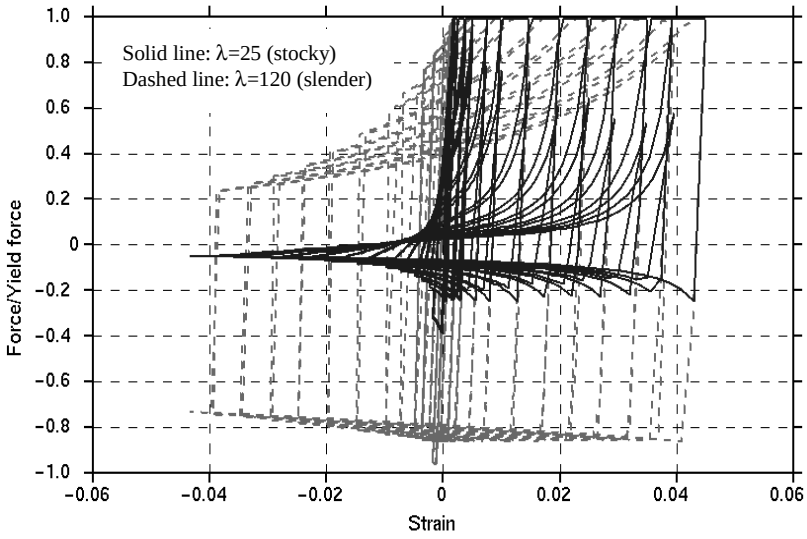


Figure 171.2

The response of stocky (low λ) and slender (high λ) braces are compared in the graph. These differences are achieved by altering the input value LAMDA (or the section properties of the beam) and FBUCK.

Output

Axial Strain and Internal Energy may be plotted from the INTEGRATED beam results menus in Oasys Ltd. Post processors: D3PLOT and T/HIS.

FEMA thresholds are the total axial strains (defined by change of length/initial length) at which the element is deemed to have passed from one category to the next, e.g. “Elastic”, “Immediate Occupancy”, “Life Safe”, etc. During the analysis, the maximum tensile and compressive strains (“high tide strains”) are recorded. These are checked against the user-defined limits TS1 to TS4 and CS1 to CS4. The output flag is then set to 0, 1, 2, 3, or 4 according to which limits have been passed. The value in the output files is the highest such flag from tensile or compressive strains. To plot this data, select INTEGRATED beam results, Integration point 4, Axial Strain.

Maximum plastic strains in tension and compression are also output. These are defined as maximum total strain to date minus the yield or first buckling strain for tensile and compressive plastic strains respectively. To plot these, select INTEGRATED beam results, Integration point 4, “shear stress XY” and “shear stress XZ” for tensile and compressive plastic strains, respectively.

*MAT_CONCRETE_EC2

This is Material Type 172, for shell and Hughes-Liu beam elements only. The material model can represent plain concrete only, reinforcing steel only, or a smeared combination of concrete and reinforcement. The model includes concrete cracking in tension and crushing in compression, and reinforcement yield, hardening and failure. Properties are thermally sensitive; the material model can be used for fire analysis. Material data and equations governing the behavior (including thermal properties) are taken from Eurocode 2 Part 1.2 (General rules – Structural fire design).

Note: This Material Type will be available starting in release 3 of version 971.

Card 1

1	2	3	4	5	6	7	8
Variable	MID	RO	FC	FT	TYPEC	UNITC		
Type	A8	F	F	F	F	F		
Default	none	none	none	0.0	1.0	1.0		

Card 2

1	2	3	4	5	6	7	8
Variable	ESOFT	LCHAR	MU					
Type	A8	F	F					
Default	See notes	0.0	0.4					

Card 3

1	2	3	4	5	6	7	8
Variable	YMREINF	PRREINF	SUREINF	TYPER	FRACRX	FRACY		
Type	F	F	F	F	F	F		
Default	none	0.0	0.0	1.0	0.0	0.0		

Card 4 - Leave blank

Card 4	1	2	3	4	5	6	7	8
Variable								
Type								
Default								

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
FC	Compressive strength of concrete (stress units)
FT	Tensile stress to cause cracking
TYPEC	Concrete aggregate type for stress-strain-temperature relationships EQ.1.0: Siliceous (default) EQ.2.0: Calcareous
UNITC	Factor to convert stress units to MPa (see Remarks)
ESOFT	Tension stiffening (Slope of stress-strain curve post-cracking in tension)
MU	Friction on crack planes (max shear = mu*compressive stress)
LCHAR	Characteristic length at which ESOFT applies
YMREINF	Young's Modulus of reinforcement
PRREINF	Poisson's Ratio of reinforcement
SUREINF	Ultimate stress of reinforcement
TYPER	Type of reinforcement for stress-strain-temperature relationships EQ.1.0: Hot rolled reinforcing steel EQ.2.0: Cold worked reinforcing steel (default) EQ.3.0: Quenched and tempered prestressing steel EQ.4.0: Cold worked prestressing steel

VARIABLE	DESCRIPTION
FRACRX	Fraction of reinforcement (x-axis) (e.g. for 1% reinforcement FRACR=0.01)
FRACRY	Fraction of reinforcement (y-axis) (e.g. for 1% reinforcement FRACR=0.01)

Remarks:

Reinforcement is treated as separate sets of bars in the local element x and y axes. The reinforcement is assumed not to carry through-thickness or in-plane shear.

Creating Reinforced Concrete Sections
This material model can be used to represent unreinforced concrete (FRACR=0), steel (FRACR=1), or reinforced concrete with evenly distributed reinforcement (0<FRACR<1).

Alternatively, use *INTEGRATION_SHELL to define the section. All parts referred to by one integration rule must have the same material type, but can have different material properties. Create one Part for concrete, and another for steel.

Material Behavior
Stress-strain curves for concrete and steel are as specified in EC2, scaled to the user-supplied FC, FT and SUREINF. Thermal expansion data is taken from EC2 directly, there is no option at present to override with user-supplied data.

Cracking in tension occurs when the maximum in-plane principal stress (bending + membrane stress at an integration point) reaches FT. Thereafter, increasing tensile strain leads to stress reduction according to the stress/strain slope ESOFT (default = 5 times initial elastic stiffness). Infinite ESOFT would in theory correspond to instantaneous loss of tensile capacity due to cracking, but values higher than the initial elastic modulus are not allowed. Finite values imply an amount of energy absorption needed to create a fully open crack, which arises in practice from the reinforcement holding the concrete together allowing it to continue to take some tension (this effect is frequently known as tension-stiffening). LCHAR can be used to maintain constant energy per unit area of crack irrespective of mesh size if desired – the value of ESOFT is scaled up for larger elements and down for smaller ones such that the stress-versus-displacement relation is the same. However, tension-stiffening effects are usually characterized by strain rather than displacement, this is obtained by LCHAR=0. If no data is available, it is recommended to set ESOFT such that FT/ESOFT = 0.0025, i.e. at a tensile strain of 0.0025 the crack is fully open, and LCHAR=0.

Output
“Plastic Strain” is the maximum of the plastic strains in the reinforcement in the two local directions.

Extra history variables may be requested for shell elements (NEIPS on *DATABASE_ EXTENT_BINARY), which have the following meaning:

Extra Variable 1: Current crack opening strain (if two cracks are present, max of the two)
Extra Variable 2: Equivalent uniaxial strain for concrete compressive behavior
Extra Variable 3: Number of cracks (0, 1 or 2)
Extra Variable 4: Temperature
Extra Variable 5: Thermal strain
Extra Variable 6: Current crack opening strain – first crack to form
Extra Variable 7: Current crack opening strain – crack at 90 degrees to first crack
Extra Variable 8: Max crack opening strain – first crack to form
Extra Variable 9: Max crack opening strain – crack at 90 degrees to first crack

MAXINT (shells) and/or BEAMIP (beams) on *DATABASE_EXTENT_BINARY should be set to the maximum number of integration points, so that results for all integration points can be plotted separately.

*MAT_MOHR_COULOMB

This is Material Type 173 for solid elements only, is intended to represent sandy soils and other granular materials. Joints (planes of weakness) may be added if required; the material then represents rock. The joint treatment is identical to that of *MAT_JOINTED_ROCK.

Note: This Material Type will be available starting in release 3 of version 971.

Card 1

1	2	3	4	5	6	7	8
Variable	MID	RO	GMOD	RNU	(blank)	PHI	CVAL	PSI
Type	A8	F	F	F		F	F	F
Default								0.0

Card 2

Variable	(blank)	NPLANES	(blank)	LCCPDR	LCCPT	LCCJDR	LCCJT	LCSFAC
Type		I		I	I	I	I	I
Default		0		0	0	0	0	0

Card 3

Variable	GMODDP	GMODGR	LCGMEP	LCPHIEP	LCPSIEP	LCGMST	CVALGR	ANISO
Type	F	F	F	F	F	F	F	F
Default	0.0	0.0	0.0	0.0	0.0	0.0	0.0	1.0

Card 4 – Repeat for each plane (maximum 3 planes)

Card 4	1	2	3	4	5	6	7	8
Variable	DIP	DIPANG	CPLANE	FRPLANE	TPLANE	SHRMAX	LOCAL	
Type	F	F	F	F	F	F	F	
Default	0.0	0.0	0.0	0.0	0.0	1.e20	0.0	

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
GMOD	Elastic shear modulus
RNU	Poisson's ratio
PHI	Angle of friction (radians)
CVAL	Cohesion value (shear strength at zero normal stress)
PSI	Dilation angle (radians)
NPLANES	Number of joint planes (maximum 3)
LCCPDR	Load curve for extra cohesion for parent material (dynamic relaxation)
LCCPT	Load curve for extra cohesion for parent material (transient)
LCCJDR	Load curve for extra cohesion for joints (dynamic relaxation)
LCCJT	Load curve for extra cohesion for joints (transient)
LCSFAC	Load curve giving factor on strength vs. time
GMODDP	Z-coordinate at which GMOD and CVAL are correct
GMODGR	Gradient of GMOD versus z-coordinate (usually negative)
LCGMEP	Load curve of GMOD versus plastic strain (overrides GMODGR)
LCPHIEP	Load curve of PHI versus plastic strain

VARIABLE	DESCRIPTION
LCPSIEP	Load curve of PSI versus plastic strain
LCGMST	(Leave blank)
CVALGR	Gradient of CVAL versus z-coordinate (usually negative)
ANISO	Factor applied to elastic shear stiffness in global XZ and YZ planes
DIP	Angle of the plane in degrees below the horizontal
DIPANG	Plan view angle (degrees) of downhill vector drawn on the plane
CPLANE	Cohesion for shear behavior on plane
PHPLANE	Friction angle for shear behavior on plane (degrees)
TPLANE	Tensile strength across plane (generally zero or very small)
SHRMAX	Max shear stress on plane (upper limit, independent of compression)
LOCAL	EQ.0: DIP and DIPANG are with respect to the global axes EQ.1: DIP and DIPANG are with respect to the local element axes

Remarks:

1. The material has a Mohr Coulomb yield surface, given by $\tau_{max} = C + \sigma_n \tan(phi)$, where τ_{max} =maximum shear stress on any plane, σ_n=normal stress on that plane (positive in compression), C=cohesion, phi=friction angle. The plastic potential function is of the form $\beta\sigma_k - \sigma_l$ + constant, where σ_k = maximum principal stress, σ_i = minimum principal stress, and $\beta = \frac{(1+\sin(psi))}{1-\sin(psi)}$.

2. The tensile strength of the material is given by $\sigma_{max} = \frac{C}{\tan(phi)}$ where C is the cohesion.

   After the material reaches its tensile strength, further tensile straining leads to volumetric voiding; the voiding is reversible if the strain is reversed.

3. If depth-dependent properties are used, the model must be oriented with the z-axis in the upward direction.

4. Plastic strain is defined as $SQRT(2/3.\varepsilon_{pij}.\varepsilon_{pij})$, i.e. the same way as for other elasto-plastic material models.

5. Friction and dilation angles PHI and PSI may vary with plastic strain, to model heavily consolidated materials under large shear strains – as the strain increases, the dilation angle typically reduces to zero and the friction angle to a lower, pre-consolidation value.

6. For similar reasons, the shear modulus may reduce with plastic strain, but this option may sometimes give unstable results.

7. The loadcurves LCCPDR, LCCPT, LCCJDR, LCCJT allow extra cohesion to be specified as a function of time. The cohesion is additional to that specified in the material parameters. This is intended for use during the initial stages of an analysis to allow application of gravity or other loads without cracking or yielding, and for the cracking or yielding then to be introduced in a controlled manner. This is done by specifying extra cohesion that exceeds the expected stresses initially, then declining to zero. If no curves are specified, no extra cohesion is applied.

8. The loadcurve for factor on strength applies simultaneously to the cohesion and tan(friction angle) of parent material and all joints. This feature is intended for reducing the strength of the material gradually, to explore factors of safety. If no curve is present, a constant factor of 1 is assumed. Values much greater than 1.0 may cause problems with stability.

9. The anisotropic factor ANISO applies the elastic shear stiffness in the global XZ and YZ planes. It can be used only in pure Mohr-Coulomb mode (NPLANES=0).

10. To model soil, set NJOINT=0. The joints are to allow modeling of rock, and are treated identically to those of *MAT_JOINTED_ROCK.

11. The joint plane orientations are defined by the angle of a “downhill vector” drawn on the plane, i.e. the vector is oriented within the plane to obtain the maximum possible downhill angle. DIP is the angle of this line below the horizontal. DIPANG is the plan-view angle of the line (pointing down hill) measured clockwise from the global Y-axis about the global Z-axis.

12. Joint planes would generally be defined in the global axis system if they are taken from survey data. However, the material model can also be used to represent masonry, in which case the weak planes represent the cement and lie parallel to the local element axes.

13. The joint planes rotate with the rigid body motion of the elements, irrespective of whether their initial definitions are in the global or local axis system.

14. Extra variables for plotting. By setting NEIPH on *DATABASE_EXTENT_BINARY to 15, the following variables can be plotted in D3PLOT and T/HIS:

    Extra Variable 1: Mobilized strength fraction for base material
    Extra Variable 2: Volumetric void strain
    Extra Variable 3: Maximum stress overshoot during plastic calculation
    Extra Variable 4: crack opening strain for plane 1
    Extra Variable 5: crack opening strain for plane 2
    Extra Variable 6: crack opening strain for plane 3
    Extra Variable 7: crack accumulated shear strain for plane 1
    Extra Variable 8: crack accumulated shear strain for plane 2
    Extra Variable 9: crack accumulated shear strain for plane 3

Extra Variable 10: current shear utilization for plane 1
Extra Variable 11: current shear utilization for plane 2
Extra Variable 12: current shear utilization for plane 3
Extra Variable 13: maximum shear utilization to date for plane 1
Extra Variable 14: maximum shear utilization to date for plane 2
Extra Variable 15: maximum shear utilization to date for plane 3

***MAT_RC_BEAM**

This is Material Type 174, for Hughes-Liu beam elements only. The material model can represent plain concrete only, reinforcing steel only, or a smeared combination of concrete and reinforcement. The main emphasis of this material model is the cyclic behavior – it is intended primarily for seismic analysis.

Note: This Material Type will be available starting in release 3 of version 971.

Card 1

Variable	MID	RO	EUNL	PR	FC	EC1	EC50	RESID
Type	A8	F	F	F	F	F	F	F
Default	none	none	See Remarks	0.0	none	0.0022	See Remarks	0.2

Card 2

Variable	FT	UNITC	(blank)	(blank)	(blank)	ESOFT	LCHAR	OUTPUT
Type	F	F	F	F	F	F	F	F
Default	See Remarks	1.0	none	none	none	See Remarks	none	0

Card 3

Variable	FRACR	YMREINF	PRREINF	SYREINF	SUREINF	ESHR	EUR	RREINF
Type	F	F	F	F	F	F	F	F
Default	0.0	none	0.0	0.0	SYREINF	0.03	0.2	4.0

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
EUNL	Initial unloading elastic modulus (See Remarks).
PR	Poisson's ratio.
FC	Cylinder strength (stress units)
EC1	Strain at which stress FC is reached.
EC50	Strain at which the stress has dropped to 50% FC
RESID	Residual strength factor
FT	Maximum tensile stress
UNITC	Factor to convert stress units to MPa (See Remarks)
ESOFT	Slope of stress-strain curve post-cracking in tension
LCHAR	Characteristic length for strain-softening behavior
OUTPUT	Output flag controlling what is written as "plastic strain" EQ.0.0: Curvature EQ.1.0: "High-tide" plastic strain in reinforcement
FRACR	Fraction of reinforcement (e.g. for 1% reinforcement FRACR=0.01)
YMREINF	Young's Modulus of reinforcement
PRREINF	Poisson's Ratio of reinforcement
SYREINF	Yield stress of reinforcement
SUREINF	Ultimate stress of reinforcement
ESHR	Strain at which reinforcement begins to harden
EUR	Strain at which reinforcement reaches ultimate stress
R_REINF	Dimensionless Ramberg-Osgood parameter r. If zero, a default value r=4.0 will be used. If set to -1, parameters will be calculated from Kent & Park formulae. (See Remarks)

Remarks:

Creating sections for reinforced concrete beams
This material model can be used to represent unreinforced concrete (FRACR=0), steel (FRACR=1), or reinforced concrete with evenly distributed reinforcement (0<FRACR<1).

Alternatively, use *INTEGRATION_BEAM to define the section. A new option in allows the user to define a Part ID for each integration point, similar to the facility already available with *INTEGRATION_SHELL. All parts referred to by one integration rule must have the same material type, but can have different material properties. Create one Part for concrete, and another for steel. These Parts should reference Materials, both of type *MAT_RC_BEAM, one with FRACR=0, the other with FRACR=1. Then, by assigning one or other of these Part Ids to each integration point the reinforcement can be applied to the correct locations within the section of the beam.

Concrete
In monotonic compression, the approach of Park and Kent, as described in Park & Paulay [1975] is used. The material follows a parabolic stress-strain curve up to a maximum stress equal to the cylinder strength FC; therafter the strength decays linearly with strain until the residual strength is reached. Default values for some material parameters will be calculated automatically as follows:

$$EC50 = \frac{(3+0.29FC)}{145FC-1000}$$ (FC in MPa units – Park and Kent, from test data)

EUNL = initial tangent slope = 2FC/EC1 (User-defined values lower than this are not permitted, but higher values may be defined if desired)

$$FT = 1.4\left(\frac{FC}{10}\right)^{\frac{2}{3}}$$ (FC in MPa units – from CEB Code 1993)

ESOFT = EUNL (User-defined values higher than EUNL are not permitted)

UNITC is used only to calculate default values for the above parameters from FC.

Strain-softening behavior tends to lead to deformations being concentrated in one element, and hence the overall force-deflection behavior of the structure can be mesh-size-dependent if the softening is characterized by strain. To avoid this, a characteristic length (LCHAR) may be defined. This is the length of specimen (or element) that would exhibit the defined monotonic stress-strain relationship. LS-DYNA adjusts the stress-strain relationship after ultimate load for each element, such that all elements irrespective of their length will show the same deflection during strain softening (i.e. between ultimate load and residual load). Therefore, although deformation will still be concentrated in one element, the load-deflection behavior should be the same irrespective of element size. For tensile behavior, ESOFT is similarly scaled.

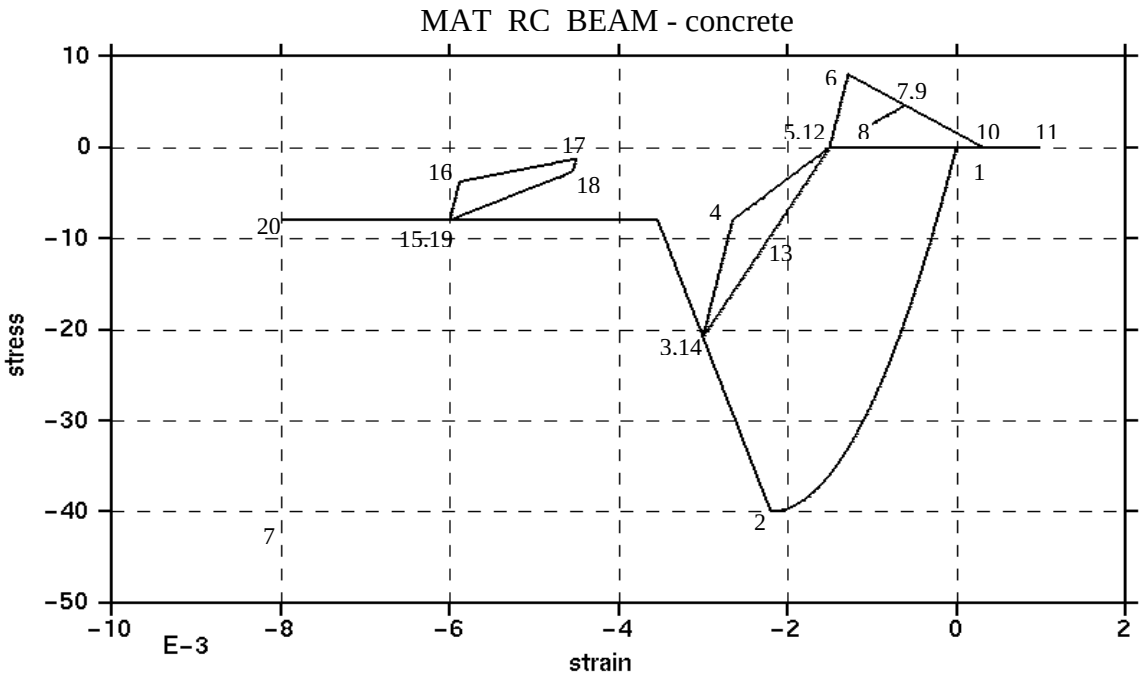


Figure 174.1

Cyclic behavior is broadly suggested by Blakeley and Park [1973] as described in Park & Paulay [1975]; the stress-strain response lies within the Park-Kent envelope, and is characterized by stiff initial unloading response at slope EUNL followed by a less stiff response if it unloads to less than half the current strength. Reloading stiffness degrades with increasing strain.

In tension, the stress rises linearly with strain until a tensile limit FT is reached. Thereafter the stiffness and strength decays with increasing strain at a rate ESOFT. The stiffness also decays such that unloading always returns to strain at which the stress most recently changed to tensile.

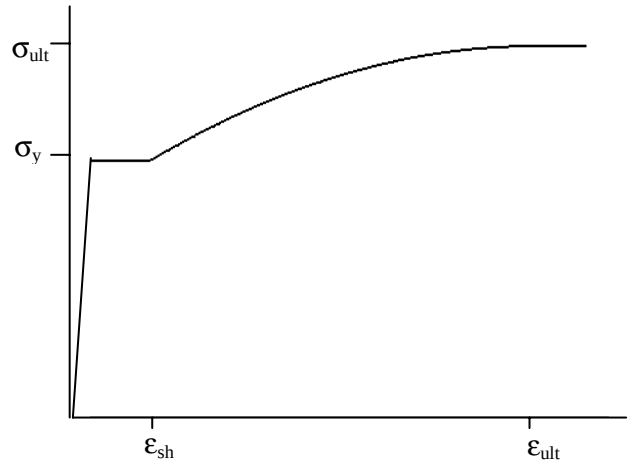


Figure 174.2

Monotonic loading of the reinforcement results in the stress-strain curve shown, which is parabolic between ε_{sh} and ε_{ult}. The same curve acts as an envelope on the hysteretic behavior, when the x-axis is cumulative plastic strain.

Unloading from the yielded condition is elastic until the load reverses. Thereafter, the Bauschinger Effect (reduction in stiffness at stresses less than yield during cyclic deformation) is represented by following a Ramberg-Osgood relationship until the yield stress is reached:

$$\varepsilon - \varepsilon_s = \left(\frac{\sigma}{E}\right)\left\{1+\left(\frac{\sigma}{\sigma_{CH}}\right)^{r-1}\right\}$$

where ε and σ are strain and stress, ε_s is the strain at zero stress, E is Young's Modulus, and r and σ_{CH} are as defined below

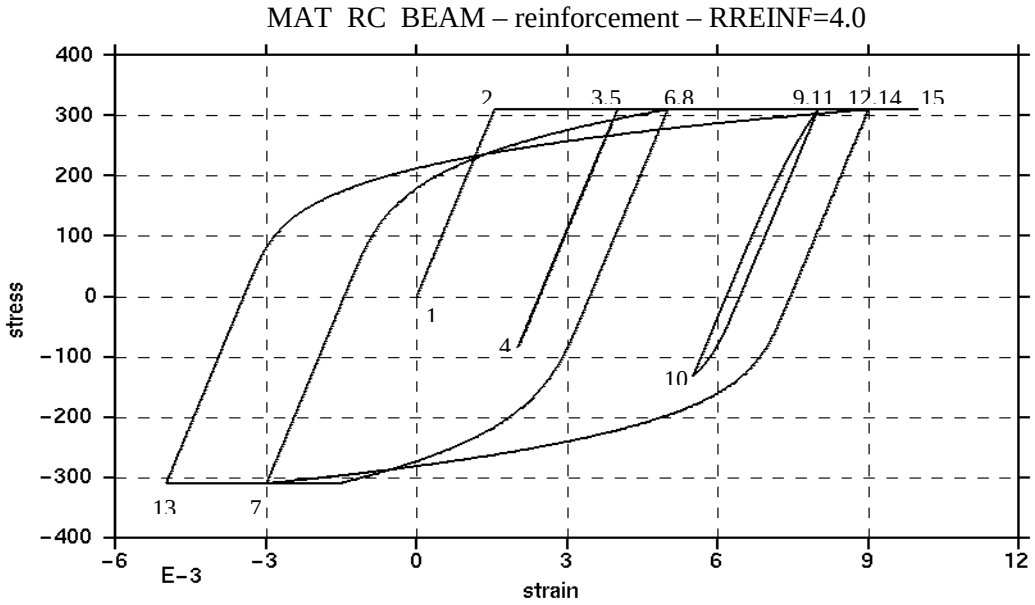


Figure 174.3

Two options are given for calculation r and σ_{CH}, which is performed at each stress reversal:

1. If RREINF is input as -1, r and σ_{CH} are calculated internally from formulae given in Kent and Park. Parameter r depends on the number of stress reversals. Parameter σ_{CH} depends on the plastic strain that occurred between the previous two stress reversals. The formulae were statistically derived from experiments, but may not fit all circumstances. In particular, large differences in behavior may be caused by the presence or absence of small stress reversals such as could be caused by high frequency oscillations. Therefore, results might sometimes be unduly sensitive to small changes in the input data.

2. If RREINF is entered by the user or left blank, r is held constant while σ_{CH} is calculated on each reversal such that the Ramberg-Osgood curve meets the monotonic stress-strain curve at the point from which it last unloaded, e.g. points 6 and 8 are coincident in the graph below. The default setting RREINF=4.0 gives similar hysteresis behavior to that described by Kent & Park but is unlikely to be so sensitive to small changes of input data.

Output

It is recommended to use BEAMIP on *DATABASE_EXTENT_BINARY to request stress and strain output at the individual integration points. If this is done, for MAT_RC_BEAM only, element curvature is written to the output files in place of plastic strain. In the post-processor, select “plastic strain” to display curvature (whichever of the curvatures about local y and z axes has greatest absolute value will be plotted). Alternatively, select “axial strain” to display the total axial strain (elastic + plastic) at that integration point; this can be combined with axial stress to create hysteresis plots such as those shown above.

***MAT_VISCOELASTIC_THERMAL**

This is Material Type 175. This material model provides a general viscoelastic Maxwell model having up to 12 terms in the prony series expansion and is useful for modeling dense continuum rubbers and solid explosives. Either the coefficients of the prony series expansion or a relaxation curve may be specified to define the viscoelastic deviatoric and bulk behavior.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	BULK	PCF	EF	TREF	A	B
Type	A8	F	F	F	F	F	F	F

Insert a blank card here if constants are defined on cards 3,4,... below.

If an elastic layer is defined in a laminated shell this card must be blank.

Card 2	1	2	3	4	5	6	7	8
Variable	LCID	NT	BSTART	TRAMP	LCIDK	NTK	BSTARTK	TRAMPK
Type	F	I	F	F	F	I	F	F

Card Format for viscoelastic constants. Up to 12 cards may be input. A keyword card (with a "*" in column 1) terminates this input if less than 12 cards are used. These cards are not needed if relaxation data is defined. The number of terms for the shear behavior may differ from that for the bulk behavior: simply insert zero if a term is not included.

If an elastic layer is defined you only need to define GI and KI (note in an elastic layer only one card is needed)

Optional Cards	1	2	3	4	5	6	7	8
Variable	GI	BETAI	KI	BETAKI				
Type	F	F	F	F				

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
BULK	Elastic bulk modulus.
PCF	Tensile pressure elimination flag for solid elements only. If set to unity tensile pressures are set to zero.
EF	Elastic flag (if equal 1, the layer is elastic. If 0 the layer is viscoelastic).
TREF	Reference temperature for shift function (must be greater than zero).
A	Coefficient for the Arrhenius and the Williams-Landau-Ferry shift functions.
B	Coefficient for the Williams-Landau-Ferry shift function.
LCID	Load curve ID for deviatoric behavior if constants, G_i, and β_i are determined via a least squares fit. This relaxation curve is shown below.
NT	Number of terms in shear fit. If zero the default is 6. Fewer than NT terms will be used if the fit produces one or more negative shear moduli. Currently, the maximum number is set to 6.
BSTART	In the fit, β_1 is set to zero, β_2 is set to BSTART, β_3 is 10 times β_2, β_4 is 100 times greater than β_3 , and so on. If zero, BSTART is determined by an iterative trial and error scheme.
TRAMP	Optional ramp time for loading.
LCIDK	Load curve ID for bulk behavior if constants, K_i, and $\beta\kappa_i$ are determined via a least squares fit. This relaxation curve is shown below.
NTK	Number of terms desired in bulk fit. If zero the default is 6. Currently, the maximum number is set to 6.
BSTARTK	In the fit, $\beta\kappa_1$ is set to zero, $\beta\kappa_2$ is set to BSTARTK, $\beta\kappa_3$ is 10 times $\beta\kappa_2$, $\beta\kappa_4$ is 100 times greater than $\beta\kappa_3$, and so on. If zero, BSTARTK is determined by an iterative trial and error scheme.
TRAMPK	Optional ramp time for bulk loading.
GI	Optional shear relaxation modulus for the ith term

VARIABLE	DESCRIPTION
BETAI	Optional shear decay constant for the ith term
KI	Optional bulk relaxation modulus for the ith term
BETAKI	Optional bulk decay constant for the ith term

Remarks:

Rate effects are taken into accounted through linear viscoelasticity by a convolution integral of the form:

$$\sigma_{ij} = \int_0^t g_{ijkl}\left(t-\tau\right)\frac{\partial \varepsilon_{kl}}{\partial \tau}d\tau$$

where $g_{ijkl(t-\tau)}$ is the relaxation functions for the different stress measures. This stress is added to the stress tensor determined from the strain energy functional.

If we wish to include only simple rate effects, the relaxation function is represented by six terms from the Prony series:

$$g\left(t\right) = \sum_{m=1}^{N} G_m e^{-\beta_m t}$$

We characterize this in the input by shear moduli, G_i, and decay constants, β_i. An arbitrary number of terms, up to 6, may be used when applying the viscoelastic model.

For volumetric relaxation, the relaxation function is also represented by the Prony series in terms of bulk moduli:

$$k\left(t\right) = \sum_{m=1}^{N} K_m e^{-\beta_{k_m} t}$$

The Arrhenius and Williams-Landau-Ferry (WLF) shift functions account for the effects of the temperature on the stress relaxation. A scaled time, *t'*,

$$t' = \int_0^t \Phi(T)dt$$

is used in the relaxation function instead of the physical time. The Arrhenius shift function is

$$\Phi(T) = \exp(-A\{\frac{1}{T} - \frac{1}{T_{REF}}\})$$

and the Williams-Landau-Ferry shift function is

$$\Phi(T) = \exp(-A\frac{T - T_{REF}}{B + T - T_{REF}})$$

If all three values (TREF, A, and B) are not zero, the WLF function is used; the Arrhenius function is used if B is zero; and no scaling is applied if all three values are zero.

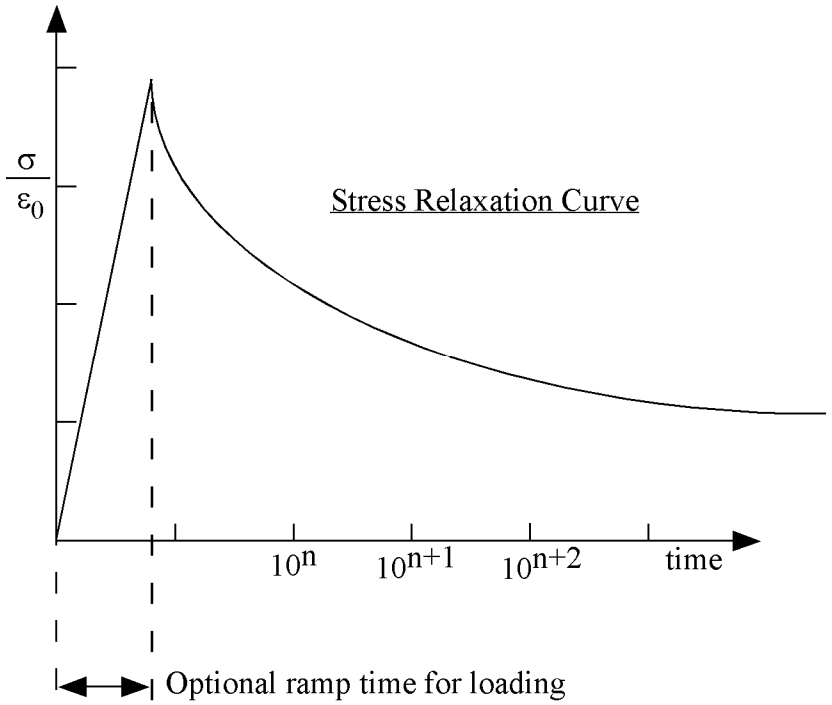


Figure 175.1. Relaxation curve. This curve defines stress versus time where time is defined on a logarithmic scale. For best results, the points defined in the load curve should be equally spaced on the logarithmic scale. Furthermore, the load curve should be smooth and defined in the positive quadrant. If nonphysical values are determined by least squares fit, LS-DYNA will terminate with an error message after the initialization phase is completed. If the ramp time for loading is included, then the relaxation which occurs during the loading phase is taken into account. This effect may or may not be important.

*MAT_QUASILINEAR_VISCOELASTIC

Purpose: This is Material Type 176. This is a quasi-linear, isotropic, viscoelastic material based on a one-dimensional model by Fung [1993], which represents biological soft tissues such as brain, skin, kidney, spleen, etc. This model is implemented for solid and shell elements. The formulation has recently been changed to allow larger strains, and, in general, will not give the same results as the previous implementation which remains the default.

Card 1

Variable	MID	RO	K	LC1	LC2	N	GSTART	M
Type	A8	F	F	I	I	F	F	F
Default	none	none	none	0	0	6	1/TMAX	6

Card 2

Variable	SO	E_MIN	E_MAX	GAMA1	GAMA2	K	EH	FORM
Type	F	F	F	F	F	F	F	I
Default	0.0	-0.9	5.1	0.0	0.0	0.0	0.0	0

Define the following 3 cards if and only if LC1 is 0.

Card 3

Variable	G1	BETA1	G2	BETA2	G3	BETA3	G4	BETA4
Type	F	F	F	F	F	F	F	F

Card 4	1	2	3	4	5	6	7	8
Variable	G5	BETA5	G6	BETA6	G7	BETA7	G8	BETA8
Type	F	F	F	F	F	F	F	F

Card 5

Variable	G9	BETA9	G10	BETA10	G11	BETA11	G12	BETA12
Type	F	F	F	F	F	F	F	F

Define the following card if and only if LC2 is 0.

Card 3 or 6	1	2	3	4	5	6	7	8
Variable	C1	C2	C3	C4	C5	C6		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
K	Bulk modulus.
LC1	Load curve ID that defines the relaxation function in shear. This curve is used to fit the coefficients *Gi* and *BETAi*. If zero, define the coefficients directly. The latter is recommended.
LC2	Load curve ID that defines the instantaneous elastic response in compression and tension. If zero, define the coefficients directly. *Symmetry is not assumed if only the tension side is define; therefore, defining the response in tension only, may lead to nonphysical behavior in compression. Also, this curve should give a softening response for increasing strain without any negative or zero slopes. A stiffening curve or one with negative slopes is generally unstable.*

VARIABLE	DESCRIPTION
N	Number of terms used in the Prony series, a number less than or equal to 6. This number should be equal to the number of decades of time covered by the experimental data. Define this number if LC1 is nonzero. Carefully check the fit in the D3HSP file to ensure that it is valid, since the least square fit is not always reliable.
GSTART	Starting value for least square fit. If zero, a default value is set equal to the inverse of the largest time in the experiment. Define this number if LC1 is nonzero.
M	Number of terms used to determine the instantaneous elastic response. This variable is ignored with the new formulation but is kept for compatibility with the previous input.
SO	Strain (logarithmic) output option to control what is written as component 7 to the d3plot database. (LS-Prepost always blindly labels this component as effective plastic strain.) The maximum values are updated for each element each time step: EQ.0.0: maximum principal strain that occurs during the calculation, EQ.1.0: maximum magnitude of the principal strain values that occurs during the calculation, EQ.2.0: maximum effective strain that occurs during the calculation.
E_MIN	Minimum strain used to generate the load curve from Ci. The default range is -0.9 to 5.1. The computed solution will be more accurate if the user specifies the range used to fit the Ci. Linear extrapolation is used outside the specified range.
E_MAX	Maximum strain used to generate the load curve from Ci.
K	Material failure parameter that controls the volume enclosed by the failure surface, see *MAT_SIMPLIFIED_RUBBER. LE.0.0: ignore failure criterion; GT.0.0: use actual K value for failure criterions.
GAMA1	Material failure parameter, see *MAT_SIMPLIFIED_RUBBER and Figure 181.1.
GAMA2	Material failure parameter, see *MAT_SIMPLIFIED_RUBBER.
EH	Damage parameter, see *MAT_SIMPLIFIED_RUBBER.

VARIABLE	DESCRIPTION
FORM	Formulation of model. FORM=0 gives the original model developed by Fung, which always relaxes to a zero stress state as time approaches infinity, and FORM=1 gives the alternative model, which relaxes to the quasi-static elastic response. In general, the two formulations won't give the same responses. Formulation, FORM=-1, is an improvement on FORM=0 where the instantaneous elastic response is used in the viscoelastic stress update, not just in the relaxation, as in FORM=0. Consequently, the constants for the elastic response do not need to be scaled.
Gi	Coefficients of the relaxation function. The number of coefficients is currently limited to 6 although 12 may be read in to maintain compatibility with the previous formulation's input. Define these coefficients if LC1 is set to zero. At least 2 coefficients must be nonzero.
BETAi	Decay constants of the relaxation function. Define these coefficients if LC1 is set to zero. The number of coefficients is currently limited to 6 although 12 may be read in to maintain compatibility with the previous formulation's input.
Ci	Coefficients of the instantaneous elastic response in compression and tension. Define these coefficients only if LC2 is set to zero.

Remarks:

The equations for the original model (FORM=0) are given as:

$$\sigma_V(t) = \int_0^t G(t-\tau)\frac{\partial \sigma_\varepsilon\left[\varepsilon(\tau)\right]}{\partial \varepsilon}\frac{\partial \varepsilon}{\partial \tau}d\tau$$

$$G(t) = \sum_{i=1}^{n} G_i e^{-\beta t}$$

$$\sigma_\varepsilon(\varepsilon) = \sum_{i=1}^{k} C_i \varepsilon^i$$

where G is the shear modulus. Effective plastic strain (which can be written to the d3plot database) is calculated as follows:

$$\varepsilon^{effectuve} = \sqrt{\frac{2}{3}\varepsilon_{ij}\varepsilon_{ij}}$$

The polynomial for instantaneous elastic response should contain only odd terms if symmetric tension-compression response is desired.

The new model (FORM=1) is based on the hyperelastic model used *MAT_SIMPLIFIED_RUBBER assuming incompressibility. The one-dimensional expression for σ_ε generates the uniaxial stress-strain curve and an additional visco-elastic term is added on,

$$\sigma(\varepsilon,t)=\sigma_{SR}(\varepsilon)+\sigma_V(t)$$

$$\sigma_V(t)=\int_0^t G(t-\tau)\frac{\partial\varepsilon}{\partial\tau}d\tau$$

where the first term to the right of the equals sign is the hyperelastic stress and the second is the viscoelastic stress. Unlike the previous formulation, where the stress always relaxed to zero, the current formulation relaxes to the hyperelastic stress.

*MAT_HILL_FOAM

Purpose: This is Material Type 177. This is a highly compressible foam based on the strain-energy function proposed by Hill [1978]; also see Storakers [1986]. Poisson's ratio effects are taken into account.

Card 1

Variable	MID	RO	K	N	MU	LCID	FITTYPE	LCSR
Type	A8	F	F	F	F	I	I	I
Default	none	none	none	0	0	0	0	0

Define the following 2 cards if and only if LCID is 0.

Card 2

Variable	C1	C2	C3	C4	C5	C6	C7	C8
Type	F	F	F	F	F	F	F	F

Card 3

Variable	B1	B2	B3	B4	B5	B6	B7	B8
Type	F	F	F	F	F	F	F	F

Card 4

Variable	R	M						
Type	F	F						

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
K	Bulk modulus. This modulus is used for determining the contact interface stiffness.
N	Material constant. Define if LCID=0 below; otherwise, N is fit from the load curve data. See equations below.
MU	Damping coefficient.
LCID	Load curve ID that defines the force per unit area versus the stretch ratio. This curve can be given for either uniaxial or biaxial data depending on FITTYPE.
FITTYPE	Type of fit: EQ.1: uniaxial data, EQ.2: biaxial data, EQ.3: pure shear data.
LCSR	Load curve ID that defines the uniaxial or biaxial stretch ratio (see FITTYPE) versus the transverse stretch ratio.
Ci	Material constants. See equations below. Define up to 8 coefficients if LCID=0.
Bi	Material constants. See equations below. Define up to 8 coefficients if LCID=0.
R	Mullins effect model *r* coefficient
M	Mullins effect model *m* coefficient

Remarks:

If load curve data is defined, the fit generated by LS-DYNA must be closely checked in the D3HSP output file. It may occur that the nonlinear least squares procedure in LS-DYNA, which is used to fit the data, is inadequate.

The Hill strain energy density function for this highly compressible foam is given by:

$$W = \sum_{j=1}^{m} \frac{C_j}{b_j} \left[\lambda_1^{b_j} + \lambda_2^{b_j} + \lambda_3^{b_j} - 3 + \frac{1}{n} \left(J^{-nb_j} - 1 \right) \right]$$

where C_j, b_j, and n are material constants and $J = \lambda_1\lambda_2\lambda_3$ represents the ratio of the deformed to the undeformed state. The constant m is internally set to 4. In case number of points in the curve is less than 8, then m is set to the number of points divided by 2. The principal Cauchy stresses are

$$t_i = \sum_{j=1}^{m} \frac{C_j}{J}\left[\lambda_i^{b_j} - J^{-nb_j}\right] \quad i = 1,2,3$$

From the above equations the shear modulus is:

$$\mu = \frac{1}{2}\sum_{j=1}^{m} C_j b_j$$

and the bulk modulus is:

$$K = 2\mu\left(n + \frac{1}{3}\right)$$

The value for K defined in the input is used in the calculation of contact forces and for the material time step. Generally, this value should be equal to or greater that the K given in the above equation.

*MAT_VISCOELASTIC_HILL_FOAM

Purpose: This is Material Type 178. This is a highly compressible foam based on the strain-energy function proposed by Hill [1978]; also see Storakers [1986]. The extension to include large strain viscoelasticity is due to Feng and Hallquist [2002].

Card 1

Variable	MID	RO	K	N	MU	LCID	FITTYPE	LCSR
Type	A8	F	F	F	F	I	I	I
Default	none	none	none	0	0	0	0	0

Card 2

Variable	LCVE	NT	GSTART					
Type	I	F	F					
Default	0	6	1/TMAX					

Define the following 2 cards if and only if LCID is 0.

Card 3

Variable	C1	C2	C3	C4	C5	C6	C7	C8
Type	F	F	F	F	F	F	F	F

Card 4

Variable	B1	B2	B3	B4	B5	B6	B7	B8
Type	F	F	F	F	F	F	F	F

Card Format for Viscoelastic Constants. Up to 12 cards may be input. A keyword card (with a "*" in column 1) terminates this input if less than 12 cards are used.

Optional Cards	1	2	3	4	5	6	7	8
Variable	GI	BETAI						
Type	F	F						

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
K	Bulk modulus. This modulus is used for determining the contact interface stiffness.
N	Material constant. Define if LCID=0 below; otherwise, N is fit from the load curve data. See equations below.
MU	Damping coefficient.
LCID	Load curve ID that defines the force per unit area versus the stretch ratio. This curve can be given for either uniaxial or biaxial data depending on FITTYPE. Load curve LCSR below must also be defined.
FITTYPE	Type of fit: EQ.1: uniaxial data, EQ.2: biaxial data.
LCSR	Load curve ID that defines the uniaxial or biaxial stress ratio (see FITTYPE) versus the transverse stretch ratio.
LCVE	Optional load curve ID that defines the relaxation function in shear. This curve is used to fit the coefficients *Gi* and *BETAi*. If zero, define the coefficients directly. The latter is recommended.
NT	Number of terms used to fit the Prony series, which is a number less than or equal to 12. This number should be equal to the number of decades of time covered by the experimental data. Define this number if LCVE is nonzero. Carefully check the fit in the D3HSP file to ensure that it is valid, since the least square fit is not always reliable.

VARIABLE	DESCRIPTION
GSTART	Starting value for least square fit. If zero, a default value is set equal to the inverse of the largest time in the experiment. Define this number if LC1 is nonzero, Ci, Material constants. See equations below. Define up to 8 coefficients.
Ci	Material constants. See equations below. Define up to 8 coefficients if LCID=0.
Bi	Material constants. See equations below. Define up to 8 coefficients if LCID=0.
GI	Optional shear relaxation modulus for the ith term
BETAI	Optional decay constant if ith term

Remarks:

If load curve data is defined, the fit generated by LS-DYNA must be closely checked in the D3HSP output file. It may occur that the nonlinear least squares procedure in LS-DYNA, which is used to fit the data, is inadequate.

The Hill strain energy density function for this highly compressible foam is given by:

$$p^{n+1} = p^n e^{-\beta\cdot\Delta t} + K\dot{\varepsilon}_{kk}\left(\frac{1-e^{-\beta\cdot\Delta t}}{\beta}\right) \qquad where\ \beta = |BETA|$$

where C_j, b_j, and n are material constants and $J = \lambda_1\lambda_2\lambda_3$ represents the ratio of the deformed to the undeformed state. The principal Cauchy stresses are

$$t_i = \sum_{j=1}^{m}\frac{C_j}{J}\left[\lambda_i^{b_j} - J^{-nb_j}\right]\ i = 1,2,3$$

From the above equations the shear modulus is:

$$\mu = \frac{1}{2}\sum_{j=1}^{m} C_j b_j$$

and the bulk modulus is:

$$K = 2\mu\left(n + \frac{1}{3}\right)$$

The value for K defined in the input is used in the calculation of contact forces and for the material time step. Generally, this value should be equal to or greater that the K given in the above equation.

Rate effects are taken into account through linear viscoelasticity by a convolution integral of the form:

$$\sigma_{ij} = \int_0^t g_{ijkl}\left(t-\tau\right)\frac{\partial \varepsilon_{kl}}{\partial \tau}d\tau$$

or in terms of the second Piola-Kirchhoff stress, S_{ij}, and Green's strain tensor, E_{ij},

$$S_{ij} = \int_0^t G_{ijkl}\left(t-\tau\right)\frac{\partial E_{kl}}{\partial \tau}d\tau$$

where $g_{ijkl}(t-\tau)$ and $G_{ijkl}(t-\tau)$ are the relaxation functions for the different stress measures. This stress is added to the stress tensor determined from the strain energy functional.

If we wish to include only simple rate effects, the relaxation function is represented by six terms from the Prony series:

$$g(t) = \alpha_0 + \sum_{m=1}^{N}\alpha_m e^{-\beta t}$$

given by,

$$g(t) = \sum_{i=1}^{n} G_i e^{-\beta_i t}$$

This model is effectively a Maxwell fluid which consists of a dampers and springs in series. We characterize this in the input by shear moduli, G_i, and decay constants, β_i. The viscoelastic behavior is optional and an arbitrary number of terms may be used.

***MAT_LOW_DENSITY_SYNTHETIC_FOAM_**{*OPTION*}

This is Material Type 179 (and 180 if the ORTHO option below is active) for modeling rate independent low density foams, which have the property that the hysteresis in the loading-unloading curve is considerably reduced after the first loading cycle. In this material we assume that the loading-unloading curve is identical after the first cycle of loading is completed and that the damage is isotropic, i.e., the behavior after the first cycle of loading in the orthogonal directions also follows the second curve. The main application at this time is to model the observed behavior in the compressible synthetic foams that are used in some bumper designs. Tables may be used in place of load curves to account for strain rate effects.

Available options include:

<BLANK>

ORTHO

WITH_FAILURE

ORTHO_WITH_FAILURE

If the foam develops orthotropic behavior, i.e., after the first loading and unloading cycle the material in the orthogonal directions are unaffected then the ORTHO option should be used. If the ORTHO option is active the directionality of the loading is stored. This option is requires additional storage to store the history variables related to the orthogonality and is slightly more expensive.

An optional failure criterion is included. A description of the failure model is provided below for material type 181, *MAT_SIMPLIFIED_RUBBER/FOAM.

Card 1

Variable	MID	RO	E	LCID1	LCID2	HU	BETA	DAMP
Type	A8	F	F	F	F	F	F	F
Default						1.		0.05
Remarks						3	1	

Card 2	1	2	3	4	5	6	7	8
Variable	SHAPE	FAIL	BVFLAG	ED	BETA1	KCON	REF	TC
Type	F	F	F	F	F	F	F	F
Default	1.0	0.0	0.0	0.0	0.0	0.0	0.0	1.E+20
Remarks	3		2	5	5	6		

Define the following optional card if and only if LCID1 is negative

Card 3	1	2	3	4	5	6	7	8
Variable	RFLAG	DTRT						
Type	F	F						
Default	0.0	0.0						
Remarks								

Define card 3 if and only if the option, WITH_FAILURE, is active.

Card 3	1	2	3	4	5	6	7	8
Variable	K	GAMA1	GAMA2	EH				
Type	F	F	F	F				

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density

VARIABLE	DESCRIPTION
E	Young's modulus. This modulus is used if the elongations are tensile as described for the *MAT_LOW_DENSITY_FOAM.
LCID1	Load curve or table ID: GT.0: Load curve ID, see *DEFINE_CURVE, for nominal stress versus strain for the undamaged material. LT.0: -LCID1 is Table ID, see *DEFINE_TABLE, for nominal stress versus strain for the undamaged material as a function of strain rate
LCID2	Load curve or table ID. The load curve ID, see *DEFINE_CURVE, defines the nominal stress versus strain for the damaged material. The table ID, see *DEFINE_TABLE, defines the nominal stress versus strain for the damaged material as a function of strain rate
HU	Hysteretic unloading factor between 0 and 1 (default=1, i.e., no energy dissipation), see also Figure 57.1.
BETA	β, decay constant to model creep in unloading
DAMP	Viscous coefficient (.05< recommended value <.50) to model damping effects. LT.0.0: |DAMP| is the load curve ID, which defines the damping constant as a function of the maximum strain in compression defined as: $\varepsilon_{\max} = \max(1-\lambda_1, 1-\lambda_2, 1.-\lambda_3)$. In tension, the damping constant is set to the value corresponding to the strain at 0. The abscissa should be defined from 0 to 1.
SHAPE	Shape factor for unloading. Active for nonzero values of the hysteretic unloading factor. Values less than one reduces the energy dissipation and greater than one increases dissipation, see also Figure 57.1.
FAIL	Failure option after cutoff stress is reached: EQ.0.0: tensile stress remains at cut-off value, EQ.1.0: tensile stress is reset to zero.
BVFLAG	Bulk viscosity activation flag, see remark below: EQ.0.0: no bulk viscosity (recommended), EQ.1.0: bulk viscosity active.
ED	Optional Young's relaxation modulus, E_d, for rate effects. See comments below.

VARIABLE	DESCRIPTION
BETA1	Optional decay constant, β_1.
KCON	Stiffness coefficient for contact interface stiffness. If undefined the maximum slope in stress vs. strain curve is used. When the maximum slope is taken for the contact, the time step size for this material is reduced for stability. In some cases Δt may be significantly smaller, and defining a reasonable stiffness is recommended.
REF	Use reference geometry to initialize the stress tensor. The reference geometry is defined by the keyword:*INITIAL_FOAM_REFERENCE_GEOMETRY (see there for more details). EQ.0.0: off, EQ.1.0: on.
TC	Tension cut-off stress
RFLAG	Rate type for input: EQ.0.0: LCID1 and LCID2 should be input as functions of true strain rate EQ.1.0: LCID1 and LCID2 should be input as functions of engineering strain rate.
DTRT	Strain rate averaging flag: EQ.0.0: use weighted running average LT.0.0: average the last 11 values GT.0.0: average over the last DTRT time units.
K	Material failure parameter that controls the volume enclosed by the failure surface. LE.0.0: ignore failure criterion; GT.0.0: use actual K value for failure criterions.
GAMA1	Material failure parameter, see equations below and Figure 181.1.
GAMA2	Material failure parameter, see equations below.
EH	Damage parameter.

Remarks:

This model is based on *MAT_LOW_DENSITY_FOAM. The uniaxial response is shown below with a large shape factor and small hysteretic factor. If the shape factor is not used, the unloading will occur on the loading curve for the second and subsequent cycles.

The damage is defined as the ratio of the current volume strain to the maximum volume strain, and it is used to interpolate between the responses defined by LCID1 and LCID2.

HU defines a hysteretic scale factor that is applied to the stress interpolated from LCID1 and LCID2,

$$\sigma = (HU + (1 - HU) \bullet \min(1, \frac{e_{\text{int}}}{e_{\text{int}}^{\max}})^S)\sigma[LCID1, LCID2]$$

where e_{int} is the internal energy and S is the shape factor. Setting HU to 1 results in a scale factor of 1. Setting HU close to zero scales the stress by the ratio of the internal energy to the maximum internal energy raised to the power S, resulting in the stress being reduced when the strain is low.

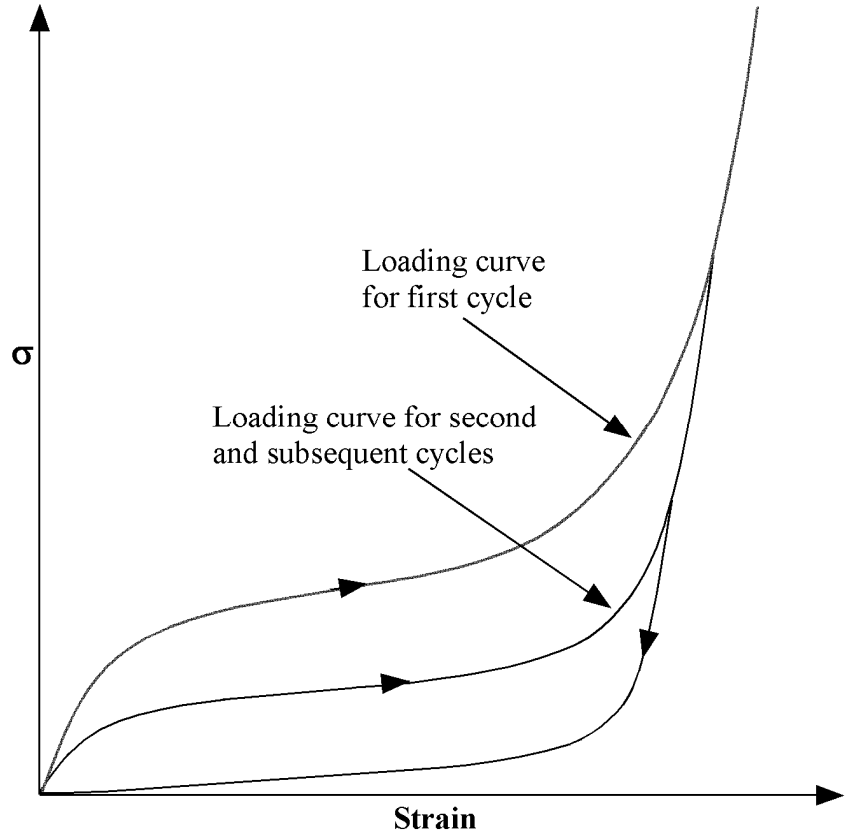


Figure 179.1. Loading and reloading curves.

***MAT_SIMPLIFIED_RUBBER/FOAM_**{*OPTION*}

This is Material Type 181. This material model provides a rubber and foam model defined by a single uniaxial load curve or by a family of uniaxial curves at discrete strain rates. The definition of hysteretic unloading is optional and can be realized via a single uniaxial unloading curve or a two-parameter formulation (starting with 971 release R5). The foam formulation is triggered by defining a Poisson's ratio. This material may be used with both shell and solid elements.

Available options include:

<BLANK>

WITH_FAILURE

When active, a strain based failure surface is defined suitable for incompressible polymers that models failure in both tension and compression.

This material law has been developed at DaimlerChrysler, Sindelfingen, in collaboration with Paul Du Bois, LSTC, and Prof. Dave J. Benson, UCSD.

Card 1

Variable	MID	RO	KM	MU	G	SIGF	REF	PRTEN
Type	A8	F	F	F	F	F	F	F

Card 2

Variable	SGL	SW	ST	LC/TBID	TENSION	RTYPE	AVGOPT	PR/BETA
Type	F	F	F	F	F	F	F	F

Define card 3 if and only if the option, WITH_FAILURE, is active.

Card 3

Variable	K	GAMA1	GAMA2	EH				
Type	F	F	F	F				

Optional card 3 (<BLANK> option) or 4 (WITH_FAILURE option)

Card 3/4	1	2	3	4	5	6	7	8
Variable	LCUNLD	HU	SHAPE					
Type	F	F	F					

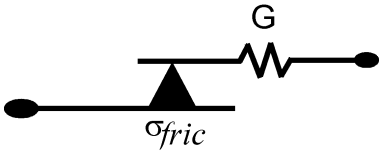
VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
KM	Linear bulk modulus.
MU	Damping coefficient.
G	Shear modulus for frequency independent damping. Frequency independent damping is based of a spring and slider in series. The critical stress for the slider mechanism is SIGF defined below. For the best results, the value of G should be 250-1000 times greater than SIGF.
SIGF	Limit stress for frequency independent, frictional, damping.
REF	Use reference geometry to initialize the stress tensor. The reference geometry is defined by the keyword:*INITIAL_FOAM_REFERENCE_GEOMETRY (see there for more details). EQ.0.0: off, EQ.1.0: on.
PRTEN	The tensile Poisson's ratio for shells (optional). If PRTEN is zero, PR/BETA will serve as the Poisson's ratio for both tension and compression in shells. If PRTEN is nonzero, PR/BETA will serve only as the compressive Poisson's ratio for shells.
SGL	Specimen gauge length
SW	Specimen width
ST	Specimen thickness

VARIABLE	DESCRIPTION
LC/TBID	Load curve or table ID, see *DEFINE_TABLE, defining the force versus actual change in the gauge length. If the table definition is used a family of curves are defined for discrete strain rates. The load curves should cover the complete range of expected loading, i.e., the smallest stretch ratio to the largest.
TENSION	Parameter that controls how the rate effects are treated. Applicable to the table definition. EQ.-1.0: rate effects are considered during tension and compression loading, but not during unloading, EQ. 0.0: rate effects are considered for compressive loading only, EQ.1.0: rate effects are treated identically in tension and compression.
RTYPE	Strain rate type if a table is defined: EQ.0.0: true strain rate, EQ.1.0: engineering strain rate
AVGOPT	Averaging option determine strain rate to reduce numerical noise. EQ.0.0: simple average of twelve time steps, EQ.1.0: running average of last 12 averages.
PR/BETA	If the value is specified between 0 and 0.5 exclusive, i.e., $0 < PR < 0.50$ the number defined here is taken as Poisson's ratio. If zero, an incompressible rubber like behavior is assumed and a default value of 0.495 is used internally. If a Poisson's ratio of 0.0 is desired, input a small value for PR such as 0.001. When fully integrated solid elements are used and when a nonzero Poisson's ratio is specified, a foam material is assumed and selective-reduced integration is not used due to the compressibility. This is true even if PR approaches 0.500. If any other value excluding zero is define, then BETA is taken as the absolute value of the given number and a nearly incompressible rubber like behavior is assumed. An incrementally updated mean viscous stress develops according to the equation: $p^{n+1} = p^n e^{-\beta \cdot \Delta t} + K\dot{\varepsilon}_{kk}\left(\frac{1-e^{-\beta \cdot \Delta t}}{\beta}\right) \qquad where\ \beta = \lvert BETA \rvert$ The BETA parameter does not apply to highly compressible foam materials.
K	Material failure parameter that controls the volume enclosed by the failure surface. LE.0.0: ignore failure criterion; GT.0.0: use actual K value for failure criterions.

VARIABLE	DESCRIPTION
GAMA1	Material failure parameter, see equations below and Figure 181.1.
GAMA2	Material failure parameter, see equations below.
EH	Damage parameter.
LCUNLD	Load curve, see *DEFINE_CURVE, defining the force versus actual length during unloading. The unload curve should cover exactly the same range as LC or the load curves of TBID and its end points should have identical values, i.e., the combination of LC and LCUNLD or the first curve of TBID and LCUNLD describes a complete cycle of loading and unloading. See also material *MAT_083.
HU	Hysteretic unloading factor between 0 and 1 (default=1., i.e. no energy dissipation), see also material *MAT_083 and Figure 57.1. This option is ignored if LCUNLD is used.
SHAPE	Shape factor for unloading. Active for nonzero values of the hysteretic unloading factor HU. Values less than one reduces the energy dissipation and greater than one increases dissipation, see also material *MAT_083 and Figure 57.1.

Remarks:

The frequency independent damping is obtained by the having a spring and slider in series as shown in the following sketch:



The general failure criterion for polymers is proposed by Feng and Hallquist as

$$f(I_1, I_2, I_3) = (I_1 - 3) + \Gamma_1(I_1 - 3)^2 + \Gamma_2(I_2 - 3) = K$$

where K is a material parameter which controls the size enclosed by the failure surface, and I_1, I_2 and I_3 are the three invariants of right Cauchy-Green deformation tensor ($\mathbf{C}$)

$$I_1 = \mathrm{C}_{ii} = \lambda_1^2 + \lambda_2^2 + \lambda_3^2$$

$$I_2 = \frac{1}{2}(\mathrm{C}_{ii}\mathrm{C}_{jj} - \mathrm{C}_{ij}\mathrm{C}_{ij}) = \lambda_1^2\,\lambda_2^2 + \lambda_1^2\,\lambda_3^2 + \lambda_2^2\,\lambda_3^2$$

$$I_3 = \det(\mathbf{C}) = \lambda_1^2\,\lambda_2^2\,\lambda_3^2$$

with λ_i are the stretch ratios in three principal directions.

To avoid sudden failure and numerical difficulty, material failure, which is usually a time point, is modeled as a process of damage growth. In this case, the two threshold values are chosen as (1 - *h*)*K* and *K*, where *h* (also called EH) is a small number chosen based on experimental results reflecting the range between damage initiation and material failure.
The damage is defined as function of f :

$$D = \begin{cases} 0 & \text{if} \quad f \le (1-h)K \\ \dfrac{1}{2}\left[1+\cos\dfrac{\pi(f-K)}{hK}\right] & \text{if} \quad (1-h)K < f < K \\ 1 & \text{if} \quad f \ge K \end{cases}$$

This definition indicates that damage is first-order continuous. Under this definition, the tangent stiffness matrix will be continuous. The reduced stress considering damage effect is

$$\sigma_{ij} = (1-D)\,\sigma_{ij}^{o}$$

where σ_{ij}^{o} is the undamaged stress. It is assumed that prior to final failure, material damage is recoverable. Once material failure occurs, damage will become permanent.

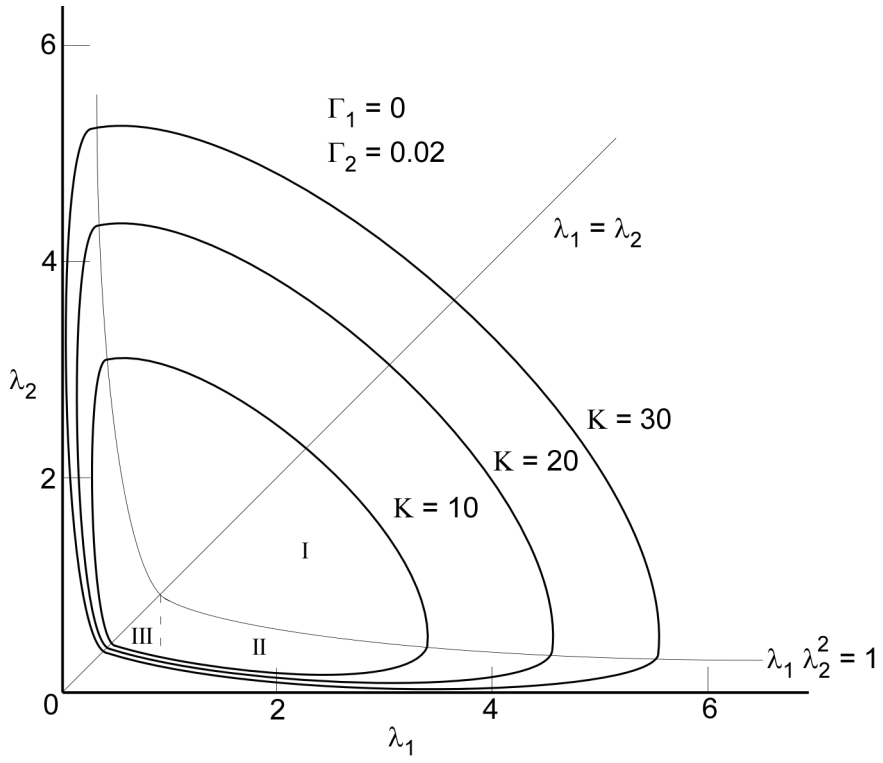


Figure 181.1. Failure surface for polymer.

*MAT_SIMPLIFIED_RUBBER_WITH_DAMAGE

This is Material Type 183. This material model provides an incompressible rubber model defined by a single uniaxial load curve for loading (or a table if rate effects are considered) and a single uniaxial load curve for unloading. This model is similar to *MAT_SIMPLIFIED_RUBBER/FOAM This material may be used with both shell and solid elements.

This material law has been developed at DaimlerChrysler, Sindelfingen, in collaboration with Paul Du Bois, LSTC, and Prof. Dave J. Benson, UCSD.

Card 1

	1	2	3	4	5	6	7	8
Variable	MID	RO	K	MU	G	SIGF		
Type	A8	F	F	F	F	F		

Card 2

Variable	SGL	SW	ST	LC/TBID	TENSION	RTYPE	AVGOPT	
Type	F	F	F	F	F	F	F	

Card 3

Variable	LCUNLD	REF						
Type	F	F						

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
K	Linear bulk modulus.
MU	Damping coefficient.

VARIABLE	DESCRIPTION
G	Shear modulus for frequency independent damping. Frequency independent damping is based of a spring and slider in series. The critical stress for the slider mechanism is SIGF defined below. For the best results, the value of G should be 250-1000 times greater than SIGF.
SIGF	Limit stress for frequency independent, frictional, damping.
SGL	Specimen gauge length
SW	Specimen width
ST	Specimen thickness
LC/TBID	Load curve or table ID, see *DEFINE_TABLE, defining the force versus actual change in the gauge length. If the table definition is used a family of curves are defined for discrete strain rates. The load curves should cover the complete range of expected loading, i.e., the smallest stretch ratio to the largest.
TENSION	Parameter that controls how the rate effects are treated. Applicable to the table definition. EQ.-1.0: rate effects are considered during tension and compression loading, but not during unloading, EQ. 0.0: rate effects are considered for compressive loading only, EQ.1.0: rate effects are treated identically in tension and compression.
RTYPE	Strain rate type if a table is defined: EQ.0.0: true strain rate, EQ.1.0: engineering strain rate
AVGOPT	Averaging option determine strain rate to reduce numerical noise. EQ.0.0: simple average of twelve time steps, EQ.1.0: running 12 point average.
LCUNLD	Load curve, see *DEFINE_CURVE, defining the force versus actual change in the gauge length during unloading. The unload curve should cover exactly the same range as LC (or as the first curve of table TBID) and its end points should have identical values, i.e., the combination of LC (or as the first curve of table TBID) and LCUNLD describes a complete cycle of loading and unloading.
REF	Use reference geometry to initialize the stress tensor. The reference geometry is defined by the keyword:*INITIAL_FOAM_REFERENCE_GEOMETRY (see there for more details). EQ.0.0: off, EQ.1.0: on.

*MAT_COHESIVE_ELASTIC

This is Material Type 184. It is a simple cohesive elastic model for use with solid element types 19 and 20, and is not available for other solid element formulation. See the remarks after *SECTION_SOLID for a description of element types 19 and 20.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	ROFLG	INTFAIL	ET	EN	FN_FAIL	
Type	A8	F	F	F	F	F	F	

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
ROFLG	Flag for whether density is specified per unit area or volume. ROFLG=0 specified density per unit volume (default), and ROFLG=1 specifies the density is per unit area for controlling the mass of cohesive elements with an initial volume of zero.
INTFAIL	The number of integration points required for the cohesive element to be deleted. If it is zero, the element won't be deleted even if it satisfies the failure criterion. The value of INTFAIL may range from 1 to 4, with 1 the recommended value.
ET	The stiffness in the plane of the cohesive element.
EN	The stiffness normal to the plane of the cohesive element.
FN_FAIL	The force in the normal direction for tensile failure.

Remarks:

This material cohesive model outputs three force resultants into the D3PLOT database rather than the usual six stress components. The in plane shear resultant along the 1-2 edge replaces the x-stress, the orthogonal in plane shear resultant replaces the y-stress, and the normal stress resultant replaces the z-stress.

*MAT_COHESIVE_TH

This is Material Type 185. It is a cohesive model by Tvergaard and Hutchinson [1992] for use with solid element types 19 and 20, and is not available for any other solid element formulation. See the remarks after *SECTION_SOLID for a description of element types 19 and 20. The implementation is based on the description of the implementation in the Sandia National Laboratory code, Tahoe [2003].

Card 1

Variable	MID	RO	ROFLG	INTFAIL	SIGMAX	NLS	TLS	
Type	A8	F	F	F	F	F	F	

Card 2

Variable	LAMDA1	LAMDA2	LAMDAF	STFSF				
Type	F	F	F	F				

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
ROFLG	Flag for whether density is specified per unit area or volume. ROFLG=0 specified density per unit volume (default), and ROFLG=1 specifies the density is per unit area for controlling the mass of cohesive elements with an initial volume of zero.
INTFAIL	The number of integration points required for the cohesive element to be deleted. If it is zero, the element won't be deleted even if it satisfies the failure criterion. The value of INTFAIL may range from 1 to 4, with 1 the recommended value.
SIGMAX	Peak traction.
NLS	Length scale (maximum separation) in the normal direction.
TLS	Length scale (maximum separation) in the tangential direction.

VARIABLE	DESCRIPTION
LAMDA1	Scaled distance to peak traction (Λ_1).
LAMDA2	Scaled distance to beginning of softening (Λ_2).
LAMDAF	Scaled distance for failure (Λ_{fail}).
STFSF	Penetration stiffness multiplier. The penetration stiffness, *PS*, in terms of input parameters becomes:

$$PS = \frac{STFSF * SIGMAX}{NLS * \left(\frac{LAMDA1}{LAMDAF} \right)}$$

Remarks:

In this cohesive material model, a dimensionless separation measure λ is used, which grasps for the interaction between relative displacements in normal (δ_3 - mode I) and tangential (δ_1, δ_2 - mode II) directions (see Figure 185.1 left):

$$\lambda = \sqrt{\left(\frac{\delta_1}{TLS} \right)^2 + \left(\frac{\delta_2}{TLS} \right)^2 + \left(\frac{\langle \delta_3 \rangle}{NLS} \right)^2}$$

where the Mc-Cauley bracket is used to distinguish between tension ($\delta_3 \geq 0$) and compression ($\delta_3 < 0$). *NLS* and *TLS* are critical values, representing the maximum separations in the interface in normal and tangential direction. For stress calculation, a trilinear traction-separation law is used, which is given by (see Figure 185.1 right):

$$t(\lambda) = \begin{cases} \sigma_{max} \dfrac{\lambda}{\Lambda_1/\Lambda_{fail}} & : \lambda < \Lambda_1/\Lambda_{fail} \\ \sigma_{max} & : \Lambda_1/\Lambda_{fail} < \lambda < \Lambda_2/\Lambda_{fail} \\ \sigma_{max} \dfrac{1-\lambda}{1-\Lambda_2/\Lambda_{fail}} & : \Lambda_2/\Lambda_{fail} < \lambda < 1 \end{cases}$$

With these definitions, the traction drops to zero when λ =1. Then, a potential ϕ is defined as:

$$\phi(\delta_1, \delta_2, \delta_3) = NLS \cdot \int_0^{\lambda} t(\bar{\lambda})\, d\bar{\lambda}$$

Finally, tangential components (t_1, t_2) and normal component (t_3) of the traction acting on the interface in the fracture process zone are given by:

$$t_{1,2} = \frac{\partial \phi}{\partial \delta_{1,2}} = \frac{t(\lambda)}{\lambda} \frac{\delta_{1,2}}{TLS} \frac{NLS}{TLS}, \quad t_3 = \frac{\partial \phi}{\partial \delta_3} = \frac{t(\lambda)}{\lambda} \frac{\delta_3}{NLS}$$

which in matrix notation is

$$\begin{bmatrix} t_1 \\ t_2 \\ t_3 \end{bmatrix} = \frac{t(\lambda)}{\lambda} \begin{bmatrix} \frac{NLS}{TLS^2} & 0 & 0 \\ 0 & \frac{NLS}{TLS^2} & 0 \\ 0 & 0 & \frac{1}{NLS} \end{bmatrix} \begin{bmatrix} \delta_1 \\ \delta_2 \\ \delta_3 \end{bmatrix}$$

In case of compression (δ_3<0), penetration is avoided by:

$$t_3 = \frac{STFSF \cdot \sigma_{\max}}{NLS \cdot \Lambda_1 / \Lambda_{fail}} \delta_3$$

Loading and unloading follows the same path, i.e. this model is completely reversible.

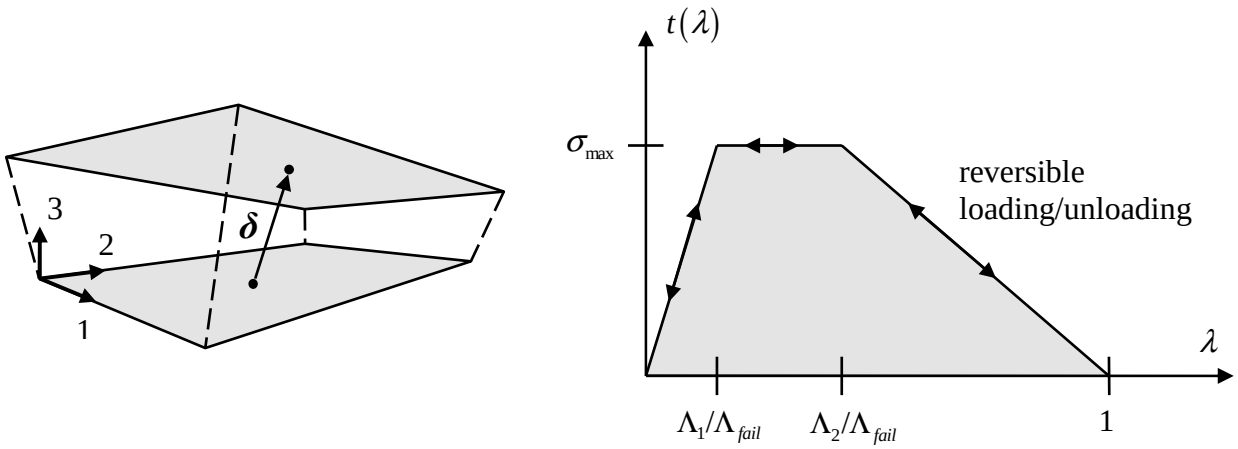


Figure 185.1. Relative displacement and trilinear traction-separation law.

This cohesive material model outputs three force resultants (tractions) into the D3PLOT database rather than the usual six stress components. The in plane shear resultant t_1 along the 1-2 edge replaces the x-stress, the orthogonal in plane shear resultant t_2 replaces the y-stress, and the normal stress resultant t_3 replaces the z-stress.

*MAT_COHESIVE_GENERAL

This is Material Type 186. This model includes three general irreversible mixed-mode interaction cohesive formulations with arbitrary normalized traction-separation law given by a load curve (TSLC). These three formulations are differentiated via the type of effective separation parameter (TES). The interaction between fracture modes I and II is considered, and irreversible conditions are enforced via a damage formulation (unloading/reloading path pointing to/from the origin). See remarks for details.

Card 1

Variable	MID	RO	ROFLG	INTFAIL	TES	TSLC	GIC	GIIC
Type	A8	F	F	F	F	F	F	F

Card 2

Variable	XMU	T	S	STFSF				
Type	F	F	F	F				

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
ROFLG	Flag for whether density is specified per unit area or volume. ROFLG=0 specifies density per unit volume (default), and ROFLG=1 specifies the density is per unit area for controlling the mass of cohesive elements with an initial volume of zero.
INTFAIL	Number of integration points required for a cohesive element to be deleted. If it is zero, the element won't be deleted even if it satisfies failure criterion. The value of INTFAIL may range from 1 to 4, with 1 the recommended value.
TES	Type of effective separation parameter (ESP). EQ. 0.0 or 1.0: a dimensional separation measure is used. For the interaction between mode I and II, a mixed-mode propagation criterion is used. For TES=0.0 this is a power-law, and for TES=1.0 this is the Benzeggagh-Kenane law [1996]. See remarks below.

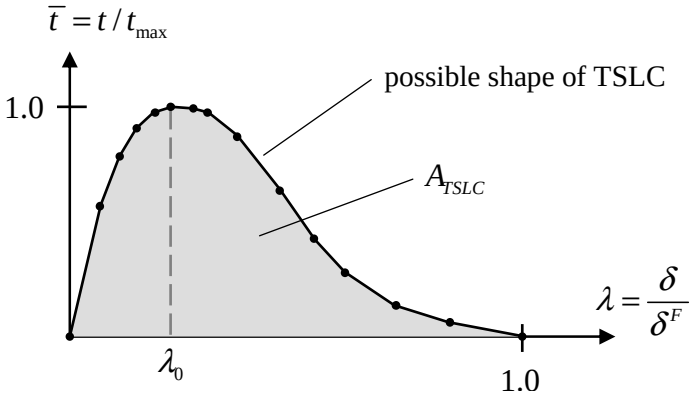
VARIABLE	DESCRIPTION
	EQ. 2.0: a dimensionless separation measure is used, which grasps for the interaction between mode I displacements and mode II displacements (similar to MAT_185, but with damage and general traction-separation law). See remarks below.
TSLC	Normalized traction-separation load curve ID. The curve must be normalized in both coordinates and must contain at least three points: (0.0, 0.0), (λ_0, 1.0), and (1.0, 0.0), which represents the origin, the peak and the complete failure, respectively (see Figure 186.1). A platform can exist in the curve like the tri-linear TSLC (see MAT_185).
GIC	Fracture toughness / energy release rate G_I^c for mode I
GIIC	Fracture toughness / energy release rate G_{II}^c for mode II
XMU	Exponent that appears in the power failure criterion (TES=1.0) or the Benzeggagh-Kenane failure criterion (TES=2.0). Recommended values for XMU are between 1.0 and 2.0.
T	Peak traction in normal direction (mode I)
S	Peak traction in tangential direction (mode II)
STFSF	Penetration stiffness multiplier for compression. Factor = (1.0+STFSF) is used to scale the compressive stiffness, i.e. no scaling is done with STFSF=0.0 (recommended).

Remarks:

All three formulations have in common, that the traction-separation behavior of this model is mainly given by G_I^c and T for normal mode I, G_{II}^c and S for tangential mode II and an arbitrary normalized traction-separation load curve for both modes (see Figure 186.1). The maximum (or failure) separations are then given by:

$$\delta_I^F = \frac{G_I^c}{A_{TSLC} \cdot T} \ , \ \ \delta_{II}^F = \frac{G_{II}^c}{A_{TSLC} \cdot S}$$

where A_{TSLC} is the area under the normalized traction-separation curve.



	mode I	mode II
t_{max}	T	S
δ^F	$\frac{G_I^c}{A_{TSLC}T}$	$\frac{G_{II}^c}{A_{TSLC}S}$
G^c	G_I^c	G_{II}^c

Figure 186.1. Normalized traction-separation law

For mixed-mode behavior, three different formulations are possible (where default TES=0.0 with XMU=1.0 is recommended as first guess):

First and second formulation (TES=0.0 and TES=1.0)

Here, the total mixed-mode relative displacement δ_m is defined as $\delta_m = \sqrt{\delta_I^2 + \delta_{II}^2}$, where $\delta_I = \delta_3$ is the separation in normal direction (mode I) and $\delta_{II} = \sqrt{\delta_1^2 + \delta_2^2}$ is the separation in tangential direction (mode II) (see Figure 186.2). The ultimate mixed-mode displacement δ^F (total failure) for the power law (TES=0.0) is:

$$\delta^F = \frac{1+\beta^2}{A_{TSLC}}\left[\left(\frac{T}{G_I^c}\right)^{XMU} + \left(\frac{S\cdot\beta^2}{G_{II}^c}\right)^{XMU}\right]^{-\frac{1}{XMU}}$$

and alternatively for the Benzeggagh-Kenane law [1996] (TES=1.0):

$$\delta^F = \frac{1+\beta^2}{A_{TSLC}\left(T+\beta^2 S\right)}\left[G_I^c + \left(G_{II}^c - G_I^c\right)\left(\frac{\beta^2\cdot S}{T+\beta^2\cdot S}\right)^{XMU}\right]$$

where $\beta = \delta_{II} / \delta_I$ is the "mode mixity". The larger the exponent *XMU* is chosen, the larger the fracture toughness in mixed-mode situations will be. In this model, damage of the interface is considered, i.e. irreversible conditions are enforced with loading/unloading paths coming from/pointing to the origin. This formulation is similar to MAT_COHESIVE_MIXED_MODE (MAT_138), but with the arbitrary traction-separation law TSLC.

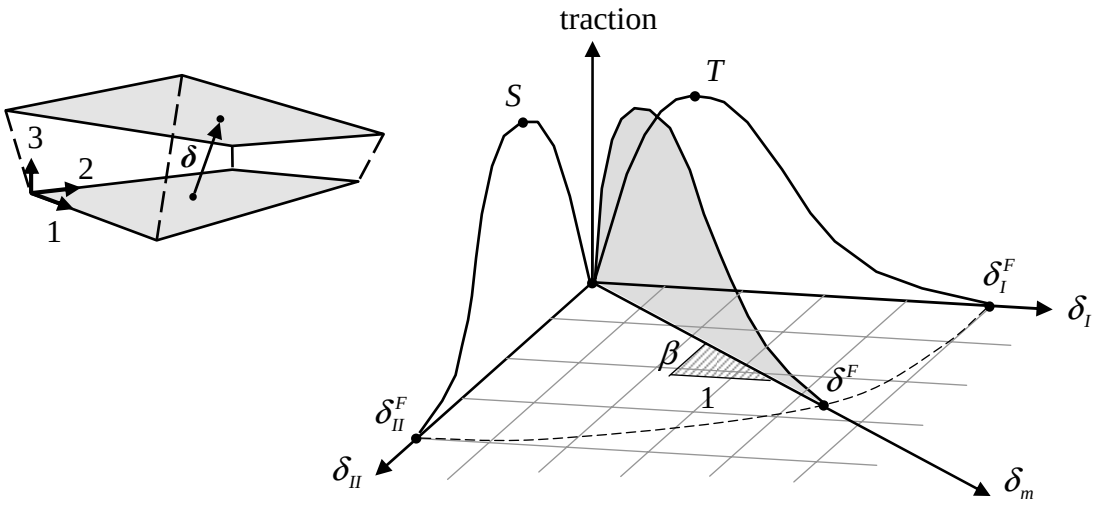


Figure 186.2. Mixed-mode traction-separation law

Third formulation (TES=2.0)

Here, a dimensionless effective separation parameter λ is used, which grasps for the interaction between relative displacements in normal (δ_3 - mode I) and tangential (δ_1, δ_2 - mode II) directions:

$$\lambda = \sqrt{\left(\frac{\delta_1}{\delta_{II}^F}\right)^2 + \left(\frac{\delta_2}{\delta_{II}^F}\right)^2 + \left\langle\frac{\delta_3}{\delta_I^F}\right\rangle^2}$$

where the Mc-Cauley bracket is used to distinguish between tension ($\delta_3 \geq 0$) and compression ($\delta_3 < 0$). δ_I^F and δ_{II}^F are critical values, representing the maximum separations in the interface in normal and tangential direction . For stress calculation, the normalized traction-separation load curve TSLC is used: $t = t_{\max} \cdot \bar{t}(\lambda)$. This formulation is similar to MAT_COHESIVE_TH (MAT_185), but with the arbitrary traction-separation law and a damage formulation (i.e. irreversible conditions are enforced with loading/unloading paths coming from/pointing to the origin).

***MAT_SAMP-1**

Purpose: This is Material Type 187 (Semi-Analytical Model for Polymers). This material model uses an isotropic C-1 smooth yield surface for the description of non-reinforced plastics. Details of the implementation are given in [Kolling, Haufe, Feucht and Du Bois 2005].

This material law has been developed at DaimlerChrysler, Sindelfingen, in collaboration with Paul Du Bois and Dynamore, Stuttgart.

Card 1

1	2	3	4	5	6	7	8
MID	RO	BULK	GMOD	EMOD	NUE	RBCFAC	NUMINT
A8	F	F	F	F	F	F	

Card 2

LCID-T	LCID-C	LCID-S	LCID-B	NUEP	LCID-P		INCDAM
I	I	I	I	F	I		

Card 3

LCID-D	EPFAIL	DEPRPT	LCID_TRI	LCID_LC			
I	F	F	I	I			

Card 4

MITER	MIPS		INCFAIL	ICONV	ASAF		
I	I		I	I	F		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
BULK	Bulk modulus, used by LS-DYNA in the time step calculation
GMOD	Shear modulus, used by LS-DYNA in the time step calculation
EMOD	Young's modulus
NUE	Poisson ratio
RBCFAC	Ratio of yield in biaxial compression vs. yield in uniaxial compression. If nonzero this will activate the use of a multi-linear yield surface. Default is 0.
LCID-T	Load curve or table ID giving the yield stress as a function of plastic strain, these curves should be obtained from quasi-static and (optionally) dynamic uniaxial tensile tests, this input is mandatory and the material model will not work unless at least one tensile stress-strain curve is given.
LCID-C	Load curve ID giving the yield stress as a function of plastic strain, this curve should be obtained from a quasi-static uniaxial compression test, this input is optional.
LCID-S	Load curve ID giving the yield stress as a function of plastic strain, this curve should be obtained from a quasi-static shear test, this input is optional.
LCID-B	Load curve ID giving the yield stress as a function of plastic strain, this curve should be obtained from a quasi-static biaxial tensile test, this input is optional.
NUEP	Plastic Poisson's ratio: an estimated ratio of transversal to longitudinal plastic rate of deformation under uniaxial loading should be given.
LCID-P	Load curve ID giving the plastic Poisson ratio as a function of equivalent plastic deformation during uniaxial tensile and compressive testing. Abcissa should be negative for plastic strains in compression and positive for plastic strains in tension, it is important to cover both tension and compression. If the load curve is given, the constant value in the previous field will be ignored. This input is optional.
INCDAM	Flag to control the damage evolution as a function of triaxiality. If INCDAM=0 damage evolution is independent of the triaxialty. If INCDAM=1 an incremental formulation is used to compute the damage.

VARIABLE	DESCRIPTION
LCID-D	Load curve ID giving the damage parameter as a function of equivalent plastic strain during uniaxial tensile testing. By default this option assumes that effective (i.e. undamaged) yield values are used in the load curves LCID-T, LCID-C, LCID-S and LCID-B. If LCID-D is given a negative value, true (i.e. damaged) yield stress values can be used. In this case an automatic stress-strain recalibration (ASSR) algorithm is activated. The damage value must be defined in the range 0<=d<1.
EPFAIL	This parameter is the equivalent plastic strain at failure. If EPFAIL is given as a negative integer, a load curve is expected that defines EPFAIL as a function of the plastic strain rate. Default value is 1.0e+5
DEPRPT	Increment of equivalent plastic strain between failure point and rupture point. Stresses will fade out to zero between EPFAIL and EPFAIL+DEPRPT. If DEPRPT is given a negative value a curve definition is expected where DEPRPT is defined as function of the triaxiality.
LCID_TRI	Load curve that specifies a factor that works multiplicatively on the value of EPFAIL depending on the triaxiality (i.e. pressure/sigma_vm). For a triaxiality of -1/3 a value of 1.0 should be specified.
LCID_LC	Load curve that specifies a factor that works multiplicatively on the value of EPFAIL depending on a characteristic element length.
MITER	Maximum number of iterations in the cutting plane algorithm, default is set to 400
MIPS	Maximum number of iterations in the secant iteration performed to enforce plane stress (shell elements only), default set to 10
INCFAIL	Flag to control the failure evolution as a function of triaxiality. If INCFAIL=0 failure evolution is independent of the triaxiality. If INCFAIL=1 an incremental formulation is used to compute the failure value. If INCFAIL=-1 the failure model is deactivated.
ICONV	Formulation flag: ICONV=0: default ICONV=1: yield surface is internally modified by increasing the shear yield until a convex yield surface is achieved
ASAF	Safety factor, used only if ICONV=1, values between 1 and 2 can improve convergence, however the shear yield will be artificially increased if this option is used, default is set to 1.

Remarks:

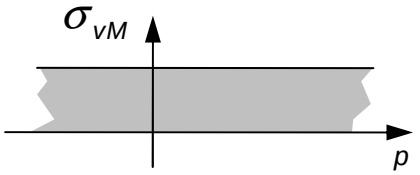
1. Material SAMP-1 uses three yield curves internally, hence the yield surface has a quadratic shape in general. If less than three curves are defined the remaining curves are generated internally as follows:

$$\left.\begin{aligned} LCID\_C &= 0 \\ LCID\_S &= 0 \\ LCID\_B &= 0 \end{aligned}\right\} \Rightarrow \begin{cases} \sigma_c = \sigma_t \\ \sigma_s = \dfrac{\sigma_t}{\sqrt{3}} \end{aligned}$$

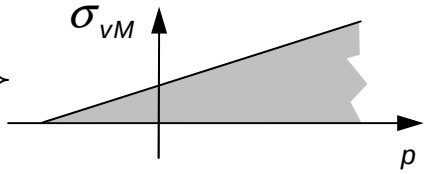
$$\left.\begin{aligned} LCID\_C &= 0 \\ LCID\_S &\neq 0 \\ LCID\_B &= 0 \end{aligned}\right\} \Rightarrow \sigma_c = \frac{\sqrt{3}\sigma_t\sigma_s}{\left(2\sigma_t - \sqrt{3}\sigma_s\right)}$$

$$\left.\begin{aligned} LCID\_C &\neq 0 \\ LCID\_S &= 0 \\ LCID\_B &= 0 \end{aligned}\right\} \Rightarrow \sigma_s = \frac{2\sigma_t\sigma_c}{\sqrt{3}\left(\sigma_t + \sigma_c\right)}$$

$$\left.\begin{aligned} LCID\_C &= 0 \\ LCID\_S &= 0 \\ LCID\_B &\neq 0 \end{aligned}\right\} \Rightarrow \begin{cases} \sigma_c = \dfrac{\sigma_t\sigma_b}{\left(3\sigma_b - 2\sigma_t\right)} \\ \sigma_s = \dfrac{\sigma_t\sigma_b}{\sqrt{3}\left(2\sigma_b - \sigma_t\right)} \end{cases}$$



von Mises cylinder



Drucker-Prager cone

A linear yield surface in the invariant space spanned by the pressure and the von Mises stress is generated using the available data points.

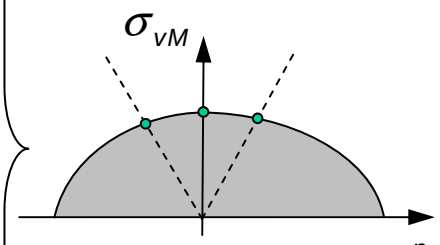
If more then 2 load curves are available the following cases can be distinguished:

$$LCID\_S \neq 0 \quad \Rightarrow SAMP-1$$
$$LCID\_B = 0$$

$$\left.\begin{array}{ll} LCID\_C & \neq 0 \\ LCID\_S & = 0 \\ LCID\_B & \neq 0 \end{array}\right\} \Rightarrow \sigma_s \frac{1}{\sqrt{3}} \sqrt{\frac{3\sigma_b^2 \sigma_c \sigma_t}{(2\sigma_b + \sigma_c)(2\sigma_b - \sigma_t)}}$$

$$\left.\begin{array}{ll} LCID\_C & = 0 \\ LCID\_S & \neq 0 \\ LCID\_B & \neq 0 \end{array}\right\} \Rightarrow \sigma_c = \frac{6\left(162\sigma_b^2\sigma_s^2 + \sigma_b\sigma_s^2\sigma_t\right)}{6\sigma_b\sigma_s^2 + 323\sigma_b^2\sigma_t + 3\sigma_s^2\sigma_t}$$

$$\left.\begin{array}{ll} LCID\_C & \neq 0 \\ LCID\_S & \neq 0 \\ LCID\_B & \neq 0 \end{array}\right\} \Rightarrow \textit{least squares}$$



SAMP-1 yield surface defined through load curves

2. If the LCID_D is given, then a damage curve as a function of equivalent plastic strains acting on the stresses is defined as depicted in the following picture. EPFAIL and DEPRPT defined the failure and fading behaviour of a single element:

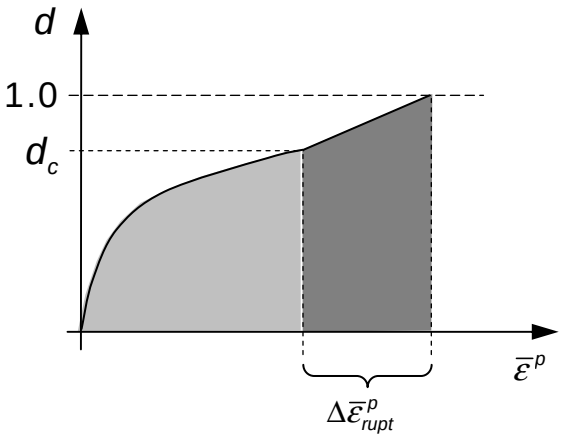
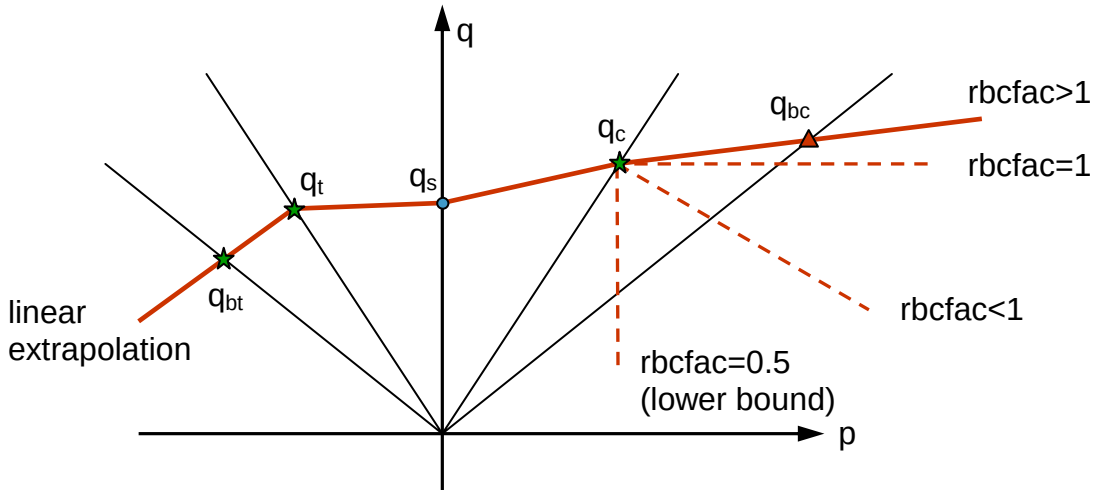


Figure 187.1

Since the damaging curve acts on the yield values, the inelastic results are effected by the damage curve. As a means to circumvent this, the load curve LCID-D may be given a negative ID. This will lead to an internal conversion of from nominal to effective stresses (ASSR).

3. Since the generality of arbitrary curve inputs allows to generate unsolvable yield surfaces, SAMP may modify curves internally. This will always lead to warning messages at the beginning of the simulation run. One modification that is not allowed are negative tangents of the last two data points of any of the yield curves.

4. If RBCFAC is nonzero the yield surface in I_1-σ_{vm}-stress space is constructed such, that a multi-linear yield surface is gained. RBCFAC allows to modify then behavior in biaxial compression.



q	von Mises stress
p	pressure
★	required input data
●	optional input data
▲	extrapolated data

bt	biaxial tension
t	tension
s	shear
c	compression
bc	biaxial

$$\text{rbcfac} \quad \frac{q_{bc}}{q_c}$$

*MAT_THERMO_ELASTO_VISCOPLASTIC_CREEP

This is Material Type 188. In this model, creep is described separately from plasticity using Garafalo's steady-state hyperbolic sine creep law. Viscous effects of plastic strain rate are considered using the Cowper-Symonds model. Young's modulus, Poisson's ratio, thermal expansion coefficient, yield stress, material parameters of Cowper-Symonds model as well as the isotropic and kinematic hardening parameters are all assumed to be temperature dependent. Application scope includes: simulation of solder joints in electronic packaging, modeling of tube brazing process, creep age forming, etc.

Card 1

1	2	3	4	5	6	7	8
MID	RO	E	PR	SIGY	ALPHA	LCSS	REFTEM
A8	F	F	F	F	F	F	F

Card 2

1	2	3	4	5	6	7	8
QR1	CR1	QR2	CR2	QX1	CX1	QX2	CX2
F	F	F	F	F	F	F	F

Card 3

1	2	3	4	5	6	7	8
C	P	LCE	LCPR	LCSIGY	LCQR	LCQX	LCALPH
F	F	F	F	F	F	F	F

Card 4

1	2	3	4	5	6	7	8
LCC	LCP	LCCR	LCCX	CRPA	CRPB	CRPQ	CRPM
F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
E	Young's modulus
PR	Poisson's ratio
SIGY	Initial yield stress
ALPHA	Thermal expansion coefficient
LCSS	Load curve ID or Table ID. The load curve ID defines effective stress versus effective plastic strain. The table ID defines for each temperature value a load curve ID giving the stress versus effective plastic strain for that rate. The stress versus effective plastic strain curve for the lowest value of temperature is used if the temperature falls below the minimum value. Likewise, the stress versus effective plastic strain curve for the highest value of temperature is used if the temperature exceeds the maximum value. Card 2 is ignored with this option.
REFTEM	Reference temperature that defines thermal expansion coefficient
QR1	Isotropic hardening parameter Q_{r1}
CR1	Isotropic hardening parameter C_{r1}
QR2	Isotropic hardening parameter Q_{r2}
CR2	Isotropic hardening parameter C_{r2}
QX1	Kinematic hardening parameter $Q_{\chi 1}$
CX1	Kinematic hardening parameter $C_{\chi 1}$
QX2	Kinematic hardening parameter $Q_{\chi 2}$
CX2	Kinematic hardening parameter $C_{\chi 2}$
C	Viscous material parameter C
P	Viscous material parameter P
LCE	Load curve for scaling Young's modulus as a function of temperature

VARIABLE	DESCRIPTION
LCPR	Load curve for scaling Poisson's ratio as a function of temperature
LCSIGY	Load curve for scaling initial yield stress as a function of temperature
LCQR	Load curve for scaling the isotropic hardening parameters QR1 and QR2 or the stress given by the load curve LCSS as a function of temperature
LCQX	Load curve for scaling the kinematic hardening parameters QX1 and QX2 as a function of temperature
LCALPH	Load curve for scaling the thermal expansion coefficient as a function of temperature
LCC	Load curve for scaling the viscous material parameter C as a function of temperature
LCP	Load curve for scaling the viscous material parameter P as a function of temperature
LCCR	Load curve for scaling the isotropic hardening parameters CR1 and CR2 as a function of temperature
LCCX	Load curve for scaling the isotropic hardening parameters CX1 and CX2 as a function of temperature
CRPA	Constant A of Garafalo's hyperbolic sine creep law (see Remarks)
CRPB	Constant B of Garafalo's hyperbolic sine creep law (see Remarks)
CRPQ	Constant Q of Garafalo's hyperbolic sine creep law (see Remarks)
CRPM	Constant m of Garafalo's hyperbolic sine creep law (see Remarks)

Remarks:

If LCSS is not given any value the uniaxial stress-strain curve has the form

$$\sigma(\varepsilon_{eff}^{p}) = \sigma_0 + Q_{r1}(1-\exp(-C_{r1}\varepsilon_{eff}^{p})) + Q_{r2}(1-\exp(-C_{r2}\varepsilon_{eff}^{p}))$$
$$+Q_{\chi 1}(1-\exp(-C_{\chi 1}\varepsilon_{eff}^{p})) + Q_{\chi 2}(1-\exp(-C_{\chi 2}\varepsilon_{eff}^{p}))$$

Viscous effects are accounted for using the Cowper-Symonds model, which scales the yield stress with the factor:

$$1+\left(\frac{\dot{\varepsilon}_{eff}^{p}}{C}\right)^{1/p}$$

The steady-state creep strain rate of Garafalo's hyperbolic sine equation is given by

$$\dot{\varepsilon}^c = A\left[\sinh(B\tau^e)\right]^m \exp\left(-\frac{Q}{T}\right)$$

*MAT_ANISOTROPIC_THERMOELASTIC

This is Material Type 189. This model characterizes elastic materials whose elastic properties are temperature-dependent.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	TA1	TA2	TA3	TA4	TA5	TA6
Type	A8	F	F	F	F	F	F	F

Card 2

Variable	C11	C12	C13	C14	C15	C16	C22	C23
Type	F	F	F	F	F	F	F	F

Card 3

Variable	C24	C25	C26	C33	C34	C35	C36	C44
Type	F	F	F	F	F	F	F	F

Card 4

Variable	C45	C46	C55	C56	C66	TGE	TREF	AOPT
Type	F	F	F	F	F	F	F	F

Card 5

Variable	XP	YP	ZP	A1	A2	A3	MACF	
Type	F	F	F	F	F	F	F	

Card 6	1	2	3	4	5	6	7	8
Variable	V1	V2	V3	D1	D2	D3	BETA	REF
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
TAi	Curve IDs defining the coefficients of thermal expansion for the six components of strain tensor as function of temperature.
CIJ	Curve IDs defining the 6×6 symmetric constitutive matrix in material coordinate system as function of temperature. Note that 1 corresponds to the *a* material direction, 2 to the *b* material direction, and 3 to the *c* material direction.
TGE	Curve ID defining the structural damping coefficient as function of temperature.
TREF	Reference temperature for the calculation of thermal loads or the definition of thermal expansion coefficients.
AOPT	Material axes option, (see MAT_ANISOTROPIC_ELASTIC/MAT_002 for a complete description.) EQ.0.0: locally orthotropic with material axes determined by element nodes 1, 2, and 4, as with *DEFINE_COORDINATE_NODES. EQ.1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center; this is the a-direction. EQ.2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR. EQ.3.0: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle, BETA, from a line in the plane of the element defined by the cross product of the vector **v** with the element normal. EQ.4.0: locally orthotropic in cylindrical coordinate system with the material axes determined by a vector **v**, and an originating point, P, which define the centerline axis.

VARIABLE	DESCRIPTION
	LT.0.0: the absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_NODES, *DEFINE_COORDINATE_SYSTEM or *DEFINE_COORDINATE_VECTOR). Available in R3 version of 971 and later.
XP, YP, ZP	XP, YP, ZP define coordinates of point **p** for AOPT=1 and 4.
A1, A2, A3	a1, a2, a3 define components of vector **a** for AOPT=2.
MACF	Material axis change flag for brick elements (see MAT_002 for a complete description.)
D1, D2, D3	d1, d2, d3 define components of vector **d** for AOPT=2.
V1, V2, V3	v1, v2, v3 define components of vector **v** for AOPT=3 and 4.
BETA	Material angle in degrees for AOPT=3, may be overwritten on the element card, see *ELEMENT_SHELL_BETA or *ELEMENT_SOLID_ORTHO.
REF	Use initial geometry to initialize the stress tensor (see MAT_002 for a complete description.)

*MAT_FLD_3-PARAMETER_BARLAT

This is Material Type 190. This model was developed by Barlat and Lian [1989] for modeling sheets with anisotropic materials under plane stress conditions. This material allows the use of the Lankford parameters for the definition of the anisotropy. This particular development is due to Barlat and Lian [1989]. It has been modified to include a failure criterion based on the Forming Limit Diagram. The curve can be input as a load curve, or calculated based on the n-value and sheet thickness.

Card 1

Variable	MID	RO	E	PR	HR	P1	P2	ITER
Type	A8	F	F	F	F	F	F	F

Card 2

Variable	M	R00	R45	R90	LCID	E0	SPI	P3
Type	F	F	F	F	I	F	F	F

Card 3

Variable	AOPT	C	P	FLDCID	RN	RT	FLDSAFE	FLDNIPF
Type	F	F	F	I	F	F	F	I

Card 4

Variable				A1	A2	A3		
Type				F	F	F		

Card 5	1	2	3	4	5	6	7	8
Variable	V1	V2	V3	D1	D2	D3	BETA	
Type	F	F	F	F	F	F	F	

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
E	Young's modulus, E
PR	Poisson's ratio, ν
HR	Hardening rule: EQ.1.0: linear (default), EQ.2.0: exponential (Swift) EQ.3.0: load curve EQ.4.0: exponential (Voce) EQ.5.0: exponential (Gosh) EQ.6.0: exponential (Hocket-Sherby)
P1	Material parameter: HR.EQ.1.0: Tangent modulus, HR.EQ.2.0: k, strength coefficient for Swift exponential hardening HR.EQ.4.0: a, coefficient for Voce exponential hardening HR.EQ.5.0: k, strength coefficient for Gosh exponential hardening HR.EQ.6.0: a, coefficient for Hocket-Sherby exponential hardening
P2	Material parameter: HR.EQ.1.0: Yield stress HR.EQ.2.0: n, exponent for Swift exponential hardening HR.EQ.4.0: c, coefficient for Voce exponential hardening HR.EQ.5.0: n, exponent for Gosh exponential hardening HR.EQ.6.0: c. coefficient for Hocket-Sherby exponential hardening
ITER	Iteration flag for speed: ITER.EQ.0.0: fully iterative ITER.EQ.1.0: fixed at three iterations Generally, ITER=0 is recommended. However, ITER=1 is somewhat faster and may give acceptable results in most problems.

VARIABLE	DESCRIPTION
M	m, exponent in Barlat's yield surface
R00	R_{00}, Lankford parameter determined from experiments
R45	R_{45}, Lankford parameter determined from experiments
R90	R_{90}, Lankford parameter determined from experiments
LCID	load curve ID for the load curve hardening rule
E0	Material parameter HR.EQ.2.0: ε_0 for determining initial yield stress for Swift exponential hardening. (Default=0.0) HR.EQ.4.0: b, coefficient for Voce exponential hardening HR.EQ.5.0: ε_0 for determining initial yield stress for Gosh exponential hardening. (Default=0.0) HR.EQ.6.0: b, coefficient for Hocket-Sherby exponential hardening
SPI	*spi*, if ε_0 is zero above and HR.EQ.2.0. (Default=0.0) EQ.0.0: $\varepsilon_0 = (E/k)^{**}[1/(n-1)]$ LE.0.2: $\varepsilon_0 = spi$ GT.0.2: $\varepsilon_0 = (spi/k)^{**}[1/n]$
P3	Material parameter: HR.EQ.5.0: p, parameter for Gosh exponential hardening HR.EQ.6.0: n, exponent for Hocket-Sherby exponential hardening
AOPT	Material axes option (see MAT_*OPTION* TROPIC_ELASTIC for a more complete description): EQ.0.0: locally orthotropic with material axes determined by element nodes 1, 2, and 4, as with *DEFINE_COORDINATE_NODES. EQ.2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR. EQ.3.0: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle, BETA, from a line in the plane of the element defined by the cross product of the vector v with the element normal. LT.0.0: the absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_NODES, *DEFINE_COORDINATE_SYSTEM or *DEFINE_COORDINATE_VECTOR). Available in R3 version of 971 and later.

VARIABLE	DESCRIPTION
C	C in Cowper-Symonds strain rate model
P	p in Cowper-Symonds strain rate model, p=0.0 for no strain rate effects
FLDCID	Load curve ID defining the Forming Limit Diagram. Minor strains in percent are defined as abscissa values and Major strains in percent are defined as ordinate values. The forming limit diagram is shown in Figure 39.1. In defining the curve list pairs of minor and major strains starting with the left most point and ending with the right most point, see *DEFINE_CURVE.
RN	Hardening exponent equivalent to the n-value in a power law hardening law. If the parameter FLDCID is not defined, this value in combination with the value RT can be used to calculate a forming limit curve to allow for failure.
RT	Sheet thickness used for calculating a forming limit curve. This value does not override the sheet thickness in any way. It is only used in conjunction with the parameter RN to calculate a forming limit curve if the parameter FLDCID is not defined.
FLDSAFE	A safety offset of the forming limit curve. This value should be input as a percentage (ex. 10 not 0.10). This safety margin will be applied to the forming limit curve defined by FLDCID or the curve calculated by RN and RT.
FLDNIPF	The number of element integration points that must fail before the element is deleted. By default, if one integration point has strains above the forming limit curve, the element is flagged for deletion.
A1 A2 A3	Components of vector **a** for AOPT = 2.
V1 V2 V3	Components of vector **v** for AOPT = 3.
D1 D2 D3	Components of vector **d** for AOPT = 2.
BETA	Material angle in degrees for AOPT = 3, may be overridden on the element card, see *ELEMENT_SHELL_BETA.

Remarks:

See material 36 for the theoretical basis.

The forming limit curve can be input directly as a curve by specifying a load curve id with the parameter FLDCID. When defining such a curve, the major and minor strains must be input as percentages.

Alternatively, the parameters RN and RT can be used to calculate a forming limit curve. The use of RN and RT is not recommended for non-ferrous materials. RN and RT are ignored if a non-zero FLDCID is defined.

The first history variable is the maximum strain ratio defined by:

$$\frac{\varepsilon_{major_{workpiece}}}{\varepsilon_{major_{fld}}}$$

corresponding to $\varepsilon_{\text{minor}_{workpiece}}$. A value between 0 and 1 indicates that the strains lie below the forming limit curve. Values above 1 indicate that the strains are above the forming limit curve.

***MAT_SEISMIC_BEAM**

Purpose: This is Material Type 191. This material enables lumped plasticity to be developed at the 'node 2' end of Belytschko-Schwer beams (resultant formulation). The plastic yield surface allows interaction between the two moments and the axial force.

Note: The following options will be available in release 3 of version 971: DEGRAD, IFEMA, FOFFS, FEMAi.

Card 1

Variable	MID	RO	E	PR	AOPT	FTYPE	DEGRAD	IFEMA
Type	A8	F	F	F	F	I	I	I
Default	none	none	none	none	0.1	1	0	0

Card 2

Variable	LCPMS	SFS	LCPMT	SFT	LCAT	SFAT	LCAC	SFAC
Type	F	F	F	F	F	F	F	F
Default	none	1.0	LCMPS	1.0	none	1.0	LCAT	1.0

Define the following card for interaction formulation, FTYPE, type 1 (Default)

Card 3

Variable	ALPHA	BETA	GAMMA	DELTA	A	B	FOFFS	
Type	F	F	F	F	F	F	F	
Default	see note	see note	see note	see note	see note	see note	0.0	

Define the following card for interaction formulation, FTYPE, type 2

Card 3	1	2	3	4	5	6	7	8
Variable	SIGY	D	W	TF	TW			
Type	F	F	F	F	F			
Default	none	none	none	none	none			

Define the following card for FEMA limits only if IFEMA > 0

Card 4	1	2	3	4	5	6	7	8
Variable	PR1	PR2	PR3	PR4				
Type	F	F	F	F				
Default	0	0	0	0				

Define the following card for FEMA limits only if IFEMA = 2

Card 4	1	2	3	4	5	6	7	8
Variable	TS1	TS2	TS3	TS4	CS1	CS2	CS3	CS4
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	=TS1	=TS2	=TS3	=TS4

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.

VARIABLE	DESCRIPTION
AOPT	Axial force option EQ.0.0: Axial load curves are collapse load vs. strain EQ.1.0: Axial load curves are collapse load vs. change in length EQ.2.0: Axial load curves are collapse load vs. nominal total strain ("total strain" is elastic+plastic strains) LT.0.0: the absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_NODES, *DEFINE_COORDINATE_SYSTEM or *DEFINE_COORDINATE_VECTOR). Available in R3 version of 971 and later.
FTYPE	Formulation type for interaction EQ.1: Parabolic coefficients, axial load and biaxial bending (default). EQ.2: Japanese code, axial force and major axis bending.
DEGRADE	Flag for degrading moment behavior (see Remarks) EQ.0: Behavior as in previous versions EQ.1: Fatigue-type degrading moment-rotation behavior EQ.2: FEMA-type degrading moment-rotation behavior
IFEMA	Flag for input of FEMA thresholds EQ.0: No input EQ.1: Input of rotation thresholds only EQ.2: Input of rotation and axial strain thresholds
LCPMS	Load curve ID giving plastic moment vs. Plastic rotation at node 2 about local s-axis. See *DEFINE_CURVE.
SFS	Scale factor on s-moment at node 2.
LCPMT	Load curve ID giving plastic moment vs. Plastic rotation at node 2 about local t-axis. See *DEFINE_CURVE.
SFT	Scale factor on t-moment at node 2.
LCAT	Load curve ID giving axial tensile yield force vs. total tensile (elastic + plastic) strain or vs. elongation. See AOPT above. All values are positive. See *DEFINE_CURVE.
SFAT	Scale factor on axial tensile force.
LCAC	Load curve ID giving compressive yield force vs. total compressive (elastic + plastic) strain or vs. elongation. See AOPT above. All values are positive. See *DEFINE_CURVE.
SFAC	Scale factor on axial tensile force.

VARIABLE	DESCRIPTION
ALPHA	Parameter to define yield surface.
BETA	Parameter to define yield surface.
GAMMA	Parameter to define yield surface.
DELTA	Parameter to define yield surface.
A	Parameter to define yield surface.
B	Parameter to define yield surface.
FOFFS	Force offset for yield surface (see Remarks).
SIGY	Yield stress of material.
D	Depth of section used to calculate interaction curve.
W	Width of section used to calculate interaction curve.
TF	Flange thickness of section used to calculate interaction curve.
TW	Web thickness used to calculate interaction curve.
PR1-PR4	Plastic rotation thresholds 1 to 4
TS1-TS4	Tensile axial strain thresholds 1 to 4
CS1-CS4	Compressive axial strain thresholds 1 to 4

Remarks:

Yield surface for formulation type 1 is of the form:

$$\psi = \left(\frac{M_s}{M_{ys}}\right)^{\alpha} + \left(\frac{M_t}{M_{yt}}\right)^{\beta} + A\left(\frac{F}{F_y}\right)^{\gamma} + B\left(\frac{F}{F_y}\right)^{\delta} - 1$$

$$\psi = (M_s/M_{ys})^{\alpha} + (M_t/M_{yt})^{\beta} + A(F/F_y)^{\gamma} + B(F/F_y)^{\delta} - 1$$

where: M_s, M_t, F are the current moments about local s and t axes and axial force respectively

M_{ys}, M_{yt}, F_y are the current yield moments and forces; F_y is taken from LCAC or LCAT as appropriate.

α, β, γ, δ, A, B are input parameters. α, β, γ, δ must be greater than or equal to 1.1; non-integer values are now allowed.

If α, β, γ, δ, A and B are all set to zero then the following default values are used:

ALPHA	=	2.0
BETA	=	2.0
GAMMA	=	2.0
DELTA	=	4.0
A	=	2.0
B	=	-1.0

FOFFS offsets the yield surface parallel to the axial force axis. It is the compressive axial force at which the maximum bending moment capacity about the local s-axis (determined by LCPMS and SFS), and that about the local t-axis (determined by LCPMT and SFT), occur. For steel beams and columns, the value of FOFFS is usually zero. For reinforce concrete beams, columns and shear walls, the maximum bending moment capacity occurs corresponding to a certain compressive axial force, FOFFS. The value of FOFFS can be input as either positive or negative. Internally, LS-DYNA converts FOFFS to, and regards compressive axial force as, negative.

The option for degrading moment behavior changes the meaning of the plastic moment-rotation curve as follows:

If DEGRAD=0, the x-axis points on the curve represent current plastic rotation (i.e. total rotation minus the elastic component of rotation). This quantity can be positive or negative depending on the direction of rotation; during hysteresis the behavior will repeatedly follow backwards and forwards along the same curve. The curve should include negative and positive rotation and moment values. This option is retained so that results from existing models will be unchanged.

If DEGRAD=1, the x-axis points represent cumulative absolute plastic rotation. This quantity is always positive, and increases whenever there is plastic rotation in either direction. Thus, during hysteresis, the yield moments are taken from points in the input curve with increasingly positive rotation. If the curve shows a degrading behavior (reducing moment with rotation), then, once degraded by plastic rotation, the yield moment can never recover to its initial value. This option can be thought of as having “fatigue-type” hysteretic damage behavior, where all plastic cycles contribute to the total damage.

If DEGRAD=2, the x-axis points represent the high-tide value (always positive) of the plastic rotation. This quantity increases only when the absolute value of plastic rotation exceeds the previously recorded maximum. If smaller cycles follow a larger cycle, the plastic moment during the small cycles will be constant, since the high-tide plastic rotation is not altered by the small cycles. Degrading moment-rotation behavior is possible. This option can be thought of as showing rotation-controlled damage, and follows the FEMA approach for treating fracturing joints.

DEGRAD applies also to the axial behavior. The same options are available as for rotation: DEGRAD=0 gives unchanged behavior from previous versions; DEGRAD=1 gives a fatigue-type behavior using cumulative plastic strain; and DEGRAD=2 gives FEMA-type behavior, where the axial load capacity depends on the high-tide tensile and compressive strains. The definition of strain for this purpose is according to AOPT on Card 1 – it is expected that

AOPT=2 will be used with DEGRAD=2. The "axial strain" variable plotted by post-processors is the variable defined by AOPT.

The output variables plotted as "plastic rotation" have special meanings for this material model as follows – note that hinges form only at Node 2:

"Plastic rotation at End 1" is really a high-tide mark of absolute plastic rotation at Node 2, defined as follows:

1. Current plastic rotation is the total rotation minus the elastic component of rotation.
2. Take the absolute value of the current plastic rotation, and record the maximum achieved up to the current time. This is the high-tide mark of plastic rotation.

If DEGRAD=0, "Plastic rotation at End 2" is the current plastic rotation at Node 2.

If DEGRAD=1 or 2, "Plastic rotation at End 2" is the current total rotation at Node 2.

The total rotation is a more intuitively understood parameter, e.g. for plotting hysteresis loops. However, with DEGRAD=0, the previous meaning of that output variable has been retained such that results from existing models are unchanged.

FEMA thresholds are the plastic rotations at which the element is deemed to have passed from one category to the next, e.g. "Elastic", "Immediate Occupancy", "Life Safe", etc. The high-tide plastic rotation (maximum of Y and Z) is checked against the user-defined limits FEMA1, FEMA2, etc. The output flag is then set to 0, 1, 2, 3, or 4: 0 means that the rotation is less than FEMA1; 1 means that the rotation is between FEMA1 and FEMA2, and so on. By contouring this flag, it is possible to see quickly which joints have passed critical thresholds.

For this material model, special output parameters are written to the d3plot and d3thdt files. The number of output parameters for beam elements is automatically increased to 20 (in addition to the six standard resultants) when parts of this material type are present. Some post-processors may interpret this data as if the elements were integrated beams with 4 integration points. Depending on the post-processor used, the data may be accessed as follows:

Extra variable 16 (or Integration point 4 Axial Stress):	FEMA rotation flag
Extra variable 17 (or Integration point 4 XY Shear Stress):	Current utilization
Extra variable 18 (or Integration point 4 ZX Shear Stress):	Maximum utilization to date
Extra variable 20 (or Integration point 4 Axial Strain):	FEMA axial flag

"Utilization" is the yield parameter, where 1.0 is on the yield surface.

***MAT_SOIL_BRICK**

Purpose: This is Material Type 192. It is intended for modeling over-consolidated clay.

Card 1

Variable	MID	RO	RLAMDA	RKAPPA	RIOTA	RBETA1	RBETA2	RMU
Type	A8	F	F	F	F	F	F	F
Default								1.0

Card 2

Variable	RNU	RLCID	TOL	PGCL	SUB-INC	BLK	GRAV	THEORY
Type	F	F	F	F	F	F	F	I
Default			0.0005				9.807	0

Define Card 3 only if THEORY>0

Card 3

Variable	RVHHH	XSICRIT	ALPHA	RVH	RNU21	ANISO_4		
Type	F	F	F	F	F	F		
Default	0	0	0	0	0	0		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
RLAMDA	Material coefficient

VARIABLE	DESCRIPTION
RKAPPA	Material coefficient
RIOTA	Material coefficient
RBETA1	Material coefficient
RBETA2	Material coefficient
RMU	Shape factor coefficient. This parameter will modify the shape of the yield surface used. 1.0 implies a von Mises type surface, but 1.1 to 1.25 is more indicative of soils. The default value is 1.0.
RNU	Poisson's ratio
RLCID	Load curve identification number referring to a curve defining up to 10 pairs of 'string-length' vs G/Gmax points.
TOL	User defined tolerance for convergence checking. Default value is set to 0.02.
PGCL	Pre-consolidation ground level. This parameter defines the maximum surface level (relative to z = 0.0 in the model) of the soil throughout geological history. This is used calculate the maximum over burden pressure on the soil elements.
SUB-INC	User defined strain increment size. This is the maximum strain increment that the material model can normally cope with. If the value is exceeded a warning is echoed to the d3hsp file.
BLK	The elastic bulk stiffness of the soil. This is used for the contact stiffness only.
GRAV	The gravitational acceleration. This is used to calculate the element stresses due the overlying soil. Default is set to 9.807 m/s^2.
THEORY	Version of material subroutines used (See Remarks). EQ. 0: 1995 version, vectorized (Default) EQ. 4: 2003 version, unvectorized
RVHHH	Anisotropy ratio Gvh / Ghh (default = Isotropic behavior)
XSICRIT	Anisotropy parameter
ALPHA	Anisotropy parameter
RVH	Anisotropy ratio Ev / Eh
RNU21	Anisotropy ratio ν_2 / ν_1
ANISO_4	Anisotropy parameter

Remarks:

1. This material type requires that the model is oriented such that the z-axis is defined in the upward direction. Compressive initial stress must be defined, e.g. using *INITIAL_STRESS_SOLID or *INITIAL_STRESS_DEPTH.
The recommended unit system is kN, meters, seconds, tonnes. There are some built-in defaults that assume stress units of KN/m^2.

   Over-consolidated clays have suffered previous loading to higher stress levels than are present at the start of the analysis. This could have occurred due to ice sheets during previous ice ages, or the presence of soil or rock that has subsequently been eroded. The maximum vertical stress during that time is assumed to be:

$$\sigma_{VMAX} = RO*GRAV*(PGCL - Z_{el})$$

   where

   RO, GRAV, PGCL = input parameters
   Z_{el} = z-coordinate of center of element

   Since that time, the material has been unloaded until the vertical stress equals the user-defined initial vertical stress. The previous load/unload history has a significant effect on subsequent behavior, e.g. the horizontal stress in an over-consolidated clay may be greater than the vertical stress.

   This material model creates a load/unload cycle for a sample element of each material of this type, stores in a scratch file the horizontal stress and history variables as a function of the vertical stress, and interpolates these quantities from the defined initial vertical stress for each element. Therefore the initial horizontal stress seen in the output files will be different from the input initial horizontal stress.

   This material model is developed for a Geotechnical FE program (Oasys Ltd.'s SAFE) written by Arup. The default THEORY=0 gives a vectorized version ported from SAFE in the 1990's. Since then the material model has been developed further in SAFE; the most recent porting is accessed using THEORY=4 (recommended); however, this version is not vectorized and will run more slowly on most computer platforms.

2. The shape factor for a typical soil would be 1.25. Do not use values higher than 1.35.

*MAT_DRUCKER_PRAGER

Purpose: This is Material Type 193. This material enables soil to be modeled effectively. The parameters used to define the yield surface are familiar geotechnical parameters (i.e. angle of friction). The modified Drucker-Prager yield surface is used in this material model enabling the shape of the surface to be distorted into a more realistic definition for soils.

Card 1

Variable	MID	RO	GMOD	RNU	RKF	PHI	CVAL	PSI
Type	A8	F	F	F	F	F	F	F
Default					1.0			0.0

Card 2

Variable	STR_LIM							
Type	F							
Default	0.005							

Card 3

Variable	GMODDP	PHIDP	CVALDP	PSIDP	GMODGR	PHIGR	CVALGR	PSIGR
Type	F	F	F	F	F	F	F	F
Default	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density

VARIABLE	DESCRIPTION
GMOD	Elastic shear modulus
RNU	Poisson's ratio
RKF	Failure surface shape parameter
PHI	Angle of friction (radians)
CVAL	Cohesion value
PSI	Dilation angle (radians)
STR_LIM	Minimum shear strength of material is given by STR_LIM*CVAL
GMODDP	Depth at which shear modulus (GMOD) is correct
PHIDP	Depth at which angle of friction (PHI) is correct
CVALDP	Depth at which cohesion value (CVAL) is correct
PSIDP	Depth at which dilation angle (PSI) is correct
GMODGR	Gradient at which shear modulus (GMOD) increases with depth
PHIGR	Gradient at which friction angle (PHI) increases with depth
CVALGR	Gradient at which cohesion value (CVAL) increases with depth
PSIGR	Gradient at which dilation angle (PSI) increases with depth

Remarks:

1. This material type requires that the model is oriented such that the z-axis is defined in the upward direction. The key parameters are defined such that may vary with depth (i.e. the z-axis).

2. The shape factor for a typical soil would be 0.8, but should not be pushed further than 0.75.

3. If STR_LIM is set to less than 0.005, the value is reset to 0.005.

*MAT_RC_SHEAR_WALL

Purpose: This is Material Type 194. It is for shell elements only. It uses empirically-derived algorithms to model the effect of cyclic shear loading on reinforced concrete walls. It is primarily intended for modeling squat shear walls, but can also be used for slabs. Because the combined effect of concrete and reinforcement is included in the empirical data, crude meshes can be used. The model has been designed such that the minimum amount of input is needed: generally, only the variables on the first card need to be defined.

Card 1

Variable	MID	RO	E	PR	TMAX			I
Type	A8	F	F	F	F			
Default	none	none	none	0.0	0.0			

Define the following data if "Uniform Building Code" formula for maximum shear strength or tensile cracking are required – otherwise leave blank.

Card 2

Variable	FC	PREF	FYIELD	SIG0	UNCONV	ALPHA	FT	ERIENF
Type	F	F	F	F	F	F	F	F
Default	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0

Card 3

Variable	A	B	C	D	E	F		
Type	F	F	F	F	F	F		
Default	0.05	0.55	0.125	0.66	0.25	1.0		

Card 4	1	2	3	4	5	6	7	8
Variable	Y1	Y2	Y3	Y4	Y5			
Type	F	F	F	F	F			
Default	0.0	0.0	0.0	0.0	0.0			

Card 5

Variable	T1	T2	T3	T4	T5			
Type	F	F	F	F	F			
Default	0.0	0.0	0.0	0.0	0.0			

Card 6

Variable	AOPT							
Type	F							
Default	0.0							

Card 7

Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		
Default	0.0	0.0	0.0	0.0	0.0	0.0		

Card 8	1	2	3	4	5	6	7	8
Variable	V1	V2	V3	D1	D2	D3	BETA	
Type	F	F	F	F	F	F	F	
Default	0.0	0.0	0.0	0.0	0.0	0.0	0.0	

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
E	Young's Modulus
PR	Poisson's Ratio
TMAX	Ultimate in-plane shear stress. If set to zero, LS-DYNA will calculate TMAX based on the formulae in the Universal Building Code, using the data on card 2. See Remarks.
FC	Unconfined Compressive Strength of concrete (used in the calculation of ultimate shear stress; crushing behavior is not modeled)
PREF	Percent reinforcement, e.g. if 1.2% reinforcement, enter 1.2
FYIELD	Yield stress of reinforcement
SIG0	Overburden stress (in-plane compressive stress) - used in the calculation of ultimate shear stress. Usually sig0 is left as zero.
UCONV	Unit conversion factor. UCONV = √(1.0 psi in the model stress units). This is needed because the ultimate tensile stress of concrete is expressed as √(FC) where FC is in psi. Therefore a unit conversion factor of √(psi/stress unit) is required. Examples: UCONV = 0.083 if stress unit is MN/m2 or N/mm2 UCONV = 83.3 if stress unit is N/m2
ALPHA	Shear span factor - see below.
FT	Cracking stress in direct tension - see notes below. Default is 8% of the cylinder strength.

VARIABLE	DESCRIPTION
ERIENF	Young's Modulus of reinforcement. Used in calculation of post-cracked stiffness - see notes below.
A	Hysteresis constants determining the shape of the hysteresis loops.
B	Hysteresis constants determining the shape of the hysteresis loops.
C	Hysteresis constants determining the shape of the hysteresis loops.
D	Hysteresis constants determining the shape of the hysteresis loops.
E	Hysteresis constants determining the shape of the hysteresis loops.
F	Strength degradation factor. After the ultimate shear stress has been achieved, F multiplies the maximum shear stress from the curve for subsequent reloading. F=1.0 implies no strength degradation (default). F=0.5 implies that the strength is halved for subsequent reloading.
Y1,Y2...Y5	Shear strain points on stress-strain curve. By default these are calculated from the values on card 1. See below for more guidance.
T1,T2...T5	Shear stress points on stress-strain curve. By default these are calculated from the values on card 1. See below for more guidance.
AOPT	Material axes option: EQ. 0.0: locally orthotropic with material axes determined by element nodes as shown in Figure 2.1. Nodes 1, 2, and 4 of an element are identical to the Nodes used for the definition of a coordinate system as by *DEFINE_COORDINATE_NODES. EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_ VECTOR. EQ. 3.0: applicable to shell elements only. This option determines locally orthotropic material axes by offsetting the material axes by an angle to be specified from a line in the plane of the shell determined by taking the cross product of the vector **v** defined below with the shell normal vector. LT.0.0: the absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_NODES, *DEFINE_ COORDINATE_SYSTEM or *DEFINE_COORDINATE_ VECTOR). Available in R3 version of 971 and later.
XP,YP,ZP	Coordinates of point **p** for AOPT = 1.
A1,A2,A3	Components of vector **a** for AOPT = 2.
V1,V2,V3	Components of vector **v** for AOPT = 3.

VARIABLE	DESCRIPTION
D1,D2,D3	Components of vector **d** for AOPT = 2.
BETA	Material angle in degrees for AOPT = 3, may be overridden on the element card, see *ELEMENT_SHELL_BETA or *ELEMENT_SOLID_ORTHO.

Remarks:

The element is linear elastic except for in-plane shear and tensile cracking effects. Crushing due to direct compressive stresses are modeled only insofar as there is an in-plane shear stress component. It is not recommended that this model be used where nonlinear response to direct compressive or loads is important.

Note that the in-plane shear stress is defined as the shear stress in the element's local x-y plane (txy). This is not necessarily equal to the maximum shear stress in the plane: for example, if the principal stresses are at 45 degrees to the local axes, txy is zero. Therefore it is important to ensure that the local axes are appropriate - for a shear wall the local axes should be vertical or horizontal. By default, local X points from node 1 to node 2 of the element. It is possible to change the local axes by using AOPT>0.

If TMAX is set to zero, the ultimate shear stress is calculated using a formula in the Uniform Building Code 1997, section 1921.6.5:

$$\mathbf{TMAX_{UBC} = UCONV*ALPHA*\sqrt{(FC)} + Ro*FY}$$

where,

uconv	= unit conversion factor, 0.083 for SI units (MN)
Alpha	= aspect ratio, = 2.0 unless ratio h/l < 2.0 in which case alpha varies linearly from 2.0 at h/l=2.0 to 3.0 at h/l=1.5.
FC	= unconfined compressive strength of concrete
Ro	= fraction of reinforcement = percent reinforcement/100
FY	= yield stress of reinforcement

To this we add shear stress due to the overburden to obtain the ultimate shear stress:

$$\mathbf{TMAX_{UBC} = TMAX_{UBC} + SIG0}$$

where

SIG0 = in-plane compressive stress under static equilibrium conditions

The UBC formula for ultimate shear stress is generally conservative (predicts that the wall is weaker than shown in test), sometimes by 50% or more. A less conservative formula is that of Fukuzawa:

$$\mathbf{TMAX = a1*2.7*(1.9-M/LV)*UCONV*\sqrt{(FC)} + Ro*FY*0.5 + SIG0}$$

where

a1	= max((0.4 + Ac/Aw),1.0)
Ac	= Cross-sectional area of stiffening features such as columns or flanges
Aw	= Cross-sectional area of wall

M/LV = Aspect ratio of wall (height/length)

Other terms are as above. This formula is not included in the material model: TMAX should be calculated by hand and entered on Card 1 if the Fukuzawa formula is required.

It should be noted that none of the available formulae, including Fukuzawa, predict the ultimate shear stress accurately for all situations. Variance from the experimental results can be as great as 50%.

The shear stress vs shear strain curve is then constructed automatically as follows, using the algorithm of Fukuzawa extended by Arup:

Assume ultimate shear strain, $\gamma_u = 0.0048$

First point on curve (concrete cracking) at (0.3TMAX/G, 0.3TMAX) where G is the elastic shear modulus given by E/2(1+v)

Second point (reinforcement yield) at $(0.5\gamma_u, 0.8TMAX)$

Third point (ultimate strength) at $(\gamma_u, TMAX)$

Fourth point (onset of strength reduction) at $(2\gamma_u, TMAX)$

Fifth point (failure) at $(3\gamma_u, 0.6TMAX)$.

After failure, the shear stress drops to zero. The curve points can be entered by the user if desired, in which case they over-ride the automatically calculated curve. However, it is anticipated that in most cases the default curve will be preferred due to ease of input.

Hysteresis follows the algorithm of Shiga as for the squat shear wall spring (see *MAT_SPRING_SQUAT_SHEARWALL). The hysteresis constants A,B,C,D,E can be entered by the user if desired but it is generally recommended that the default values be used.

Cracking in tension is checked for the local x and y directions only – this is calculated separately from the in-plane shear. A trilinear response is assumed, with turning points at concrete cracking and reinforcement yielding. The three regimes are:

1. Pre-cracking, linear elastic response is assumed using the overall Young's Modulus on Card 1.

2. **Cracking occurs in the local x or y directions when the tensile stress in that direction exceeds the concrete tensile strength FT** (if not input on Card 2, this defaults to 8% of the compressive strength FC). Post-cracking, a linear stress-strain response is assumed up to reinforcement yield at a strain defined by reinforcement yield stress divided by reinforcement Young's Modulus.

3. Post-yield, a constant stress is assumed (no work hardening).

   Unloading returns to the origin of the stress-strain curve.

   For compressive strains the response is always linear elastic using the overall Young's Modulus on Card 1.

If insufficient data is entered, no cracking occurs in the model. As a minimum, FC and FY are needed.

Extra variables are available for post-processing as follows:

Extra variable 1: Current shear strain
Extra variable 2: Shear status: 0,1,2,3,4 or 5– see below
Extra variable 3: Maximum direct strain so far in local X direction (for tensile cracking)
Extra variable 4: Maximum direct strain so far in local Y direction (for tensile cracking)
Extra variable 5: Tensile status: 0,1 or 2 = elastic, cracked, or yielded respectively.

The shear status shows how far along the shear stress-strain curve each element has progressed, e.g. status 2 means that the element has passed the second point on the curve. These status levels correspond to performance criteria in building design codes such as FEMA.

*MAT_CONCRETE_BEAM

This is Material Type 195 for beam elements. An elasto-plastic material with an arbitrary stress versus strain curve and arbitrary strain rate dependency can be defined. See also Remark below. Also, failure based on a plastic strain or a minimum time step size can be defined.

Card 1

Variable	MID	RO	E	PR	SIGY	ETAN	FAIL	TDEL
Type	A8	F	F	F	F	F	F	F
Default	none	None	none	none	none	0.0	10.E+20	10.E+20

Card 2

Variable	C	P	LCSS	LCSR				
Type	F	F	F	F				
Default	0	0	0	0				

Card 3

Variable	NOTEN	TENCUT	SDR					
Type	I	F	F					
Default	0	E15.0	0.0					

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
E	Young's modulus.

VARIABLE	DESCRIPTION
PR	Poisson's ratio.
SIGY	Yield stress.
ETAN	Tangent modulus, ignored if (LCSS.GT.0) is defined.
FAIL	Failure flag. LT.0.0: user defined failure subroutine is called to determine failure EQ.0.0: failure is not considered. This option is recommended if failure is not of interest since many calculations will be saved. GT.0.0: plastic strain to failure. When the plastic strain reaches this value, the element is deleted from the calculation.
TDEL	Minimum time step size for automatic element deletion.
C	Strain rate parameter, C, see formula below.
P	Strain rate parameter, P, see formula below.
LCSS	Load curve ID or Table ID. Load curve ID defining effective stress versus effective plastic strain. If defined EPS1-EPS8 and ES1-ES8 are ignored. The table ID defines for each strain rate value a load curve ID giving the stress versus effective plastic strain for that rate, See Figure 16.1. The stress versus effective plastic strain curve for the lowest value of strain rate is used if the strain rate falls below the minimum value. Likewise, the stress versus effective plastic strain curve for the highest value of strain rate is used if the strain rate exceeds the maximum value. The strain rate parameters: C and P;
LCSR	Load curve ID defining strain rate scaling effect on yield stress.
NOTEN	No-tension flag, EQ.0: beam takes tension, EQ.1: beam takes no tension, EQ.2: beam takes tension up to value given by TENCUT.
TENCUT	Tension cutoff value.
SDR	Stiffness degradation factor.

Remarks:

The stress strain behavior may be treated by a bilinear stress strain curve by defining the tangent modulus, ETAN. An effective stress versus effective plastic strain curve (LCSS) may be

input instead of defining ETAN. The cost is roughly the same for either approach. The most general approach is to use the table definition (LCSS) discussed below.

Three options to account for strain rate effects are possible.

I. Strain rate may be accounted for using the Cowper and Symonds model which scales the yield stress with the factor

$$1+\left(\frac{\dot{\varepsilon}}{C}\right)^{1/p}$$

where $\dot{\varepsilon}$ is the strain rate. $\dot{\varepsilon}=\sqrt{\dot{\varepsilon}_{ij}\dot{\varepsilon}_{ij}}$.

II. For complete generality a load curve (LCSR) to scale the yield stress may be input instead. In this curve the scale factor versus strain rate is defined.

III. If different stress versus strain curves can be provided for various strain rates, the option using the reference to a table (LCSS) can be used.

*MAT_GENERAL_SPRING_DISCRETE_BEAM

This is Material Type 196. This model permits elastic and elastoplastic springs with damping to be represented with a discrete beam element type6 by using six springs each acting about one of the six local degrees-of-freedom. For elastic behavior, a load curve defines force or moment versus displacement or rotation. For inelastic behavior, a load curve yield force or moment versus plastic deflection or rotation, which can vary in tension and compression. The two nodes defining a beam may be coincident to give a zero length beam, or offset to give a finite length beam. For finite length discrete beams the absolute value of the variable SCOOR in the SECTION_BEAM input should be set to a value of 2.0, which causes the local r-axis to be aligned along the two nodes of the beam to give physically correct behavior. The distance between the nodes of a beam should not affect the behavior of this material model. A triad is used to orient the beam for the directional springs.

Card 1

Variable	MID	RO						
Type	A8	F						

Define the following cards, 2 and 3, for each active degree of freedom. This data is terminated by the next "*" card or when all six degrees-of-freedom are defined.

Card 2

Variable	DOF	TYPE	K	D	CDF	TDF		
Type	I	I	F	F	F	F		

Card 3

Variable	FLCID	HLCID	C1	C2	DLE	GLCID		
Type	F	F	F	F	F	I		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density, see also volume in *SECTION_BEAM definition.
DOF	Active degree-of-freedom, a number between 1 and 6 inclusive. Each value of DOF can only be used once. The active degree-of-freedom is measured in the local coordinate system for the discrete beam element.
TYPE	The default behavior is elastic. For inelastic behavior input 1.
K	Elastic loading/unloading stiffness. This is required input for inelastic behavior.
D	Optional viscous damping coefficient.
CDF	Compressive displacement at failure. Input as a positive number. After failure, no forces are carried. This option does not apply to zero length springs. EQ.0.0: inactive.
TDF	Tensile displacement at failure. After failure, no forces are carried. EQ.0.0: inactive.
FLCID	Load curve ID, see *DEFINE_CURVE. For option TYPE=0, this curve defines force or moment versus deflection for nonlinear elastic behavior. For option TYPE=1, this curve defines the yield force versus plastic deflection. If the origin of the curve is at (0,0) the force magnitude is identical in tension and compression, i.e., only the sign changes. If not, the yield stress in the compression is used when the spring force is negative. The plastic displacement increases monotonically in this implementation. The load curve is required input.
HLCID	Load curve ID, see *DEFINE_CURVE, defining force versus relative velocity (Optional). If the origin of the curve is at (0,0) the force magnitude is identical for a given magnitude of the relative velocity, i.e., only the sign changes.
C1	Damping coefficient.
C2	Damping coefficient
DLE	Factor to scale time units.
GLCID	Optional load curve ID, see *DEFINE_CURVE, defining a scale factor versus deflection for load curve ID, HLCID. If zero, a scale factor of unity is assumed.

Remarks:

If TYPE=0, elastic behavior is obtained. In this case, if the linear spring stiffness is used, the force, *F*, is given by:

$$F = F_0 + K\Delta L + D\Delta\dot{L}$$

but if the load curve ID is specified, the force is then given by:

$$F = F_0 + K\ f\left(\Delta L\right)\left[1 + C1\cdot\Delta\dot{L} + C2\cdot\mathrm{sgn}\left(\Delta\dot{L}\right)\ln\left(\max\left\{1., \frac{\left|\Delta\dot{L}\right|}{DLE}\right\}\right)\right] + D\Delta\dot{L} + g\left(\Delta L\right)h\left(\Delta\dot{L}\right)$$

In these equations, ΔL is the change in length

$$\Delta L = current\ length - initial\ length$$

If TYPE=1, inelastic behavior is obtained. In this case, the yield force is taken from the load curve:

$$F^Y = F_y\left(\Delta L^{plastic}\right)$$

where $L^{plastic}$ is the plastic deflection. A trial force is computed as:

$$F^T = F^n + K\Delta\dot{L}\left(\Delta t\right)$$

and is checked against the yield force to determine F:

$$F = \begin{cases} F^Y\ if\ F^T > F^Y \\ F^T\ if\ F^T \le F^Y \end{cases}$$

The final force, which includes rate effects and damping, is given by:

$$F^{n+1} = F\cdot\left[1 + C1\cdot\Delta\dot{L} + C2\cdot\mathrm{sgn}\left(\Delta\dot{L}\right)\ln\left(\max\left\{1., \frac{\left|\Delta\dot{L}\right|}{DLE}\right\}\right)\right] + D\Delta\dot{L} + g\left(\Delta L\right)h\left(\Delta\dot{L}\right)$$

Unless the origin of the curve starts at (0,0), the negative part of the curve is used when the spring force is negative where the negative of the plastic displacement is used to interpolate, F_y. The positive part of the curve is used whenever the force is positive.

The cross sectional area is defined on the section card for the discrete beam elements, See *SECTION_BEAM. The square root of this area is used as the contact thickness offset if these elements are included in the contact treatment.

*MAT_SEISMIC_ISOLATOR

This is Material Type 197 for discrete beam elements. Sliding and elastomeric seismic isolation bearings can be modeled, applying bi-directional coupled plasticity theory. The hysteretic behavior was proposed by Wen [1976] and Park, Wen, and Ang [1986]. The sliding bearing behavior is recommended by Zayas, Low and Mahin [1990]. The algorithm used for implementation was presented by Nagarajaiah, Reinhorn, and Constantinou [1991]. Further options for tension-carrying friction bearings are as recommended by Roussis and Constantinou [2006]. Element formulation type 6 must be used. Local axes are defined on *SECTION_BEAM; the default is the global axis system. It is expected that the local z-axis will be vertical.

(Note: Option **ITYPE=2** is available starting with the R3 release of Version 971.)

Card 1

Variable	MID	RO	A	GAMMA	BETA	DISPY	STIFFV	ITYPE
Type	A8	F	F	F	F	F	F	I
Default	none	None	1.0	0.5	0.5	0.0	0.0	0.0

Card 2

Variable	PRELOAD	DAMP	MX1	MX2	MY1	MY2		
Type	F	F	F	F	F	F		
Default	0	1.0	0	0	0	0		

Card 3 for sliding isolator, ITYPE = 0 or 2 - leave blank for elastomeric isolator:

Card 3	1	2	3	4	5	6	7	8
Variable	FMAX	DELF	AFRIC	RADX	RADY	RADB	STIFFL	STIFFTS
Type	F	F	F	F	F	F	F	F
Default	0	0	0	1.0e20	1.0e20	1.0e20	STIFFV	0

Card 4 for ITYPE = 1 or 2 - leave blank for sliding isolator ITYPE = 0:

Card 4	1	2	3	4	5	6	7	8
Variable	FORCEY	ALPHA	STIFFT	DFAIL	FMAXYC	FMAXXT	FMAXYT	YLOCK
Type	F	F	F	F	F	F	F	F
Default	0	0	0.5STIFFV	1.0e20	FMAX	FMAX	FMAX	0.0

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
A	Nondimensional variable - see below
GAMMA	Nondimensional variable - see below
BETA	Nondimensional variable - see below
DISPY	Yield displacement (length units - must be > 0.0)
STIFFV	Vertical stiffness (force/length units)
ITYPE	Type: 0=sliding (spherical or cylindrical) 1=elastomeric 2=sliding (two perpendicular curved beams)
PRELOAD	Vertical preload not explicitly modeled (force units)

VARIABLE	DESCRIPTION
DAMP	Damping ratio (nondimensional)
MX1, MX2	Moment factor at ends 1 and 2 in local X-direction
MY1, MY2	Moment factor at ends 1 and 2 in local Y-direction
FMAX (*)	Maximum friction coefficient (dynamic)
DELF (*)	Difference between maximum friction and static friction coefficient
AFRIC (*)	Velocity multiplier in sliding friction equation (time/length units)
RADX (*)	Radius for sliding in local X direction
RADY (*)	Radius for sliding in local Y direction
RADB (*)	Radius of retaining ring
STIFFL (*)	Stiffness for lateral contact against the retaining ring
STIFFTS (*)	Stiffness for tensile vertical response (sliding isolator - default = 0)
FORCEY (+)	Yield force
ALPHA (+)	Ratio of postyielding stiffness to preyielding stiffness
STIFFT (+)	Stiffness for tensile vertical response (elastomeric isolator)
DFAIL (+)	Lateral displacement at which the isolator fails
FMAXYC (**)	Max friction coefficient (dynamic) for local Y-axis (compression)
FMAXXT (**)	Max friction coefficient (dynamic) for local X-axis (tension)
FMAXYT (**)	Max friction coefficient (dynamic) for local Y-axis (tension)
YLOCK (**)	Stiffness locking the local Y-displacement (optional -single-axis sliding)

(*) - Used for sliding type. Leave blank for elastomeric type

(+) - Used for elastomeric type. Leave blank for sliding type

(**) - Used for ITYPE=2. Leave blank for ITYPE=0 or 1

Remarks:

The horizontal behavior of both types is governed by plastic history variables Zx, Zy that evolve according to equations given in the reference; A, gamma and beta and the yield displacement are the input parameters for this. The intention is to provide smooth build-up, rotation and reversal

of forces in response to bidirectional displacement histories in the horizontal plane. The theoretical model has been correlated to experiments on seismic isolators.

The RADX, RADY inputs for the sliding isolator are optional. If left blank, the sliding surface is assumed to be flat. A cylindrical surface is obtained by defining either RADX or RADY; a spherical surface can be defined by setting RADX=RADY. The effect of the curved surface is to add a restoring force proportional to the horizontal displacement from the center. As seen in elevation, the top of the isolator will follow a curved trajectory, lifting as it displaces away from the center.

The vertical behavior for all types is linear elastic, but with different stiffnesses for tension and compression. By default, the tensile stiffness is zero for the sliding types.

The vertical behavior for the elastomeric type is linear elastic; in the case of uplift, the tensile stiffness will be different to the compressive stiffness. For the sliding type, compression is treated as linear elastic but no tension can be carried.

Vertical preload can be modeled either explicitly (for example, by defining gravity), or by using the PRELOAD input. PRELOAD does not lead to any application of vertical force to the model. It is added to the compression in the element before calculating the friction force and tensile/compressive vertical behavior.

ITYPE=2 is intended to model uplift-prevention sliding isolators that consist of two perpedicular curved beams joined by a connector that can slide in slots on both beams. The beams are aligned in the local X and Y axes respectively. The vertical displacement is the sum of the displacements induced by the respective curvatures and slider displacements along the two beams. Single-axis sliding is obtained by using YLOCK to lock the local-Y displacement. To resist uplift, STIFFTS must be defined (recommended value: same as STIFFV). This isolator type allows different friction coefficients on each beam, and different values in tension and compression. The total friction, taking into account sliding velocity and the friction history functions, is first calculated using FMAX and then scaled by FMAXXT/FMAX etc as appropriate. For this reason, FMAX should not be zero.

DAMP is the fraction of critical damping for free vertical vibration of the isolator, based on the mass of the isolator (including any attached lumped masses) and its vertical stiffness. The viscosity is reduced automatically if it would otherwise infringe numerical stability. Damping is generally recommended: oscillations in the vertical force would have a direct effect on friction forces in sliding isolators; for isolators with curved surfaces, vertical oscillations can be excited as the isolator slides up and down the curved surface. It may occasionally be necessary to increase DAMP if these oscillations become significant.

This element has no rotational stiffness - a pin joint is assumed. However, if required, moments can be generated according to the vertical load times the lateral displacement of the isolator. The moment about the local X-axis (i.e. the moment that is dependent on lateral displacement in the local Y-direction) is reacted on nodes 1 and 2 of the element in the proportions MX1 and MX2 respectively. Similarly, moments about the local Y-axis are reacted in the proportions MY1, MY2. These inputs effectively determine the location of the pin joint: for example, a pin at the base of the column could be modeled by setting MX1=MY1=1.0, MX2=MY2=0.0 and ensuring that node 1 is on the foundation, node 2 at the base of the column - then all the moment is

transferred to the foundation. For the same model, MX1=MY1=0.0, MX2=MY2=1.0 would imply a pin at the top of the foundation - all the moment is transferred to the column. Some isolator designs have the pin at the bottom for moments about one horizontal axis, and at the top for the other axis - these can be modeled by setting MX1=MY2=1.0, MX2=MY1=0.0. It is expected that all MX1,2, etc lie between 0 and 1, and that MX1+MX2=1.0 (or both can be zero) - e.g. MX1=MX2=0.5 is permitted - but no error checks are performed to ensure this; similarly for MY1+MY2.

Density should be set to a reasonable value, say 2000 to 8000 kg//m^3. The element mass will be calculated as density x volume (volume is entered on *SECTION_BEAM).

Note on values for *SECTION_BEAM:

- Set ELFORM to 6 (discrete beam)
- VOL (the element volume) might typically be set to 0.1m^3
- INER needs to be non-zero (say 1.0) but the value has no effect on the solution since the element has no rotational stiffness.
- CID can be left blank if the isolator is aligned in the global coordinate system, otherwise a coordinate system should be referenced.
- By default, the isolator will be assumed to rotate with the average rotation of its two nodes. If the base of the column rotates slightly the isolator will no longer be perfectly horizontal: this can cause unexpected vertical displacements coupled with the horizontal motion. To avoid this, rotation of the local axes of the isolator can be eliminated by setting RRCON, SRCON and TRCON to 1.0. This does not introduce any rotational restraint to the model, it only prevents the orientation of the isolator from changing as the model deforms.
- All other parameters on *SECTION_BEAM can be left blank.

Post-processing note: as with other discrete beam material models, the force described in post-processors as “Axial” is really the force in the local X-direction; “Y-Shear” is really the force in the local Y-direction; and “Z-Shear” is really the force in the local Z-direction.

*MAT_JOINTED_ROCK

This is Material Type 198. Joints (planes of weakness) are assumed to exist throughout the material at a spacing small enough to be considered ubiquitous. The planes are assumed to lie at constant orientations defined on this material card. Up to three planes can be defined for each material. The matrix behavior is modified Drucker Prager, as per material type 193.

Card 1

Variable	MID	RO	GMOD	RNU	RKF	PHI	CVAL	PSI
Type	A8	F	F	F	F	F	F	F
Default					1.0			0.0

Card 2

Variable	STR_LIM	NPLANES	ELASTIC	LCCPDR	LCCPT	LCCJDR	LCCJT	LCSFAC
Type	F	I	I	I	I	I	I	I
Default	0.005	0	0	0	0	0	0	0

Card 3

Variable	GMODDP	PHIDP	CVALDP	PSIDP	GMODGR	PHIGR	CVALGR	PSIGR
Type	F	F	F	F	F	F	F	F
Default	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0

Repeat Card 4 for each plane (maximum 3 planes):

Card 4	1	2	3	4	5	6	7	8
Variable	DIP	STRIKE	CPLANE	FRPLANE	TPLANE	SHRMAX	LOCAL	
Type	F	F	F	F	F	F	F	
Default	0.0	0.0	0.0	0.0	0.0	1.e20	0.0	

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
GMOD	Elastic shear modulus
RNU	Poisson's ratio
RKF	Failure surface shape parameter
PHI	Angle of friction (radians)
CVAL	Cohesion value
PSI	Dilation angle (radians)
STR_LIM	Minimum shear strength of material is given by STR_LIM*CVAL
NPLANES	Number of joint planes (maximum 3)
ELASTIC	Flag = 1 for elastic behavior only
LCCPDR	Load curve for extra cohesion for parent material (dynamic relaxation)
LCCPT	Load curve for extra cohesion for parent material (transient)
LCCJDR	Load curve for extra cohesion for joints (dynamic relaxation)
LCCJT	Load curve for extra cohesion for joints (transient)
LCSFAC	Load curve giving factor on strength vs time
GMODDP	Depth at which shear modulus (GMOD) is correct

VARIABLE	DESCRIPTION
PHIDP	Depth at which angle of friction (PHI) is correct
CVALDP	Depth at which cohesion value (CVAL) is correct
PSIDP	Depth at which dilation angle (PSI) is correct
GMODGR	Gradient at which shear modulus (GMOD) increases with depth
PHIGR	Gradient at which friction angle (PHI) increases with depth
CVALGR	Gradient at which cohesion value (CVAL) increases with depth
PSIGR	Gradient at which dilation angle (PSI) increases with depth
DIP	Angle of the plane in degrees below the horizontal
DIPANG	Plan view angle (degrees) of downhill vector drawn on the plane
CPLANE	Cohesion for shear behavior on plane
PHPLANE	Friction angle for shear behavior on plane (degrees)
TPLANE	Tensile strength across plane (generally zero or very small)
SHRMAX	Max shear stress on plane (upper limit, independent of compression)
LOCAL	EQ.0: DIP and DIPANG are with respect to the global axes EQ.1: DIP and DIPANG are with respect to the local element axe

Remarks:

1. The joint plane orientations are defined by the angle of a “downhill vector” drawn on the plane, i.e. the vector is oriented within the plane to obtain the maximum possible downhill angle. DIP is the angle of this line below the horizontal. DIPANG is the plan-view angle of the line (pointing down hill) measured clockwise from the global Y-axis about the global Z-axis.

2. The joint planes rotate with the rigid body motion of the elements, irrespective of whether their initial definitions are in the global or local axis system.

3. The full facilities of the modified Drucker Prager model for the matrix material can be used – see description of Material type 193. Alternatively, to speed up the calculation, the ELASTIC flag can be set to 1, in which case the yield surface will not be considered and only RO, GMOD, RNU, GMODDP, GMODGR and the joint planes will be used.

4. This material type requires that the model is oriented such that the z-axis is defined in the upward direction. The key parameters are defined such that may vary with depth (i.e. the z-axis)

5. The shape factor for a typical soil would be 0.8, but should not be pushed further than 0.75.

6. If STR_LIM is set to less than 0.005, the value is reset to 0.005.

7. A correction has been introduced into the Drucker Prager model, such that the yield surface never infringes the Mohr-Coulomb criterion. This means that the model does not give the same results as a "pure" Drucker Prager model.

8. The load curves LCCPDR, LCCPT, LCCJDR, LCCJT allow additional cohesion to be specified as a function of time. The cohesion is additional to that specified in the material parameters. This is intended for use during the initial stages of an analysis to allow application of gravity or other loads without cracking or yielding, and for the cracking or yielding then to be introduced in a controlled manner. This is done by specifying extra cohesion that exceeds the expected stresses initially, then declining to zero. If no curves are specified, no extra cohesion is applied.

9. The load curve for factor on strength applies simultaneously to the cohesion and tan (friction angle) of parent material and all joints. This feature is intended for reducing the strength of the material gradually, to explore factors of safety. If no curve is present, a constant factor of 1 is assumed. Values much greater than 1.0 may cause problems with stability.

10. Extra variables for plotting. By setting NEIPH on *DATABASE_EXTENT_BINARY to 15, the following variables can be plotted in D3PLOT and T/HIS:

    Extra Variable 1: Mobilized strength fraction for base material
    Extra Variable 2: rk0 for base material
    Extra Variable 3: rlamda for base material
    Extra Variable 4: crack opening strain for plane 1
    Extra Variable 5: crack opening strain for plane 2
    Extra Variable 6: crack opening strain for plane 3
    Extra Variable 7: crack accumulated shear strain for plane 1
    Extra Variable 8: crack accumulated shear strain for plane 2
    Extra Variable 9: crack accumulated shear strain for plane 3
    Extra Variable 10: current shear utilization for plane 1
    Extra Variable 11: current shear utilization for plane 2
    Extra Variable 12: current shear utilization for plane 3
    Extra Variable 13: maximum shear utilization to date for plane 1
    Extra Variable 14: maximum shear utilization to date for plane 2
    Extra Variable 15: maximum shear utilization to date for plane 3

14. Joint planes would generally be defined in the global axis system if they are taken from survey data. However, the material model can also be used to represent masonry, in which case the weak planes represent the cement and lie parallel to the local element axes.

*MAT_RIGID_DISCRETE

This is Material Type 220. This is a rigid material which is discretized into multiple disjoint pieces. Each rigid piece can contain an arbitrary number of nodal points and solid elements that are arranged in an arbitrary shape. Rigid body mechanics is used to update each disjoint piece of any part ID which references this material type. This material can be used to model granular material where the grains interact through an automatic single sureface contact definition. Another possible use includes modeling bolts as rigid bodies where the bolts have belong to the same part ID. This model eliminates the need to represent each rigid piece with a unique part ID.

Card	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR				
Type	A8	F	F	F				
Default	none	none	none	none				

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.

*MAT_ORTHOTROPIC_SIMPLIFIED_DAMAGE

This is Material Type 221. An orthotropic material with optional simplified damage and optional failure for composites can be defined. This model is valid only for 3D solid elements, with reduced or full integration. The elastic behavior is the same as MAT_022. Nine damage variables are defined, applicable to Ea, Eb, Ec, (damage is different in tension and compression), and Gab, Gbc and Gca. In addition, nine failure criteria on strains are available. When failure occurs, elements are deleted (erosion). Failure depends on the number of integration points failed through the element. See the material description below.

Card 1

	1	2	3	4	5	6	7	8
Variable	MID	RO	EA	EB	EC	PRBA	PRCA	PRCB
Type	A8	F	F	F	F	F	F	F
Default	none	None	none	none	none	none	none	none

Card 2

Variable	GAB	GBC	GCA		AOPT	MACF		
Type	F	F	F		F	I		
Default	none	None	none		0.0	0		

Card 3

Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		
Default	0.0	0.0	0.0	0.0	0.0	0.0		

Card 4

	1	2	3	4	5	6	7	8
Variable	V1	V2	V3	D1	D2	D3	BETA	
Type	F	F	F	F	F	F	F	
Default	0.0	0.0	0.0	0.0	0.0	0.0	0.0	

Card 5

Variable	NERODE	NDAM	EPS1TF	EPS2TF	EPS3TF	EPS1CF	EPS2CF	EPS3CF
Type	I	I	F	F	F	F	F	F
Default	0	0	1.E20	1.E20	1.E20	-1.E20	-1.E20	-1.E20

Card 6

Variable	EPS12F	EPS23F	EPS13F	EPSD1T	DPSC1T	CDAM1T	EPS2DT	EPSC2T
Type	F	F	F	F	F	F	F	F
Default	1.E20	1.E20	1.E20	0.	0.	0.	0.	0.

Card 7

Variable	CDAM2T	EPSD3T	EPSC3T	CDAM3T	EPSD1C	EPSC1C	CDAM1C	EPSD2C
Type	I	I	F	F	F	F	F	F
Default	0.	0.	0.	0.	0.	0.	0.	0.

Card 8	1	2	3	4	5	6	7	8
Variable	EPSC2C	CDAM2C	EPSD3C	EPSC3C	CDAM3C	EPSD12	EPSC12	CDAM12
Type	F	F	F	F	F	F	F	F
Default	0.	0.	0.	0.	0.	0.	0.	0.

Card 9

Variable	EPSD23	EPSC23	CDAM23	EPSD31	EPSC31	CDAM31		
Type	F	F	F	F	F	F		
Default	0.	0.	0.	0.	0.	0.		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
EA	E_a, Young's modulus in a-direction.
EB	E_b, Young's modulus in b-direction.
EC	E_c, Young's modulus in c-direction.
PRBA	ν_{ba}, Poisson ratio, ba.
PRCA	ν_{ca}, Poisson ratio, ca.
PRCB	ν_{cb}, Poisson ratio, cb.
GAB	G_{ab}, Shear modulus, ab.
GBC	G_{bc}, Shear modulus, bc.
GCA	G_{ca}, Shear modulus, ca.

VARIABLE	DESCRIPTION
AOPT	Material axes option (see MAT_*OPTION* TROPIC_ELASTIC for a more complete description): EQ. 0.0: locally orthotropic with material axes determined by element nodes 1, 2, and 4, as with *DEFINE_COORDINATE_ NODES. EQ. 1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center; this is the a-direction. This option is for solid elements only. EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_ VECTOR. EQ. 3.0: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle, BETA, from a line in the plane of the element defined by the cross product of the vector **v** with the element normal. EQ. 4.0: locally orthotropic in cylindrical coordinate system with the material axes determined by a vector **v**, and an originating point, P, which define the centerline axis. This option is for solid elements only. LT.0.0: the absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_NODES, *DEFINE_ COORDINATE_SYSTEM or *DEFINE_COORDINATE_ VECTOR).
MACF	Material axes change flag for brick elements: EQ.1: No change, default, EQ.2: switch material axes a and b, EQ.3: switch material axes a and c, EQ.4: switch material axes b and c.
XP,YP,ZP	Coordinates of point **p** for AOPT = 1.
A1,A2,A3	Components of vector **a** for AOPT = 2.
V1,V2,V3	Components of vector **v** for AOPT = 3.
D1,D2,D3	Components of vector **d** for AOPT = 2.
BETA	Material angle in degrees for AOPT = 3, may be overridden on the element card, see *ELEMENT_ SOLID_ORTHO.

VARIABLE	DESCRIPTION
NERODE	Failure and erosion flag: EQ. 0: No failure (default) EQ. 1: Failure as soon as one failure criterion is reached in all integration points EQ. 2: Failure as soon as one failure criterion is reached in at least one integration point EQ. 3: Failure as soon as a tension or compression failure criterion in the a-direction is reached for one integration point EQ. 4: Failure as soon as a tension or compression failure criterion in the b-direction is reached for one integration point EQ. 5: Failure as soon as a tension or compression failure criterion in the c-direction is reached for one integration point EQ. 6: Failure as soon as tension or compression failure criteria in both the a- and b-directions are reached at a single integration point or at 2 different integration points EQ. 7: Failure as soon as tension or compression failure criteria in both the b- and c-directions are reached at a single integration point or at 2 different integration points EQ. 8: Failure as soon as tension or compression failure criteria in both the a- and c-directions are reached at a single integration point or at 2 different integration points EQ. 9: Failure as soon as tension or compression failure criteria in the 3 directions are reached at a single integration point or at different integration points
NDAM	Damage flag: EQ. 0: No damage (default) EQ. 1: Damage in tension only (null for compression) EQ. 2: Damage in tension and compression
EPS1TF	Failure strain in tension along the a-direction
EPS2TF	Failure strain in tension along the b-direction
EPS3TF	Failure strain in tension along the c-direction
EPS1CF	Failure strain in compression along the a-direction
EPS2CF	Failure strain in compression along the b-direction
EPS3CF	Failure strain in compression along the c-direction
EPS12F	Failure shear strain in the ab-plane
EPS23F	Failure shear strain in the bc-plane
EPS13F	Failure shear strain in the ac-plane

VARIABLE	DESCRIPTION
EPSD1T	Damage threshold in tension along the a-direction, ε_{1t}^{s}
EPSC1T	Critical damage threshold in tension along the a-direction, ε_{1t}^{c}
CDAM1T	Critical damage in tension along the a-direction, D_{1t}^{c}
EPS2DT	Damage threshold in tension along the b-direction, ε_{2t}^{s}
EPSC2T	Critical damage threshold in tension along the b-direction, ε_{2t}^{c}
CDAM2T	Critical damage in tension along the b-direction, D_{2t}^{c}
EPSD3T	Damage threshold in tension along the c-direction, ε_{3t}^{s}
EPSC3T	Critical damage threshold in tension along the c-direction, ε_{3t}^{c}
CDAM3T	Critical damage in tension along the c-direction, D_{3t}^{c}
EPSD1C	Damage threshold in compression along the a-direction, ε_{1c}^{s}
EPSC1C	Critical damage threshold in compression along the a-direction, ε_{1c}^{c}
CDAM1C	Critical damage in compression along the a-direction, D_{1c}^{c}
EPSD2C	Damage threshold in compression along the b-direction, ε_{2c}^{s}
EPSC2C	Critical damage threshold in compression along the b-direction, ε_{2c}^{c}
CDAM2C	Critical damage in compression along the b-direction, D_{2c}^{c}
EPSD3C	Damage threshold in compression along the c-direction, ε_{3c}^{s}
EPSC3C	Critical damage threshold in compression along the c-direction, ε_{3c}^{c}
CDAM3C	Critical damage in compression along the c-direction, D_{3c}^{c}
EPSD12	Damage threshold for shear in the ab-plane, ε_{12}^{s}
EPSC12	Critical damage threshold for shear in the ab-plane, ε_{12}^{c}
CDAM12	Critical damage for shear in the ab-plane, D_{12}^{c}
EPSD23	Damage threshold for shear in the bc-plane, ε_{23}^{s}
EPSC23	Critical damage threshold for shear in the bc-plane, ε_{23}^{c}
CDAM23	Critical damage for shear in the bc-plane, D_{23}^{c}

VARIABLE	DESCRIPTION
EPSD31	Damage threshold for shear in the ac-plane, ε_{31}^{s}
EPSC31	Critical damage threshold for shear in the ac-plane, ε_{31}^{c}
CDAM31	Critical damage for shear in the ac-plane, D_{31}^{c}

If $\varepsilon_k^c < \varepsilon_k^s$, no damage is considered. Failure occurs only when failure strain is reached.

Failure can occur along the 3 orthotropic directions, in tension, in compression and for shear behavior. Nine failure strains drive the failure. When failure occurs, elements are deleted (erosion). Under the control of the NERODE flag, failure may occur either when only one integration point has failed, when several integration points have failed or when all integrations points have failed.

Damage applies to the 3 Young's moduli and the 3 shear moduli. Damage is different for tension and compression. Nine damage variables are used: d_{1t}, d_{2t}, d_{3t}, d_{1c}, d_{2c}, d_{3c}, d_{12}, d_{23}, d_{13}. The damaged flexibility matrix is:

$$S^{dam} = \begin{pmatrix} \frac{1}{E_a(1-d_{1t/c})} & \frac{-\upsilon_{ba}}{E_b} & \frac{-\upsilon_{ca}}{E_c} & 0 & 0 & 0 \\ \frac{-\upsilon_{ba}}{E_b} & \frac{1}{E_b(1-d_{2t/c})} & \frac{-\upsilon_{cb}}{E_c} & 0 & 0 & 0 \\ \frac{-\upsilon_{ca}}{E_c} & \frac{-\upsilon_{cb}}{E_c} & \frac{1}{E_c(1-d_{3t/c})} & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1}{G_{ab}(1-d_{12})} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1}{G_{bc}(1-d_{23})} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1}{G_{ca}(1-d_{31})} \end{pmatrix}$$

The nine damage variables are calculated as follows:

$$d_k = \max\left(d_k ; D_k^c \left\langle \frac{\varepsilon_k - \varepsilon_k^s}{\varepsilon_k^c - \varepsilon_k^s} \right\rangle_+ \right)$$

with k = 1t, 2t, 3t, 1c, 2c, 3c, 12, 23, 31.

$\langle\ \rangle_+$ is the positive part: $\langle x \rangle_+ = \begin{cases} x & \text{if } x > 0 \\ 0 & \text{if } x < 0 \end{cases}$.

Damage in compression may be deactivated with the NDAM flag. In this case, damage in compression is null, and only damage in tension and for shear behavior are taken into account.

The nine damage variables may be post-processed through additional variables. The number of additional variables for solids written to the d3plot and d3thdt databases is input by the optional *DATABASE_EXTENT_BINARY card as variable NEIPH. These additional variables are tabulated below:

History Variable	Description	Value	LS-Prepost history variable
d_{1t}	*damage in traction along a*		*plastic strain*
d_{2t}	*damage in traction along b*		*1*
d_{3t}	*damage in traction along c*		*2*
d_{1c}	*damage in compression along a*	*0 – no damage*	*3*
d_{2c}	*damage in compression along b*		*4*
d_{3c}	*damage in compression along c*	$0 < d_k \leq D_k^c$ *– damage*	*5*
d_{12}	*shear damage in ab-plane*		*6*
d_{23}	*shear damage in bc-plane*		*7*
d_{13}	*shear damage in ac-plane*		*8*

The first damage variable is stored as in the place of effective plastic strain. The eight other damage variables may be plotted in LS-Prepost as element history varaiables.

*MAT_ORTHOTROPIC_ADVANCED_DAMAGE

This is Material Type 223. An orthotropic material with superimposed advanced damages and optional failure for composites can be defined. This model is valid only for 3D solid elements, with reduced or full integration. The elastic behavior is the same as MAT_022. Seven damage variables are defined: d_1, d_2, d_3, which are the matrix damages along the 3 directions, d^f_{1t}, d^f_{2t}, d^f_{1c}, d^f_{2c}, which are the fiber damages in tension and in compression along the 2 principal directions. Deactivation indices permit to de-activate matrix damages in compression (cracks closure). A delay effect can be taken into account for both damages (matrix and fibers). In addition, thirteen failure criteria are available: nine critical strains (tension, compression and shear, along the 3 directions), and four critical fiber damages (tension and compression, along the 2 principal directions). When failure occurs, elements are deleted (erosion). Failure depends on the number of integration points failed through the element. See the material description below.

Card 1

	1	2	3	4	5	6	7	8
Variable	MID	RO	EA	EB	EC	PRBA	PRCA	PRCB
Type	A8	F	F	F	F	F	F	F
Default	none	None	none	none	none	none	none	none

Card 2

Variable	GAB	GBC	GCA	NIP	AOPT	MACF		
Type	F	F	F	I	F	I		
Default	none	None	none	None	0.0	0		

Card 3

Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		
Default	0.0	0.0	0.0	0.0	0.0	0.0		

Card 4

	1	2	3	4	5	6	7	8
Variable	V1	V2	V3	D1	D2	D3	BETA	
Type	F	F	F	F	F	F	F	
Default	0.0	0.0	0.0	0.0	0.0	0.0	0.0	

Card 5

Variable	NERODE	NDAM	CRIT	EPS1TF	EPS2TF	EPS3TF	EPS1CF	EPS2CF
Type	I	I	I	F	F	F	F	F
Default	0	0	0	0.0	0.0	0.0	0.0	0.0

Card 6

Variable	EPS3CF	EPS12F	EPS23F	EPS31F	FIBD1T	FIBD2T	FIBD1C	FIBD2C
Type	F	F	F	F	F	F	F	F
Default	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0

Matrix Damage Characteristics

Card 7

Variable	THMXB1	THMXB2	DC1N	DC2N	DC3N	DC1T	DC2T	DC3T
Type	F	F	F	F	F	F	F	F
Default	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0

Card 8

Variable	Y01N	Y02N	Y03N	Y01T	Y02T	Y03T	YC1N	YC2N
Type	F	F	F	F	F	F	F	F
Default	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0

Card 9

Variable	YC3N	YC1T	YC2T	YC3T	P1N	P2N	P3N	P1T
Type	F	F	F	F	F	F	F	F
Default	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0

Card 10

Variable	P2T	P3T	DEPS01	DEPS02	DEPS03	AF1	AF2	AF3
Type	F	F	F	F	F	F	F	F
Default	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0

Card 11

Variable	H1N	H1P	H1HP	H2N	H2P	H2HP	H3N	H3P
Type	F	F	F	F	F	F	F	F
Default	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0

Card 12

Variable	H3HP	EPSR1	EPSR2	EPSR3	TAUC1	TAUC2	TAUC3	A1DEL
Type	F	F	F	F	F	F	F	F
Default	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0

Card 13

Variable	A2DEL	A3DEL						
Type	F	F						
Default	0.0	0.0						

Fiber Damage Characteristics

Card 14

Variable	DFC1T	DFC2T	DFC1C	DFC2C	EF01T	EF02T	EF01C	EF02C
Type	F	F	F	F	F	F	F	F
Default	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0

Card 15

Variable	YFC1T	YFC2T	YFC1C	YFC2C	PF1T	PF2T	PF1C	PF2C
Type	F	F	F	F	F	F	F	F
Default	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0

Card 16

Variable	H111T	H221T	H331T	H441T	H551T	H661T	H121T	H231T
Type	F	F	F	F	F	F	F	F
Default	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0

Card 17

Variable	H131T	H111C	H221C	H331C	H441C	H551C	H661C	H121C
Type	F	F	F	F	F	F	F	F
Default	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0

Card 18

Variable	H231C	H131C	H112T	H222T	H332T	H442T	H552T	H662T
Type	F	F	F	F	F	F	F	F
Default	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0

Card 19

Variable	H122T	H232T	H132T	H112C	H222C	H332C	H442C	H552C
Type	F	F	F	F	F	F	F	F
Default	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0

Card 20

Variable	H662C	H122C	H232C	H132C	TFC1T	TFC2T	TFC1C	TFC2C
Type	F	F	F	F	F	F	F	F
Default	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0

Card 21

Variable	AF1T	AF2T	AF1C	AF2C				
Type	F	F	F	F				
Default	0.0	0.0	0.0	0.0				

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
EA	E_a, Young's modulus in a-direction.
EB	E_b, Young's modulus in b-direction.
EC	E_c, Young's modulus in c-direction.
PRBA	ν_{ba}, Poisson ratio, ba.
PRCA	ν_{ca}, Poisson ratio, ca.
PRCB	ν_{cb}, Poisson ratio, cb.
GAB	G_{ab}, Shear modulus, ab.
GBC	G_{bc}, Shear modulus, bc.
GCA	G_{ca}, Shear modulus, ca.

VARIABLE	DESCRIPTION
NIP	Number of integration points in elements using the material. Necessary to manage failure and erosion through the number of integration points, according to NERODE flag (see below). This dictates that all elements using this material have the same number of integration points (same element type). NIP=1 for elements with reduced integration, NIP=8 for fully integrated elements.
AOPT	Material axes option (see MAT_*OPTION* TROPIC_ELASTIC for a more complete description): EQ. 0.0: locally orthotropic with material axes determined by element nodes 1, 2, and 4, as with *DEFINE_COORDINATE_ NODES. EQ. 1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center; this is the a-direction. This option is for solid elements only. EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_ VECTOR. EQ. 3.0: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle, BETA, from a line in the plane of the element defined by the cross product of the vector **v** with the element normal. EQ. 4.0: locally orthotropic in cylindrical coordinate system with the material axes determined by a vector **v**, and an originating point, P, which define the centerline axis. This option is for solid elements only. LT.0.0: the absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_NODES, *DEFINE_ COORDINATE_SYSTEM or *DEFINE_COORDINATE_ VECTOR). Available in R3 version of 971 and later.
MACF	Material axes change flag for brick elements: EQ.1: No change, default, EQ.2: switch material axes a and b, EQ.3: switch material axes a and c, EQ.4: switch material axes b and c.
XP,YP,ZP	Coordinates of point **p** for AOPT = 1.
A1,A2,A3	Components of vector **a** for AOPT = 2.
V1,V2,V3	Components of vector **v** for AOPT = 3.
D1,D2,D3	Components of vector **d** for AOPT = 2.
BETA	Material angle in degrees for AOPT = 3, may be overridden on the element card, see *ELEMENT_SHELL_BETA or *ELEMENT_ SOLID_ORTHO.

VARIABLE	DESCRIPTION
NERODE	Failure flag: EQ. 0: No failure (default) EQ. 1: Failure as soon as a tension failure criterion is reached in one integration point EQ. 2: Failure as soon as a tension or compression failure criterion is reached in one integration point EQ. 3: Failure as soon as a tension, compression or shear failure criterion is reached in one integration point EQ. 4: Failure as soon as a tension failure criterion is reached in all integration points EQ. 5: Failure as soon as a tension or compression failure criterion is reached in all integration points EQ. 6: Failure as soon as a tension, compression or shear failure criterion is reached in all integration points
NDAM	Damage flag: EQ. 0: No damage (default) EQ. 1: Damage without delay effect EQ. 2: Damage with delay effect
NCRIT	Flag for failure criterion, NCRIT EQ.0 Critical strain criterion EQ.1 Critical damage criterion EQ.2 Critical strain or damage criterion (the first reached)
EPS1TF	Failure strain in tension along the a-direction
EPS2TF	Failure strain in tension along the b-direction
EPS3TF	Failure strain in tension along the c-direction
EPS1CF	Failure strain in compression along the a-direction
EPS2CF	Failure strain in compression along the b-direction
EPS3CF	Failure strain in compression along the c-direction
EPS12F	Failure shear strain in the ab-plane
EPS23F	Failure shear strain in the bc-plane
EPS31F	Failure shear strain in the ac-plane
FIBD1T	Failure fiber damage in tension along the a-direction
FIBD2T	Failure fiber damage in tension along the b-direction
FIBD1C	Failure fiber damage in compression along the a-direction
FIBD2C	Failure fiber damage in compression along the a-direction

Failure can occur according a critical strain criterion, a critical damage criterion, or the first reached among the two. Failure strain criteria are defined along the 3 orthotropic directions, in tension, in compression and for shear behavior. Failure fiber damage criteria are defined along the 2 principal directions, in tension and in compression. When failure occurs, elements are deleted (erosion). Under the control of the NERODE flag, failure may occur either when only one integration point has failed, or when all integrations points have failed.

VARIABLE	DESCRIPTION
Matrix Damage Characteristics	
THMXB1	Thermodynamical forces coupling for matrix damage, b_1
THMXB2	Thermodynamical forces coupling for matrix damage, b_2
DC1N	Critical matrix damage dir 1, normal component, d_{c1}^n
DC2N	Critical matrix damage dir 2, normal component, d_{c2}^n
DC3N	Critical matrix damage dir 3, normal component, d_{c3}^n
DC1T	Critical matrix damage dir 1, tangential component, d_{c1}^t
DC2T	Critical matrix damage dir 2, tangential component, d_{c2}^t
DC3T	Critical matrix damage dir 3, tangential component, d_{c3}^t
Y01N	Threshold matrix damage force dir 1, normal component, y_{01}^n
Y02N	Threshold matrix damage force dir 2, normal component, y_{02}^n
Y03N	Threshold matrix damage force dir 3, normal component, y_{03}^n
Y01T	Threshold matrix damage force dir 1, tangential component, y_{01}^t
Y02T	Threshold matrix damage force dir 2, tangential component, y_{02}^t
Y03T	Threshold matrix damage force dir 3, tangential component, y_{03}^t
YC1N	Critical matrix damage force dir 1, normal component, y_{c1}^n
YC2N	Critical matrix damage force dir 2, normal component, y_{c2}^n
YC3N	Critical matrix damage force dir 3, normal component, y_{c3}^n
YC1T	Critical matrix damage force dir 1, tangential component, y_{c1}^t
YC2T	Critical matrix damage force dir 2, tangential component, y_{c2}^t
YC3T	Critical matrix damage force dir 3, tangential component, y_{c3}^t
P1N	Matrix damage exponent dir 1, normal component, p_1^n
P2N	Matrix damage exponent dir 2, normal component, p_2^n
P3N	Matrix damage exponent dir 3, normal component, p_3^n
P1T	Matrix damage exponent dir 1, tangential component, p_1^t
P2T	Matrix damage exponent dir 2, tangential component, p_2^t
P3T	Matrix damage exponent dir 3, tangential component, p_3^t
DEPS01	Closure domain definition dir 1 for deactivation, $\Delta\varepsilon_1^0$
DEPS02	Closure domain definition dir 2 for deactivation, $\Delta\varepsilon_2^0$
DEPS03	Closure domain definition dir 3 for deactivation, $\Delta\varepsilon_3^0$
AF1	Closure domain definition dir 1 for deactivation, a^f_1
AF2	Closure domain definition dir 2 for deactivation, a^f_2
AF3	Closure domain definition dir 3 for deactivation, a^f_3
H1N	Tensor of the matrix damage effects dir 1, h^1_n
H1P	Tensor of the matrix damage effects dir 1, h^1_p
H1HP	Tensor of the matrix damage effects dir 1, h^1_{hp}
H2N	Tensor of the matrix damage effects dir 2, h^2_n
H2P	Tensor of the matrix damage effects dir 2, h^2_p
H2HP	Tensor of the matrix damage effects dir 2, h^2_{hp}
H3N	Tensor of the matrix damage effects dir 3, h^3_n
H3P	Tensor of the matrix damage effects dir 3, h^3_p
H3HP	Tensor of the matrix damage effects dir 3, h^3_{hp}
EPSR1	Residual strains parameter dir 1, ξ_1
EPSR2	Residual strains parameter dir 2, ξ_2
EPSR3	Residual strains parameter dir 3, ξ_3
TAUC1	Delay effect on matrix damage dir 1, τ_{c1}
TAUC2	Delay effect on matrix damage dir 2, τ_{c2}

VARIABLE	DESCRIPTION
TAUC3	Delay effect on matrix damage dir 3, τ_{c3}
A1DEL	Delay effect on matrix damage dir 1, a_1
A2DEL	Delay effect on matrix damage dir 2, a_2
A3DEL	Delay effect on matrix damage dir 3, a_3

Fiber Damage Characteristics

DFC1T	Critical fiber damage dir 1, tension, d^f_{c1t}
DFC2T	Critical fiber damage dir 2, tension, d^f_{c2t}
DFC1C	Critical fiber damage dir 3, compression, d^f_{c1c}
DFC2C	Critical fiber damage dir 1, compression, d^f_{c2c}
EF01T	Threshold fiber damage strain dir 1, tension, ε^f_{01t}
EF02T	Threshold fiber damage strain dir 2, tension, ε^f_{02t}
EF01C	Threshold fiber damage strain dir 1, compression, ε^f_{01c}
EF02C	Threshold fiber damage strain dir 2, compression, ε^f_{02c}
YFC1T	Critical fiber damage d dir 1, tension, y^f_{c1t}
YFC2T	Critical fiber damage force dir 2, tension, y^f_{c2t}
YFC1C	Critical fiber damage force dir 1, compression, y^f_{c1c}
YFC2C	Critical fiber damage force dir 2, compression, y^f_{c2c}
PF1T	Fiber damage exponent dir 1, tension, p^f_{1t}
PF2T	Fiber damage exponent dir 2, tension, p^f_{2t}
PF1C	Fiber damage exponent dir 3, compression, p^f_{1c}
PF2C	Fiber damage exponent dir 1, compression, p^f_{2c}
H111T	Tensor of the fiber damage effects dir 1, tension, $h_{11}{}^{1t}$
H221T	Tensor of the fiber damage effects dir 1, tension, $h_{22}{}^{1t}$
H331T	Tensor of the fiber damage effects dir 1, tension, $h_{33}{}^{1t}$
H441T	Tensor of the fiber damage effects dir 1, tension, $h_{44}{}^{1t}$
H551T	Tensor of the fiber damage effects dir 1, tension, $h_{55}{}^{1t}$
H661T	Tensor of the fiber damage effects dir 1, tension, $h_{66}{}^{1t}$
H121T	Tensor of the fiber damage effects dir 1, tension, $h_{12}{}^{1t}$
H231T	Tensor of the fiber damage effects dir 1, tension, $h_{23}{}^{1t}$
H131T	Tensor of the fiber damage effects dir 1, tension, $h_{13}{}^{1t}$
H111C	Tensor of the fiber damage effects dir 1, compression, $h_{11}{}^{1c}$
H221C	Tensor of the fiber damage effects dir 1, compression, $h_{22}{}^{1c}$
H331C	Tensor of the fiber damage effects dir 1, compression, $h_{33}{}^{1c}$
H441C	Tensor of the fiber damage effects dir 1, compression, $h_{44}{}^{1c}$
H551C	Tensor of the fiber damage effects dir 1, compression, $h_{55}{}^{1c}$
H661C	Tensor of the fiber damage effects dir 1, compression, $h_{66}{}^{1c}$
H121C	Tensor of the fiber damage effects dir 1, compression, $h_{12}{}^{1c}$
H231C	Tensor of the fiber damage effects dir 1, compression, $h_{23}{}^{1c}$
H131C	Tensor of the fiber damage effects dir 1, compression, $h_{13}{}^{1c}$
H112T	Tensor of the fiber damage effects dir 2, tension, $h_{11}{}^{2t}$
H222T	Tensor of the fiber damage effects dir 2, tension, $h_{22}{}^{2t}$
H332T	Tensor of the fiber damage effects dir 2, tension, $h_{33}{}^{2t}$
H442T	Tensor of the fiber damage effects dir 2, tension, $h_{44}{}^{2t}$
H552T	Tensor of the fiber damage effects dir 2, tension, $h_{55}{}^{2t}$

VARIABLE	DESCRIPTION
H662T	Tensor of the fiber damage effects dir 2, tension, h_{66}^{2t}
H122T	Tensor of the fiber damage effects dir 2, tension, h_{12}^{2t}
H232T	Tensor of the fiber damage effects dir 2, tension, h_{23}^{2t}
H132T	Tensor of the fiber damage effects dir 2, tension, h_{13}^{2t}
H112C	Tensor of the fiber damage effects dir 2, compression, h_{11}^{2c}
H222C	Tensor of the fiber damage effects dir 2, compression, h_{22}^{2c}
H332C	Tensor of the fiber damage effects dir 2, compression, h_{33}^{2c}
H442C	Tensor of the fiber damage effects dir 2, compression, h_{44}^{2c}
H552C	Tensor of the fiber damage effects dir 2, compression, h_{55}^{2c}
H662C	Tensor of the fiber damage effects dir 2, compression, h_{66}^{2c}
H122C	Tensor of the fiber damage effects dir 2, compression, h_{12}^{2c}
H232C	Tensor of the fiber damage effects dir 2, compression, h_{23}^{2c}
H132C	Tensor of the fiber damage effects dir 2, compression, h_{13}^{2c}
TFC1T	Delay effect on fiber damage dir 1, tension, τ^{f}_{c1t}
TFC2T	Delay effect on fiber damage dir 2, tension, τ^{f}_{c2t}
TFC1C	Delay effect on fiber damage dir 1, compression, τ^{f}_{c1c}
TFC2C	Delay effect on fiber damage dir 2, compression, τ^{f}_{c2c}
AF1T	Delay effect on fiber damage dir 1, tension, a^{f}_{1t}
AF2T	Delay effect on fiber damage dir 2, tension, a^{f}_{2t}
AF1C	Delay effect on fiber damage dir 1, compression, a^{f}_{1c}
AF2C	Delay effect on fiber damage dir 2, compression, a^{f}_{2c}

There are two superimposed damages: matrix damages in the 3 directions, d_1, d_2 and d_3, and fiber damages in the 2 principal directions, in traction and in compression, d^{f}_{1t}, d^{f}_{2t}, d^{f}_{1c} and d^{f}_{2c}.

The seven damage variables are function of the strain tensor, according to the ONERA Continuum Damage Mechanics model: see “A New Formulation of Continuum Damage Mechanics (CDM) for Composite Materials”, J.F. Maire, J.L. Chaboche, ONERA, *Aerospace Science and Technology*, 1997, Vol 1, n° 4, pp 247-257.

The constitutive law is:

$$[\sigma] = C^{eff} : [\varepsilon] - (S_r)^{-1} : ([\varepsilon^s] + [\varepsilon^r])$$

where:

$[\sigma]$ is the stress tensor,

$[\varepsilon]$ is the strain tensor,

$[\varepsilon^s]$ is the stored strain tensor,

$[\varepsilon^r]$ is the residual strain tensor,

C^{eff} is the effective stiffness matrix, $C^{eff} = [S^{eff}]^{-1}$,

$$S^{eff} = S^0 + \sum_i \eta_i * d_i * H_i^0 + \sum_j d_j^f * H_j^{f0},$$

$$S_r = S^0 + \sum_j d_j^f * H_j^{f0},$$

$$S^0 = \begin{bmatrix} 1/E_a & -v_{ba}/E_b & -v_{ca}/E_c & 0 & 0 & 0 \\ -v_{ba}/E_b & 1/E_b & -v_{cb}/E_c & 0 & 0 & 0 \\ -v_{ca}/E_c & -v_{cb}/E_c & 1/E_c & 0 & 0 & 0 \\ 0 & 0 & 0 & 1/G_{ab} & 0 & 0 \\ 0 & 0 & 0 & 0 & 1/G_{bc} & 0 \\ 0 & 0 & 0 & 0 & 0 & 1/G_{ca} \end{bmatrix} = [C^0]^{-1},$$

η_i is the index of matrix damage deactivation, defined as follow:

$$\begin{cases} \eta_i = 1 & if \quad \varepsilon_{ii} \geq \Delta\varepsilon_i^f \\ \eta_i = \dfrac{1}{2}\left(1 - \cos\left(\dfrac{\pi}{2}\dfrac{\varepsilon_{ii} + \Delta\varepsilon_i^f}{\Delta\varepsilon_i^f}\right)\right) & if \quad -\Delta\varepsilon_i^f < \varepsilon_{ii} < \Delta\varepsilon_i^f \\ \eta_i = 0 & if \quad -\Delta\varepsilon_i^f \geq \varepsilon_{ii} \end{cases}$$

with $\Delta\varepsilon_i^f = (1 + a_i^f d_i)\Delta\varepsilon_i^0, i = 1,2,3$: defines the closure domain of the cracks, for deactivation of the matrix damage,

$$H_1^0=\begin{bmatrix} h_n^1/E_a & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & h_{hp}^1/G_{bc} & 0 \\ 0 & 0 & 0 & 0 & 0 & h_p^1/G_{ca} \end{bmatrix},$$

$$H_2^0=\begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & h_n^2/E_b & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & h_{hp}^2/G_{ab} & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & h_p^2/G_{ca} \end{bmatrix},$$

$$H_3^0=\begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & h_n^3/E_c & 0 & 0 & 0 \\ 0 & 0 & 0 & h_p^3/G_{ab} & 0 & 0 \\ 0 & 0 & 0 & 0 & h_{hp}^3/G_{bc} & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix},$$

$$H_j^{f0}=\begin{bmatrix} h_{11}^j/E_a & h_{12}^j S_{12}^0 & h_{13}^j S_{13}^0 & 0 & 0 & 0 \\ h_{12}^j S_{12}^0 & h_{22}^j/E_b & h_{23}^j S_{23}^0 & 0 & 0 & 0 \\ h_{13}^j S_{13}^0 & h_{23}^j S_{23}^0 & h_{33}^j/E_c & 0 & 0 & 0 \\ 0 & 0 & 0 & h_{44}^j/G_{bc} & 0 & 0 \\ 0 & 0 & 0 & 0 & h_{55}^j/G_{ca} & 0 \\ 0 & 0 & 0 & 0 & 0 & h_{66}^j/G_{ab} \end{bmatrix}$$

with j = 1t, 2t, 1c, 2c.

The matrix thermodynamic forces are calculated in function of positive strain:

$$\begin{cases} y_1^n = \frac{1}{2}\cdot C_{11}^0 \cdot \varepsilon_{11}^+ \cdot \varepsilon_{11}^+ \\ y_1^t = \frac{1}{2}(C_{44}^0 \cdot \varepsilon_{12}^+ \cdot \varepsilon_{12}^+ + b_1 \cdot C_{66}^0 \cdot \varepsilon_{13}^+ \cdot \varepsilon_{13}^+) \end{cases}$$

$$\begin{cases} y_2^n = \frac{1}{2}\cdot C_{22}^0 \cdot \varepsilon_{22}^+ \cdot \varepsilon_{22}^+ \\ y_2^t = \frac{1}{2}(C_{44}^0 \cdot \varepsilon_{12}^+ \cdot \varepsilon_{12}^+ + b_2 \cdot C_{55}^0 \cdot \varepsilon_{23}^+ \cdot \varepsilon_{23}^+) \end{cases}$$

$$\begin{cases} y_3^n = \frac{1}{2}\cdot C_{33}^0 \cdot \varepsilon_{33}^+ \cdot \varepsilon_{33}^+ \\ y_3^t = \frac{1}{2}(b_2 \cdot C_{55}^0 \cdot \varepsilon_{23}^+ \cdot \varepsilon_{23}^+ + b_1 \cdot C_{66}^0 \cdot \varepsilon_{13}^+ \cdot \varepsilon_{13}^+) \end{cases}$$

The fiber thermodynamic forces are calculated in function of strain:

$$\begin{cases} y_{1t}^f = \frac{1}{2}\cdot C_{11}^0 \cdot \varepsilon_{11} \cdot \varepsilon_{11} & if\ \varepsilon_{11} > 0 \\ y_{1t}^f = 0 & if\ \varepsilon_{11} \le 0 \end{cases}$$

$$\begin{cases} y_{1c}^f = \frac{1}{2}\cdot C_{11}^0 \cdot \varepsilon_{11} \cdot \varepsilon_{11} & if\ \varepsilon_{11} < 0 \\ y_{1c}^f = 0 & if\ \varepsilon_{11} \ge 0 \end{cases}$$

$$\begin{cases} y_{2t}^f = \frac{1}{2}\cdot C_{22}^0 \cdot \varepsilon_{22} \cdot \varepsilon_{22} & if\ \varepsilon_{22} > 0 \\ y_{2t}^f = 0 & if\ \varepsilon_{22} \le 0 \end{cases}$$

$$\begin{cases} y_{2c}^f = \frac{1}{2}\cdot C_{22}^0 \cdot \varepsilon_{22} \cdot \varepsilon_{22} & if\ \varepsilon_{22} < 0 \\ y_{2c}^f = 0 & if\ \varepsilon_{22} \ge 0 \end{cases}$$

Then, the matrix damage variables are calculated as follow:

$$d_i = \min(g_{i^n}(y_{i^n}) + g_{i^t}(y_{i^t}), d_{ci^n} + d_{ci^t})$$

with:

$$\sqrt{y_i^n} - \sqrt{y_{0(i)}^n} >_+ \overline{\sqrt{y_{c(i)}^n}}$$

$$1 - \exp - \left(p_i^n \right)$$

$$, \quad i = 1,2,3 \quad ,$$

$$g_i^n = d_{c(i)}^n \left(g_i^t = d_{c(i)}^t \left(1 - \exp - \left(\frac{\sqrt{y_i^t} - \sqrt{y_{0(i)}^t}_+}{\sqrt{y_{c(i)}^t}} \right)^{p_i^t} \right) \right.$$

and the fiber damage variables are calculated as follow:

$$d_{jf} = \min(g_{jf}(y_{jf}), d_{cjf})$$

with:

$$\sqrt{y_j^f} - \sqrt{y_{0(j)}^f} >_+ \overline{\sqrt{y_{c(j)}^f}}$$

$$1 - \exp - \left(p_j^f \right) \quad , \quad j = 1\mathrm{t}, 2\mathrm{t}, 1\mathrm{c}, 2\mathrm{c} \quad .$$

$$g_j^f = d_{c(j)}^f$$

$\langle \ \rangle_+$ is the positive part: $\langle x \rangle_+ = \begin{cases} x & \text{if x} > 0 . \\ 0 & \text{if x} < 0 \end{cases}$

The delay effect can be taken into account by setting the NDAM flag to 2. It permits to avoid the mesh dependency by introducing a limitation on the damage rate.

In this case, the matrix damage rate is:

$$\dot{d}_i = \frac{1}{\tau_{ci}} (1 - \exp(-a_i < g_i^n + g_i^t - d_i >^+)) \text{ with } i = 1,2,3 \quad ,$$

and the fiber damage rate is:

$$\dot{d}_{jf} = \frac{1}{\tau_{cjf}}(1-\exp(-a_{jf} < g_j^f - d_j^f >^+)) \text{ with } j = 1t, 2t, 1c, 2c .$$

The seven damage variables, the seven indices of damage deactivation, the stored, residual and total strains may be post-processed through additional variables. The number of additional variables for solids written to the d3plot and d3thdt databases is input by the optional *DATABASE_EXTENT_BINARY card as variable NEIPH. These additional variables are tabulated below:

History Variable	Description	Value	LS-Prepost history variable
d_1	*matrix damage along a*	*0 – no damage* *For matrix:* $0 < d_i \le d_{ci}^n + d_{ci}^t$ *– damage* *For fiber:* $0 < d_j^f \le d_{cj}^f$ *– damage*	*1*
d_2	*matrix damage along b*		*2*
d_3	*matrix damage along c*		*3*
d_{1t}^f	*fiber damage in traction along a*		*4*
d_{2t}^f	*fiber damage in traction along b*		*5*
d_{1c}^f	*fiber damage in compression along a*		*6*
d_{2c}^f	*fiber damage in compression along b*		*7*
η_1	*index of deactivation of matrix damage along a*	$0 \le \eta_i \le 1$	*18*
η_2	*index of deactivation of matrix damage along b*		*19*
η_3	*index of deactivation of matrix damage along c*		*20*
ε_1^s	*Stored strain 1*		*25*
ε_2^s	*Stored strain 2*		*26*
ε_3^s	*Stored strain 3*		*27*
ε_4^s	*Stored strain 4*		*28*
ε_5^s	*Stored strain 5*		*29*
ε_6^s	*Stored strain 6*		*30*
ε_1^r	*Residual strain 1*		*31*
ε_2^r	*Residual strain 2*		*32*
ε_3^r	*Residual strain 3*		*33*
ε_4^r	*Residual strain 4*		*34*
ε_5^r	*Residual strain 5*		*35*
ε_6^r	*Residual strain 6*		*36*
ε_1	*Total strain 1*		*37*
ε_2	*Total strain 2*		*38*
ε_3	*Total strain 3*		*39*
ε_4	*Total strain 4*		*40*
ε_5	*Total strain 5*		*41*
ε_6	*Total strain 6*		*42*
v_{11}	*Direction of orthotropy 1, x-comp*		*43*
v_{12}	*Direction of orthotropy 1, y-comp*		*44*

v_{13}	*Direction of orthotropy 1, z-comp*		*45*
v_{21}	*Direction of orthotropy 2, x-comp*		*46*
v_{22}	*Direction of orthotropy 2, y-comp*		*47*
v_{23}	*Direction of orthotropy 2, z-comp*		*48*

All these variables may be plotted in Ls-Prepost as element history variables 1 through 48. v_{1x}, v_{1y}, v_{1z}, v_{2x}, v_{2y}, v_{2z} define the 2 vectors of the principal orthotropic directions for each element. These variables are used to be output in the dynain file in case of prestress simulation

*MAT_TABULATED_JOHNSON_COOK

This is Material Type 224. An elasto-viscoplastic material with arbitrary stress versus strain curve(s) and arbitrary strain rate dependency can be defined. Plastic heating causes adiabatic temperature increase and material softening. Optional plastic failure strain can be defined as a function of triaxiality, strain rate, temperature and/or element size. This material model resembles the original Johnson-Cook material (see *MAT_015) but with the possibility of general tabulated input parameters. The model is available for shell and solid elements.

Card 1

Variable	MID	RO	E	PR	CP	TR	BETA	NUMINT
Type	A8	F	F	F	F	F	F	F
Default	none	none	none	none	none	0.0	1.0	1.0

Card 2

Variable	TABK1	TABKT	LCF	LCG	LCH	LCI		
Type	F	F	F	F	F	F		
Default	0	0	0	0	0	0		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.
CP	Specific heat.
TR	Room temperature.
BETA	Amount of plastic work converted into heat.

VARIABLE	DESCRIPTION
NUMINT	Number of integration points which must fail before the element is deleted. Available for shells and solids. LT.0.0: |NUMINT| is percentage of integration points/layers which must fail before element fails. For fully integrated shells, a methodology is used where a layer fails if one integration point fails and then the given percentage of layers must fail before the element fails.
LCK1	Load curve ID or Table ID. The load curve ID defines effective stress as a function of effective plastic strain. The table ID defines for each plastic strain rate value a load curve ID giving the (isothermal) effective stress versus effective plastic strain for that rate.
LCKT	Table ID defining for each temperature value a load curve ID giving the (quasi-static) effective stress versus effective plastic strain for that temperature.
LCF	Load curve ID or Table ID. The load curve ID defines plastic failure strain as a function of triaxiality. The table ID defines for each Lode angle a load curve ID giving the plastic failure strain versus triaxiality for that Lode angle. (Table option only for solids and not yet generally supported).
LCG	Load curve ID defining plastic failure strain as a function of strain rate.
LCH	Load curve ID defining plastic failure strain as a function of temperature
LCI	Load curve ID defining plastic failure strain as a function of element size.

Remarks:

The flow stress σ_y is expressed as a function of plastic strain ε_p, plastic strain rate $\dot{\varepsilon}_p$ and temperature T via the following formula (using load curves/tables LCK1 and LCKT):

$$\sigma_y = k1(\varepsilon_p, \dot{\varepsilon}_p) \cdot kt(\varepsilon_p, T)$$

Optional plastic failure strain is defined as a function of triaxiality p / σ_{vm}, plastic strain rate $\dot{\varepsilon}_p$, temperature T and element size l_0 (square root of element area for shells and volume over maximum area for solids) by

$$\varepsilon_{pf} = f\left(\frac{p}{\sigma_{vm}}\right) g(\dot{\varepsilon}_p) h(T) i(l_c)$$

using load curves/tables LCF, LCG, LCH and LCI.

Temperature increase is caused by plastic work

$$T = T_R + \frac{\beta}{C_p \rho} \int \sigma_y \dot{\varepsilon}_p$$

with room temperature T_R, dissipation factor β, specific heat C_p, and density ρ.

Relevant history variables of this material mode are plastic strain rate (#1), plastic work (#7), ratio of plastic strain to plastic failure strain (#8), element size (#9), and temperature (#10).

*MAT_ VISCOPLASTIC_MIXED_HARDENING

This is Material Type 225. An elasto-viscoplastic material with an arbitrary stress versus strain curve and arbitrary strain rate dependency can be defined. Kinematic, isotropic, or a combination of kinematic and isotropic hardening can be specified. Also, failure based on plastic strain can be defined.

Card 1

Variable	MID	RO	E	PR	LCSS	BETA		
Type	A8	F	F	F	I	F		
Default	none	none	none	none	none	0.0		

Card 2

Variable	FAIL							
Type	F							
Default	1.0E+20							

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.

VARIABLE	DESCRIPTION
LCSS	Load curve ID or Table ID. Load curve ID defining effective stress versus effective plastic strain The table ID defines for each strain rate value a load curve ID giving the stress versus effective plastic strain for that rate, See Figure 24.1. The stress versus effective plastic strain curve for the lowest value of strain rate is used if the strain rate falls below the minimum value. Likewise, the stress versus effective plastic strain curve for the highest value of strain rate is used if the strain rate exceeds the maximum value. NOTE: The strain rate values defined in the table may be given as the natural logarithm of the strain rate. If the *first* stress-strain curve in the table corresponds to a negative strain rate, LS-DYNA assumes that the natural logarithm of the strain rate value is used. Since the tables are internally discretized to equally space the points, natural logarithms are necessary, for example, if the curves correspond to rates from 10.e-04 to 10.e+04.
BETA	Hardening parameter, 0<BETA<1. EQ.0.0: Pure kinematic hardening EQ.1.0: Pure isotropic hardening 0.0<BETA<1.0: Mixed hardening
FAIL	Failure flag. LT.0.0: User defined failure subroutine is called to determine failure EQ.0.0: Failure is not considered. This option is recommended if failure is not of interest since many calculations will be saved. GT.0.0: Plastic strain to failure. When the plastic strain reaches this value, the element is deleted from the calculation.

***MAT_PML_ELASTIC**

This is Material Type 230. This is a perfectly-matched layer (PML) material — an absorbing layer material used to simulate wave propagation in an unbounded isotropic elastic medium — and is available only for solid 8-node bricks (element type 2). This material implements the 3D version of the Basu-Chopra PML [Basu and Chopra (2003,2004), Basu (2009)].

Card	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR				
Type	A8	F	F	F				
Default	none	none	none	none				

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.

Remarks:

1. A layer of this material may be placed at a boundary of a bounded domain to simulate unboundedness of the domain at that boundary: the layer absorbs and attenuates waves propagating outward from the domain, without any significant reflection of the waves back into the bounded domain. The layer cannot support any static displacement.

2. It is assumed the material in the bounded domain near the layer is, or behaves like, an isotropic linear elastic material. The material properties of the layer should be set to the corresponding properties of this material.

3. The layer should form a cuboid box around the bounded domain, with the axes of the box aligned with the coordinate axes. Various faces of this box may be open, as required by the geometry of the problem, e.g., for a half-space problem, the "top" of the box should be open.

4. Internally, LS-DYNA will partition the entire PML into regions which form the “faces”, “edges” and “corners” of the above cuboid box, and generate a new material for each region. This partitioning will be visible in the d3plot file. The user may safely ignore this partitioning.

5. The layer should have 5-10 elements through its depth. Typically, 5-6 elements are sufficient if the excitation source is reasonably distant from the layer, and 8-10 elements if it is close. The size of the elements should be similar to that of elements in the bounded domain near the layer, and should be small enough to sufficiently discretize all significant wavelengths in the problem.

6. The nodes on the outer boundary of the layer should be fully constrained.

7. The stress and strain values reported by this material do not have any physical significance.

***MAT_PML_ELASTIC_FLUID**

This is Material Type 230_FLUID. This is a perfectly-matched layer (PML) material with a pressure fluid constitutive law, to be used in a wave-absorbing layer adjacent to a fluid material (*MAT_ELASTIC_FLUID) in order to simulate wave propagation in an unbounded fluid medium. See the Remarks sections of *MAT_PML_ELASTIC (*MAT_230) and *MAT_ELASTIC_FLUID (*MAT_001_FLUID) for further details.

Card	1	2	3	4	5	6	7	8
Variable	MID	RO	K	VC				
Type	A8	F	F	F				
Default	none	none	none	none				

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
K	Bulk modulus
VC	Tensor viscosity coefficient

*MAT_PML_ACOUSTIC

This is Material Type 231. This is a perfectly-matched layer (PML) material — an absorbing layer material used to simulate wave propagation in an unbounded acoustic medium — and can be used only with the acoustic pressure element formulation (element type 14). This material implements the 3D version of the Basu-Chopra PML for anti-plane motion [Basu and Chopra (2003,2004), Basu (2009)].

Card	1	2	3	4	5	6	7	8
Variable	MID	RO	C					
Type	A8	F	F					
Default	none	none	none					

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
C	Sound speed

Remarks:

1. A layer of this material may be placed at a boundary of a bounded domain to simulate unboundedness of the domain at that boundary: the layer absorbs and attenuates waves propagating outward from the domain, without any significant reflection of the waves back into the bounded domain. The layer cannot support any hydrostatic pressure.

2. It is assumed the material in the bounded domain near the layer is an acoustic material. The material properties of the layer should be set to the corresponding properties of this material.

3. The layer should form a cuboid box around the bounded domain, with the axes of the box aligned with the coordinate axes. Various faces of this box may be open, as required by the geometry of the problem, e.g., for a half-space problem, the “top” of the box should be open.

4. Internally, LS-DYNA will partition the entire PML into regions which form the “faces”, “edges” and “corners” of the above cuboid box, and generate a new material for each

region. This partitioning will be visible in the d3plot file. The user may safely ignore this partitioning.

5. The layer should have 5-10 elements through its depth. Typically, 5-6 elements are sufficient if the excitation source is reasonably distant from the layer, and 8-10 elements if it is close. The size of the elements should be similar to that of elements in the bounded domain near the layer, and should be small enough to sufficiently discretize all significant wavelengths in the problem.

6. The nodes on the outer boundary of the layer should be fully constrained.

7. The pressure values reported by this material do not have any physical significance.

*MAT_BIOT_HYSTERETIC

This is Material Type 232. This is a Biot linear hysteretic material, to be used for modeling the nearly-frequency-independent viscoelastic behaviour of soils subjected to cyclic loading, e.g. in soil-structure interaction analysis [Spanos and Tsavachidis (2001), Makris and Zhang (2000), Muscolini, Palmeri and Ricciardelli (2005)]. The hysteretic damping coefficient for the model is computed from a prescribed damping ratio by calibrating with an equivalent viscous damping model for a single-degree-of-freedom system. The damping increases the stiffness of the model and thus reduces the computed time-step size.

Card	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	ZT	FD		
Type	A8	F	F	F	F	F		
Default	none	none	none	none	0.0	3.25		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.
ZT	Damping ratio
FD	Dominant excitation frequency in Hz

Remarks:

1. The stress is computed as a function of the strain rate as

$$\sigma(t) = \int_0^t C_R(t-\tau)\dot{\varepsilon}(\tau)d\tau$$

where

$$C_R(t) = C\left[1 + \frac{2\eta}{\pi} E_1(\beta t)\right]$$

with C being the elastic isotropic constitutive tensor, η the hysteretic damping factor, and $\beta = 2\pi f_d / 10$, where f_d is the dominant excitation frequency in Hz. The function E_1 is given by

$$E_1(s) = \int_s^\infty \frac{e^{-\xi}}{\xi} d\xi$$

For efficient implementation, this function is approximated by a 5-term Prony series as

$$E_1(s) \approx \sum_{k=1}^{5} b_k \, e^{a_k s}$$

such that $b_k > 0$.

2. The hysteretic damping factor η is obtained from the prescribed damping ratio ς as

$$\eta = \pi\varsigma / \text{atan}(10) = 2.14\varsigma$$

by assuming that, for a single degree-of-freedom system, the energy dissipated per cycle by the hysteretic material is the same as that by a viscous damper, if the excitation frequency matches the natural frequency of the system.

3. The consistent Young's modulus for this model is given by

$$E_c = E\left[1 + \frac{2\eta}{\pi} g\right]$$

where

$$g = \sum_{k=1}^{5} b_k \frac{1}{a_k \beta \Delta t_n} \left[\exp\left(a_k \beta \Delta t_n\right) - 1\right]$$

Because $g > 0$, the computed element time-step size is smaller than that for the corresponding elastic element. Furthermore, the time-step size computed at any time depends on the previous time-step size. It can be demonstrated that the new computed time-step size stays within a narrow range of the previous time-step size, and for a uniform mesh, converges to a constant value. For $f_d = 3.25\text{Hz}$ and $\varsigma = 0.05$, the percentage decrease in time-step size can be expected to be about 12-15% for initial time-step sizes of less than 0.02 secs, and about 7-10% for initial time-step sizes larger than 0.02 secs.

4. The default value of the dominant frequency is chosen to be valid for earthquake excitation.

*MAT_CAZACU_BARLAT

This is Material Type 232. This material model is for Hexagonal Closed Packet (HCP) metals and is based on the work by Cazacu et al. (2006). This model is capable of describing the yielding asymmetry between tension and compression for such materials. Moreover, a parameter fit is optional and can be used to find the material parameters that describe the experimental yield stresses. The experimental data that the user should supply consists of yield stresses for tension and compression in the 00 direction, tension in the 45 and the 90 directions, and a biaxial tension test.

Card 1

Variable	MID	RO	E	PR	HR	P1	P2	ITER
Type	A8	F	F	F	F	F	F	F

Card 2

Variable	*A*	*C11*	*C22*	*C33*	LCID	E0	*K*	P3
Type	F	F	F	F	I	F	F	F

Card 3

Variable	AOPT				*C12*	*C13*	*C23*	*C44*
Type	F				F	F	F	F

Card 4

Variable				A1	A2	A3		
Type				F	F	F		

Card 5	1	2	3	4	5	6	7	8
Variable	V1	V2	V3	D1	D2	D3	BETA	*FIT*
Type	F	F	F	F	F	F	F	I

VARIABLE	DESCRIPTION
MID	Material Identification number.
RO	Constant Mass density.
E	Young's modulus E.GT.0.0: constant value E.LT.0.0: load curve ID (–E) which defines the Young's modulus as a function of plastic strain.
PR	Poisson's ratio
HR	Hardening rules: HR.EQ.1.0: linear hardening (default) HR.EQ.2.0: exponential hardening (Swift) HR.EQ.3.0: load curve HR.EQ.4.0:exponential hardening (Voce) HR.EQ.5.0:exponential hardening (Gosh) HR.EQ.6.0:exponential hardening (Hocken-Sherby)
P1	Material parameter: HR.EQ.1.0: tangent modulus HR.EQ.2.0: q, coefficient for exponential hardening law (Swift) HR.EQ.4.0: a, coefficient for exponential hardening law (Voce) HR.EQ.5.0: q, coefficient for exponential hardening law (Gosh) HR.EQ.6.0: a, coefficient for exponential hardening law (Hocket-Sherby)
P2	Material parameter: HR.EQ.1.0: yield stress for the linear hardening law HR.EQ.2.0: n, coefficient for (Swift) exponential hardening HR.EQ.4.0: c, coefficient for exponential hardening law (Voce) HR.EQ.5.0: n, coefficient for exponential hardening law (Gosh) HR.EQ.6.0: c, coefficient for exponential hardening law (Hocket-Sherby)

VARIABLE	DESCRIPTION
ITER	Iteration flag for speed: ITER.EQ.0.0: fully iterative ITER.EQ.1.0: fixed at three iterations. Generally, ITER=0.0 is recommended. However, ITER=1.0 is faster and may give acceptable results in most problems.
A	Exponent in Cazacu-Barlat's orthotropic yield surface (A>1)
C11	Material parameter (see card 5 pos. 8): FIT.EQ.1.0 or EQ.2.0: yield stress for tension in the 00 direction FIT.EQ.0.0: material parameter c11
C22	Material parameter (see card 5 pos.8) FIT.EQ.1.0 or EQ.2.0: yield stress for tension in the 45 direction FIT.EQ.0.0: material parameter c22
C33	Material parameter (see card 5 pos.8) FIT.EQ.1.0 or EQ.2.0: yield stress for tension in the 90 direction FIT.EQ.0.0: material parameter c33
LCID	Load curve ID for the hardening law (HR.EQ.3.0)
E0	Material parameter: HR.EQ.2.0: initial yield stress for exponential hardening law (Swift) (default =0.0) HR.EQ.4.0: b, coefficient for exponential hardening (Voce) HR.EQ.5.0: initial yield stress for exponential hardening (Gosh), Default=0.0 HR.EQ.6.0: b, coefficient for exponential hardening law (Hocket-Sherby)
K	Material parameter (see card 5 pos.8) FIT.EQ.1.0 or EQ.2.0: yield stress for compression in the 00 direction FIT.EQ.0.0: material parameter (-1<k<1)
P3	Material parameter: HR.EQ.5.0: p, coefficient for exponential hardening (Gosh) HR.EQ.6.0: n, exponent for exponential hardening law (Hocket-Sherby)

VARIABLE	DESCRIPTION
AOPT	Material axes option (see MAT_*OPTION* TROPIC_ELASTIC for more complete description). AOPT.EQ.0.0 locally orthotropic with material axes determined by element nodes 1, 2 and 4, as with *DEFINE_ COORDINATE_NODES. AOPT.EQ.2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINED_ COORDINATE_VECTOR. AOPT.EQ.3.0: locally orthotropic material axes determined by rotating the material axes abut the element normal by an angle BETA, from a line in the plane of the element defined by the cross product of the vector V with the element normal. AOPT.LT.0.0: the absolute value of AOPT is coordinate system ID (CID on *DEFINE_COORDINATE_NODES, *DEFINE_ COORDINATE_SYSTEM, or *DEFINE_COORDINATE_ VECTOR). Available with the R3 release of 971 and later.
C12	Material parameter. If parameter identification (FIT=1.0) is turned on C12 is not used.
C13	Material parameter. If parameter identification (FIT=1.0) is turned on C13=0.0
C23	Material parameter. If parameter identification (FIT=1.0) is turned on C23=0.0
C44	Material parameter (see card 5 pos.8) FIT.EQ.1.0 or EQ.2.0: yield stress for the balanced biaxial tension test. FIT.EQ.0.0: material parameter c44
A1-A3	Components of vector **a** for AOPT=2.0
V1-V3	Components of vector **v** for AOPT=3.0
D1-D3	Components of vector **d** for AOPT=2.0
BETA	Material angle in degrees for AOPT=3.0. NOTE, may be overridden on the element card, see *ELEMENT_SHELL_BETA

VARIABLE	DESCRIPTION
FIT	Flag for parameter identification algorithm: FIT.EQ.0.0: No parameter identification routine is used. The variables K, C11, C22, C33, C44, C12, C13 and C23 are interpreted as material parameters. FIT.EQ.1.0: Parameter fit is used. The variables C11, C22, C33, C44 and K are interpreted as yield stresses in the 00, 45, 90 degree directions, the balanced biaxial tension and the 00 degree compression, respectively. NOTE: it is recommended to always check the d3hsp file to see the fitted parameters before complex jobs are submitted. FIT.EQ.2.0: Same as EQ.1.0 but also produce contour plots of the yield surface. For each material three LS-PREPOST ready xy-data files are created; Contour1_x, Contour2_x and Contour3_x where x equal the material numbers.

*MAT_VISCOELASTIC_LOOSE_FABRIC

This is Material Type 234 developed by Ivanov and Tabiei [2004]. The model is a mechanism incorporating the crimping of the fibers as well as the trellising with reorientation of the yarns and the locking phenomenon observed in loose fabric. The equilibrium of the mechanism allows the straightening of the fibers depending on the fiber tension. The contact force at the fiber cross over point determines the rotational friction dissipating a part of the impact energy. The stress-strain relationship is viscoelastic based on a three-element model. The failure of the fibers is strain rate dependent. *DAMPING_MASS is recommended to be used in conjunction with this material model. This material is valid for modeling the elastic and viscoelastic response of loose fabric used in body armor, blade containments, and airbags.

Card 1

Variable	MID	RO	E1	E2	G12	EU	THL	THI
Type	A8	F	F	F	F	F	F	F

Card 2

Variable	TA	W	s	T	H	S	EKA	EUA
Type	F	F	F	F	F	F	F	F

Card 3

Variable	VMB	C	G23	EKB	AOPT			
Type	F	F	F	F	F			

Card 4

Variable	Xp	Yp	Zp	A1	A2	A3		
Type	F	F	F	F	F	F		

Card 5

Variable	V1	V2	V3	D1	D2	D3		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
E1	E_1, Young's modulus in the yarn axial-direction.
E2	E_2, Young's modulus in the yarn transverse-direction.
G12	G_{12}, Shear modulus of the yarns.
EU	Ultimate strain at failure.
THL	Yarn locking angle.
THI	Initial brade angle.
TA	Transition angle to locking.
W	Fiber width.
S	Span between the fibers.
T	Real fiber thickness.
H	Effective fiber thickness.
S	Fiber cross-sectional area.
EKA	Elastic constant of element "a".
EUA	Ultimate strain of element "a".
VMB	Damping coefficient of element "b".
C	Coefficient of friction between the fibers.

VARIABLE	DESCRIPTION
G23	transverse shear modulus.
Ekb	Elastic constant of element "b"
AOPT	Material axis option

Remarks:

The parameters of the Representative Volume Cell (RVC) are: the yarn span, *s*, the fabric thickness, *t*, the yarn width, *w*, and the yarn cross-sectional area, *A*. The initially orthogonal yarns (see Fig. 2a) are free to rotate (see Fig. 2b) up to some angle and after that the lateral contact between the yarns causes the locking of the trellis mechanism and the packing of the yarns (see Fig. 2c).The minimum braid angle, $\theta_{\min}$, can be calculated from the geometry and the architecture of the fabric material having the yarn width, *w*, and the span between the yarns, *s*:

$$\sin(2\theta_{\min}) = \frac{w}{s}$$

The other constrain angles as the locking range angle, θ_{lock}, and the maximum braid angle, $\theta_{\max}$, (see Fig) are easy to be determined then:

$$\theta_{lock} = 45^\circ - \theta_{\min} \ , \quad \theta_{\max} = 45^\circ + \theta_{lock}$$

The material behavior of the yarn can be simply described by a combination of one Maxwell element without the dashpot and one Kelvin-Voigt element. The 1-D model of viscoelasticity is shown in the following figure. The differential equation of viscoelasticity of the yarns can be derived from the model equilibrium as in the following equation:

$$(K_a + K_b)\sigma + \mu_b\dot{\sigma} = K_a K_b \varepsilon + \mu_b K_a \dot{\varepsilon}$$

The input parameters for the viscoelasticity model of the material are only the static Young's modulus E_1, the Hookian spring coefficient (EKA) K_a, the viscosity coefficient (VMB) μ_b, the static ultimate strain (EU) $\varepsilon_{\max}$, and the Hookian spring ultimate strain (EUA) $\varepsilon_{a\max}$. The other parameters can be obtained as follows:

$$K_b = \frac{K_a E_1}{K_a - E_1}$$

$$\varepsilon_{b\max} = \frac{K_a - E_1}{K_a}\varepsilon_{\max}$$

Applying the Eq. (18) for the fill and the warp yarns, we obtain the stress increments in the yarns, $\Delta\sigma_f$ and $\Delta\sigma_w$,. The stress in the yarns is updated for the next time step:

$$\sigma_f^{(n+1)} = \sigma_f^{(n)} + \Delta\sigma_f^{(n)}\,, \quad \sigma_w^{(n+1)} = \sigma_w^{(n)} + \Delta\sigma_w^{(n)} \tag{38}$$

We can imagine that the RVC is smeared to the parallelepiped in order to transform the stress acting on the yarn cross-section to the stress acting on the element wall. The thickness of the membrane shell element used should be equal to the effective thickness, t_e , that can be found by dividing the areal density of the fabric by its mass density. The in-plane stress components acting on the RVC walls in the material direction of the yarns are calculated as follows for the fill and warp directions:

$$\sigma_{f11}^{(n+1)} = \frac{2\sigma_f^{(n+1)} S}{s t_e}\,, \quad \sigma_{w11}^{(n+1)} = \frac{2\sigma_w^{(n+1)} S}{s t_e}$$

$$\sigma_{f22}^{(n+1)} = \sigma_{f22}^{(n)} + \alpha E_2 \Delta\varepsilon_{f22}^{(n)}\,, \quad \sigma_{w22}^{(n+1)} = \sigma_{w22}^{(n)} + \alpha E_2 \Delta\varepsilon_{w22}^{(n)}$$

$$\sigma_{f12}^{(n+1)} = \sigma_{f12}^{(n)} + \alpha G_{12} \Delta\varepsilon_{f12}^{(n)}\,, \quad \sigma_{w12}^{(n+1)} = \sigma_{w12}^{(n)} + \alpha G_{12} \Delta\varepsilon_{w12}^{(n)}$$

where E_2 is the transverse Young's modulus of the yarns, G_{12} is the longitudinal shear modulus, and α is the lateral contact factor. The lateral contact factor is zero when the trellis mechanism is open and unity if the mechanism is locked with full lateral contact between the yarns. There is a transition range, $\Delta\theta$ *(TA)*, of the average braid angle θ in which the lateral contact factor, α, is a linear function of the average braid angle. The graph of the function $\alpha(\theta)$ is shown in Fig. 4.

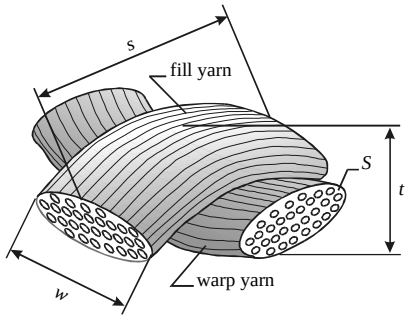


Fig. 1. Representative Volume Cell (RVC) of the model

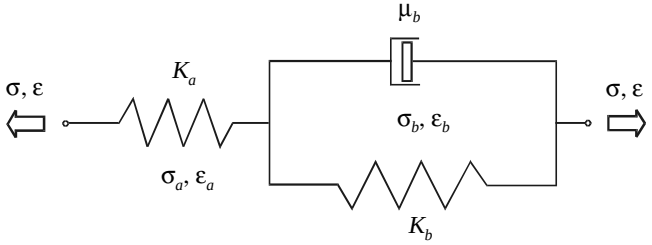


Fig. 2. Three-element viscoelasticity model

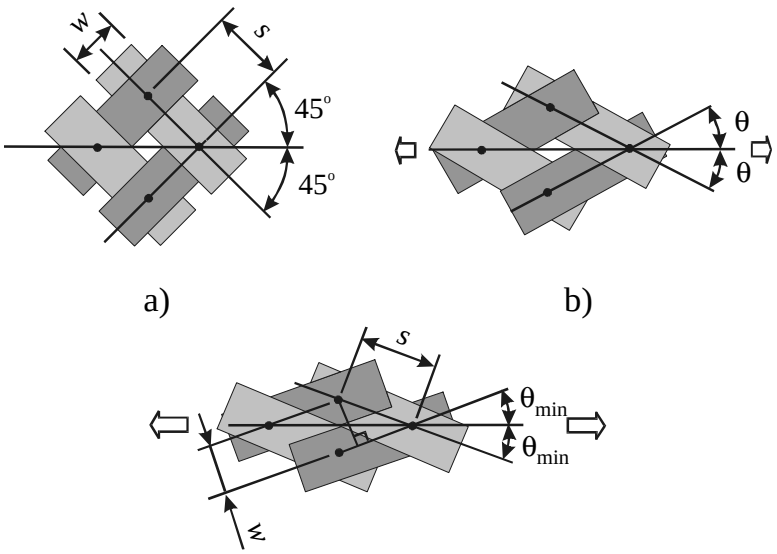


Fig. 3. Plain woven fabric as trellis mechanism: a) initial state; b) slightly stretched in bias direction; c) stretched to locking.

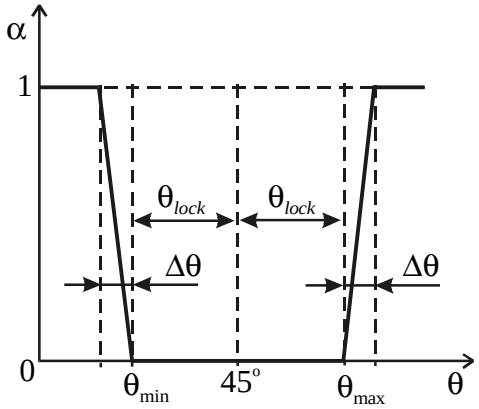


Fig. 4. The lateral contact factor as a function of average braid angle θ.

*MAT_MICROMECHANICS_DRY_FABRIC

This is Material Type 235 developed by Tabiei and Ivanov [2001]. The material model derivation utilizes the micro-mechanical approach and the homogenization technique usually used in composite material models. The model accounts for reorientation of the yarns and the fabric architecture. The behavior of the flexible fabric material is achieved by discounting the shear moduli of the material in free state, which allows the simulation of the trellis mechanism before packing the yarns. This material is valid for modeling the elastic response of loose fabric used in inflatable structures, parachutes, body armor, blade containments, and airbags.

Card 1

1	2	3	4	5	6	7	8
MID	RO	E1	E2	G12	G23	V12	V23
A8	F	F	F	F	F	F	F

Card 2

Xt	THL	THI	BFI	BWI	DSCF	CNST	ATLR
F	F	F	F	F	F	F	F

Card 3

VMB	VME	TRS					
F	F	F					

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.

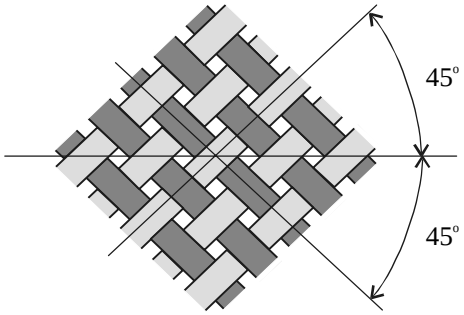
VARIABLE	DESCRIPTION
E1	E_1, Young's modulus of the yarn in axial-direction.
E2	E_2, Young's modulus of the yarn in transverse-direction.
G12	G_{12}, shear modulus of the yarns.
G23	G_{23}, transverse shear modulus of the yarns.
V12	Poisson's ratio.
V23	Transverse Poisson's ratio.
Xt	Ultimate strength at failure (not used).
THL	Yarn locking angle.
THI	Initial brade angle.
BFI	Initial undulation angle in fill direction.
BWI	Initial undulation angle in warp direction.
DSCF	Discount factor
CNST	Reorientation damping constant
ATLR	Angle tolerance for locking
VME	Viscous modulus for normal strain rate
VMS	Viscous modulus for shear strain rate
TRS	Transverse shear modulus of the fabric layer

Remarks:

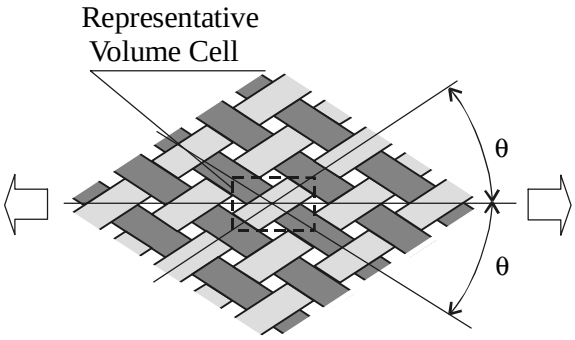
The Representative Volume Cell (RVC) approach is utilized in the micro-mechanical model development. The direction of the yarn in each sub-cell is determined by two angles – the braid angle, θ *(the initial braid angle is 45 degrees)*, and the undulation angle of the yarn, which is different for the fill and warp-yarns, β_f and β_w (the initial undulations are normal few degrees), respectively. The starting point for the homogenization of the material properties is the determination of the yarn stiffness matrices.

$$[C'] = [S']^{-1} = \begin{bmatrix} \frac{1}{E_1} & -\frac{\nu_{12}}{E_1} & -\frac{\nu_{12}}{E_1} & 0 & 0 & 0 \\ -\frac{\nu_{12}}{E_1} & \frac{1}{E_2} & -\frac{\nu_{23}}{E_2} & 0 & 0 & 0 \\ -\frac{\nu_{12}}{E_1} & -\frac{\nu_{23}}{E_2} & \frac{1}{E_2} & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1}{\mu G_{12}} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1}{\mu G_{23}} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1}{\mu G_{12}} \end{bmatrix}^{-1}$$

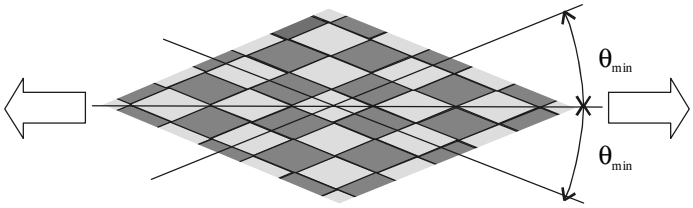
where $E_1, E_2, \nu_{12}, \nu_{23}, G_{12}$ and G_{23} are Young's moduli, Poisson's ratios, and the shear moduli of the yarn material, respectively. μ is a discount factor, which is function of the braid angle, θ, and has value between μ_0 and 1 as shown in the next figure. Initially, in free stress state, the discount factor is a small value (DSCF=μ_0 << 1) and the material has very small resistance to shear deformation if any. When the locking occurs, the fabric yarns are packed and they behave like elastic media. The discount factor is unity as showen in the next figure. The micro-mechanical model is developed to account for the reorientation of the yarns up to the locking angle. The locking angle, θ_{lock} , can be obtained from the yarn width and the spacing parameter of the fabric using simple geometrical relationship. The transition range, $\Delta\theta$ (angle tolerance for locking), can be chosen to be as small as possible, but big enough to prevent high frequency oscillations in transition to compacted state and depends on the range to the locking angle and the dynamics of the simulated problem. Reorientation damping constant is defined to damp some of the high frequency oscillations. A simple rate effect is added by defining the viscous modulus for normal or shear strain rate (VMB*$\dot{\varepsilon}_{11or22}$ for normal components and VMS*$\dot{\varepsilon}_{12}$ for the shear components).



a) Free state



b) Stretched



c) Compacted

Fig. 1. Plain-woven fabric interlacing pattern.

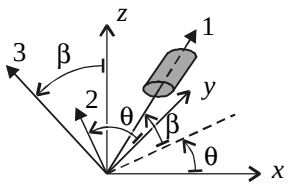


Fig. 2. Yarn orientation.

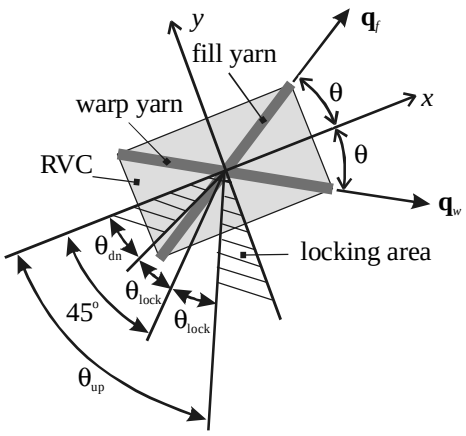


Fig. 3. Locking angles.

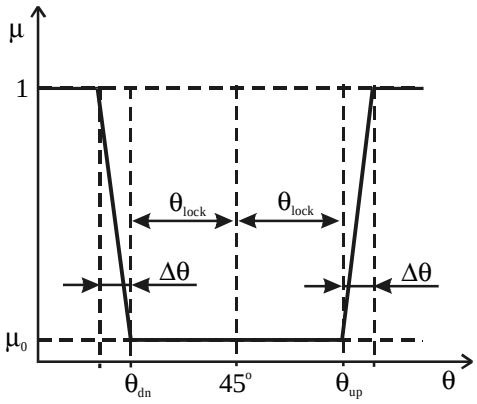


Fig. 4. Discount factor as a function of braid angle, θ.

*MAT_SCC_ON_RCC

This is Material Type 236 developed by Carney, Lee, Goldberg, and Santhanam [2007]. This model simulates silicon carbide coating on Reinforced Carbon-Carbon (RCC), a ceramic matrix and is based upon a quasi-orthotropic, linear-elastic, plane-stress model. Additional constitutive model attributes include a simple (i.e. non-damage model based) option that can model the tension crack requirement: a "stress-cutoff" in tension. This option satisfies the tension crack requirements by limiting the stress in tension but not compression, and having the tensile "yielding" (i.e. the stress-cutoff) be fully recoverable – not plasticity or damage based.

Card 1

1	2	3	4	5	6	7	8
MID	RO	E0	E1	E2	E3	E4	E5
A8	F	F	F	F	F	F	F

Card 2

PR	G	G_SCL	TSL	EPS_TAN			
F	F	F	F	F			

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
E0	E_0, See Remarks below.
E1	E_1, See Remarks below.
E2	E_2, See Remarks below.
E3	E_3, See Remarks below.
E4	E_4, See Remarks below.
E5	E_5, Young's modulus of the yarn in transverse-direction.
PR	Poisson's ratio.

VARIABLE	DESCRIPTION
G	Shear modulus
G_SCL	Shear modulus multiplier (default=1.0).
TSL	Tensile limit stress
EPS_TAN	Strain at which E=tangent to the polynomial curve.

Remarks:

This model for the silicon carbide coating on RCC is based upon a quasi-orthotropic, linear-elastic, plane-stress model, given by:

$$\begin{Bmatrix} \sigma_1 \\ \sigma_2 \\ \tau_{12} \end{Bmatrix} = \begin{bmatrix} \frac{E}{1-\nu^2} & \frac{\nu E}{1-\nu^2} & 0 \\ \frac{\nu E}{1-\nu^2} & \frac{E}{1-\nu^2} & 0 \\ 0 & 0 & G_{12} \end{bmatrix} \begin{Bmatrix} \varepsilon_1 \\ \varepsilon_2 \\ \gamma_{12} \end{Bmatrix}$$

Additional constitutive model requirements include a simple (i.e. non-damage model based) option that can model the tension crack requirement: a “stress-cutoff” in tension. This option satisfies the tension crack requirements by limiting the stress in tension but not compression, and having the tensile “yielding” (i.e. the stress-cutoff) be fully recoverable – not plasticity or damage based.

The tension stress-cutoff separately resets the stress to a limit value when it is exceeded in each of the two principal directions. There is also a strain-based memory criterion that ensures unloading follows the same path as loading: the “memory criterion” is the tension stress assuming that no stress cutoffs were in effect. In this way, when the memory criterion exceeds the user-specified cutoff stress, the actual stress will be set to that value. When the element unloads and the memory criterion falls back below the stress cutoff, normal behavior resumes. Using this criterion is a simple way to ensure that unloading does not result in any hysteresis. The cutoff criterion cannot be based on an effective stress value because effective stress does not discriminate between tension and compression, and also includes shear. This means that the in plane, 1- and 2- directions must be modeled as independent to use the stress cutoff. Because the Poisson’s ratio is not zero, this assumption is not true for cracks that may arbitrarily lie along any direction. However, careful examination of damaged RCC shows that generally, the surface cracks do tend to lie in the fabric directions as seen in Figure 3.2, meaning that cracks tend to open in the 1- or the 2- direction independently. So the assumption of directional independence for tension cracks may be appropriate for the coating because of this observed orthotropy.

The quasi-orthotropic, linear-elastic, plane-stress model with tension stress cutoff (to simulate tension cracks) can model the as-fabricated coating properties, which do not show nonlinearities, but not the non-linear response of the flight-degraded material. Explicit finite element analysis (FEA) lends itself to *nonlinear-elastic* stress-strain relationship instead of linear-elastic. Thus, instead of $\underline{\sigma} = \mathbf{E} \cdot \underline{\varepsilon}$, the modulus will be defined as a function of some effective strain quantity, or $\underline{\sigma} = \mathbf{E}(\varepsilon_{eff}) \cdot \underline{\varepsilon}$, even though it is uncertain, from the available data, whether or not the coating response is completely nonlinear-elastic, and does not include some damage mechanism.

This nonlinear-elastic model cannot be implemented into a closed form solution or into an implicit solver; however, for explicit FEA such as is used for LS-DYNA impact analysis, the modulus can be adjusted at each time step to a higher or lower value as desired. In order to model the desired S-shape response curve of flight-degraded RCC coating, a function of strain that replicates the desired response must be found. It is assumed that the nonlinearities in the material are recoverable (elastic) and that the modulus is communicative between the 1- and 2-directions (going against the tension-crack assumption that the two directions do not interact). Sometimes stability can be a problem for this type of nonlinearity modeling, however, stability was not found to be a problem with the material constants used for the coating.

The von Mises strain is selected for the effective strain definition as it couples the 3-dimensional loading but reduces to uniaxial data, so that the desired uniaxial compressive response can be reproduced. So,

$$\varepsilon_{eff} = \frac{1}{\sqrt{2}} \frac{1}{1+\nu} \sqrt{(\varepsilon_1 - \varepsilon_2)^2 + (\varepsilon_2 - \varepsilon_3)^2 + (\varepsilon_1 - \varepsilon_3)^2 + 3\gamma_{12}^2}$$

where for a 2-D, isotropic shell element case, the z-direction strain is given by:

$$\varepsilon_3 = \frac{-\nu}{1-\nu}(\varepsilon_1 + \varepsilon_2)$$

The function for modulus is implemented as an arbitrary 5th order polynomial:

$$E(\varepsilon_{eff}) = A_0 \cdot \varepsilon_{eff}^0 + A_1 \cdot \varepsilon_{eff}^1 + \ldots + A_5 \cdot \varepsilon_{eff}^5$$

In the case of as-fabricated material the first coefficient (A_0) is simply the modulus E, and the other coefficients ($A_{n>0}$) are zero, reducing to a 0th order polynomial, or linear. To match the degraded stress-strain compression curve, a higher order polynomial is needed. Six conditions on stress were used (stress and its derivative at beginning, middle, and end of the curve) to obtain a 5th order polynomial, and then the derivative of that equation was taken to obtain modulus as a function of strain, yielding a 4th order polynomial that represents the degraded coating modulus vs. strain curve.

For values of strain which exceed the failure strain observed in the laminate compression tests, the higher order polynomial will no longer match the test data. Therefore, after a specified effective-strain, representing failure, the modulus is defined to be the tangent of the polynomial

curve. As a result, the stress/strain response has a continuous derivative, which aids in avoiding numerical instabilities. The test data does not clearly define the failure strain of the coating, but in the impact test it appears that the coating has a higher compressive failure strain in bending than the laminate failure strain.

The two dominant modes of loading which cause coating loss on the impact side of the RCC (the front-side) are in-plane compression and transverse shear. The in-plane compression is measured by the peak out of plane tensile strain, ε_3. As there is no direct loading of a shell element in this direction, ε_3 is computed through Poisson's relation $\varepsilon_3 = \frac{-\nu}{1-\nu}(\varepsilon_1 + \varepsilon_2)$. When ε_3 is tensile, it implies that the average of ε_1 and ε_2 is compressive. This failure mode will likely dominate when the RCC undergoes large bending, putting the front-side coating in high compressive strains. It is expected that a transverse shear failure mode will dominate when the debris source is very hard or very fast. By definition, the shell element cannot give a precise account of the transverse shear throughout the RCC's thickness. However, the Belytschko-Tsay shell element formulation in LS-DYNA has a first-order approximation of transverse shear that is based on the out-of-plane nodal displacements and rotations that should suffice to give a qualitative evaluation of the transverse shear. By this formulation, the transverse shear is constant through the entire shell thickness and thus violates surface-traction conditions. The constitutive model implementation records the peak value of the tensile out-of-plane strain (ε_3) and peak root-mean-sum transverse-shear: $\sqrt{\varepsilon_{13}^2 + \varepsilon_{23}^2}$.

*MAT_PML_HYSTERETIC

This is Material Type 237. This is a perfectly-matched layer (PML) material with a Biot linear hysteretic constitutive law, to be used in a wave-absorbing layer adjacent to a Biot hysteretic material (*MAT_BIOT_HYSTERETIC) in order to simulate wave propagation in an unbounded medium with material damping. This material is the visco-elastic counterpart of the elastic PML material (*MAT_PML_ELASTIC). See the Remarks sections of *MAT_PML_ELASTIC (*MAT_230) and *MAT_BIOT_HYSTERETIC (*MAT_232) for further details.

Card	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	ZT	FD		
Type	A8	F	F	F	F	F		
Default	none	none	none	none	0.0	3.25		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.
ZT	Damping ratio
FD	Dominant excitation frequency in Hz

*MAT_PERT_PIECEWISE_LINEAR_PLASTICITY

This is Material Type 238. It is a duplicate of Material Type 24 (*MAT_PIECEWISE_LINEAR_PLASTICITY) modified for use with *PERTURBATION_MATERIAL and solid elements in an explicit analysis. It should give exactly the same values as the original material, if used exactly the same. It exists as a separate material type because of the speed penalty (an approximately 10% increase in the overall execution time) associated with the use of a material perturbation.

See Material Type 24 (*MAT_PIECEWISE_LINEAR_PLASTICITY) for a description of the material parameters. All of the documentation for Material Type 24 applies. Recommend practice is to first create the input deck using Material Type 24. Additionally, the CMP variable in the *PERTURBATION_MATERIAL must be set to affect a specific variables in the MAT_238 definition as defined in the following table; for example, CMP=5 will perturb the yield stress.

***PERTURBATION_MATERIAL CMP value**	**Material variable**
3	E
5	SIGY
6	ETAN
7	FAIL

*MAT_COHESIVE_MIXED_MODE_ELASTOPLASTIC_RATE

This is Material Type 240. This model is a rate-dependent, elastic-ideally plastic cohesive zone model. It includes a tri-linear traction-separation law with a quadratic yield and damage initiation criterion in mixed-mode loading, while the damage evolution is governed by a power-law formulation. It can be used with solid element types 19 and 20, and is not available for other solid element formulations. See the remarks after *SECTION_SOLID for a description of element types 19 and 20.

Card 1

1	2	3	4	5	6	7	8
Variable	MID	RO	ROFLG	INTFAIL	EMOD	GMOD	THICK	OUTPUT
Type	A8	F	F	F	F	F	F	F

Card 2

Variable	G1C_0	G1C_INF	EDOT_G1	T0	T1	EDOT_T	FG1	
Type	F	F	F	F	F	F	F	

Card 3

Variable	G2C_0	G2C_INF	EDOT_G2	S0	S1	EDOT_S	FG2	
Type	F	F	F	F	F	F	F	

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
ROFLG	Flag for whether density is specified per unit area or volume. ROFLG=0 specified density per unit volume (default), and ROFLG=1 specifies the density is per unit area for controlling the mass of cohesive elements with an initial volume of zero.

VARIABLE	DESCRIPTION
EMOD	The Young's modulus of the material
GMOD	The shear modulus of the material
THICK	GT.0.0: Cohesive thickness LE.0.0: Initial thickness is calculated from nodal coordinates
OUTPUT	Time interval at which output is written into FORT.11-File
G1C_0	GT 0.0: Energy release rate G_{IC} in Mode I LE. 0.0: Lower bound value of rate-dependent G_{IC}
G1C_INF	Upper bound value of rate-dependent G_{IC} (only considered if G1C_0<0)
EDOT_G1	Equivalent strain rate at yield initiation to describe the rate dependency of G_{IC} (only considered if G1C_0<0)
T0	GT.0.0: Yield stress in Mode I LT.0.0: Rate-dependency is considered, Parameter T0
T1	Parameter T1, only considered if T0 < 0: GT.0.0: Quadratic logarithmic model LT.0.0: Linear logarithmic model
EDOT_T	Equivalent strain rate at yield initiation to describe the rate dependency of the yield stress in Mode I (only considered if T0<0)
FG1	Parameter f_{G1} to describe the tri-linear shape of the traction-separation law in Mode I
G2C_0	GT.0.0: Energy release rate G_{IIC} in Mode II LE.0.0: Lower bound value of rate-dependent G_{IIC}
G2C_INF	Upper bound value of G_{IIC} (only considered if G2C_0<0)
EDOT_G2	Equivalent strain rate at yield initiation to describe the rate dependency of G_{IIC} (only considered if G2C_0<0)
S0	GT.0.0: Yield stress in Mode II LT.0.0: Rate-dependency is considered, Parameter S0
S1	Parameter S1, only considered if S0<0: GT.0.0: Quadratic logarithmic model is applied LT.0.0: Linear logarithmic model is applied
EDOT_S	Equivalent strain rate at yield initiation to describe the rate dependency of the yield stress in Mode II (only considered if S0<0)

VARIABLE	DESCRIPTION
FG2	Parameter f_{G2} to describe the tri-linear shape of the traction-separation law in Mode II

Remarks:

The model is a tri-linear elastic-ideally plastic Cohesive Zone Model, which was developed by Marzi et al. [2009]. It looks similar to *MAT_185, but considers effects of plasticity and rate-dependency. Since the entire separation at failure is plastic, no brittle fracture behavior can be modeled with this material type.

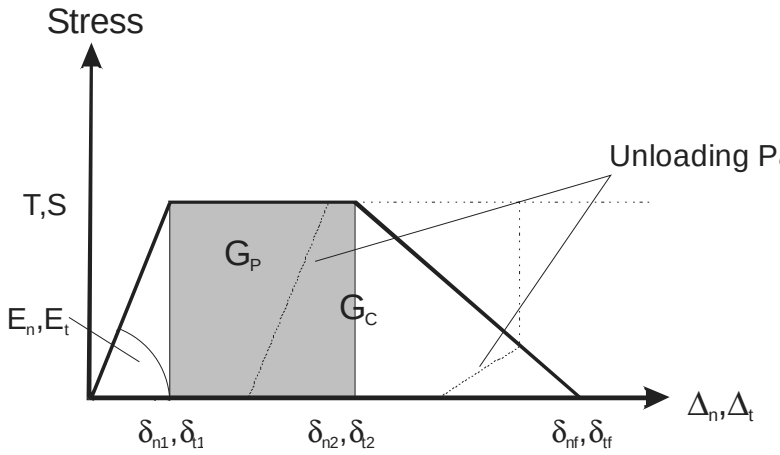


Figure 240.1. Trilinear traction-separation law

The separations Δ_n in normal (peel) and Δ_t in tangential (shear) direction are calculated from the element's separations in the integration points,

$$\Delta_n = \langle u_n \rangle \text{ and } \Delta_t = \sqrt{{u_{t1}}^2 + {u_{t2}}^2}\,,\ \langle x \rangle = \begin{cases} x & \text{if } x > 0 \\ 0 & \text{else} \end{cases}.$$

u_n, u_{t1} and u_{t2} are the separations in normal and in the both tangential directions of the element coordinate system. The total (mixed-mode) separation Δ_m is determined by

$$\Delta_m = \sqrt{{\Delta_n}^2 + {\Delta_t}^2}\,.$$

The initial stiffnesses in both modes are calculated from the elastic Young's and shear modulus,

$$E_n = EMODUL / THICK \text{ and } E_t = GMODUL / THICK\,,$$

where $THICK$, the element's thickness, is a user defined value if $THICK > 0$, otherwise it is calculated as distance between the initial positions of the element's corner nodes (Nodes 1-5, 2-6, 3-7 and 4-8, respectively).

While the total energy under the traction-separation law is given by G_C, one further parameter is needed to describe the exact shape of the tri-linear material model. If the area (energy) under the

constant stress (plateau) region is denoted G_P (see Figure 240.1), a parameter f_G defines the shape of the traction-separation law,

$$0 \le f_{G1} = \frac{G_{I,P}}{G_{IC}} < 1 - \frac{T^2}{2G_{IC}E_n} < 1 \quad \text{for mode I loading and}$$

$$0 \le f_{G2} = \frac{G_{II,P}}{G_{IIC}} < 1 - \frac{S^2}{2G_{IIC}E_t} < 1 \quad \text{for mode II.}$$

While f_{G1} and f_{G2} are always constant values, T, S, G_{IC} and G_{IIC} may be chosen as functions of an equivalent strain rate $\dot{\varepsilon}_{eq}$, which is evaluated by

$$\dot{\varepsilon}_{eq} = \frac{\sqrt{\dot{u}_n^{\,2} + \dot{u}_{t1}^{\,2} + \dot{u}_{t2}^{\,2}}}{THICK},$$

where $\dot{u}_n, \dot{u}_{t1}$ and $\dot{u}_{t2}$ are the velocities corresponding to the separations u_n, u_{t1} and u_{t2}.

For the yield stresses, two rate dependent formulations are implemented:

1. A quadratic logarithmic function:

   $T(\dot{\varepsilon}_{eq}) = |T0| + |T1| \left\langle \ln \frac{\dot{\varepsilon}_{eq}}{EDOT\_T} \right\rangle^2$ in Mode I, if T0 < 0 and T1 >0, and

   $S(\dot{\varepsilon}_{eq}) = |S0| + |S1| \left\langle \ln \frac{\dot{\varepsilon}_{eq}}{EDOT\_S} \right\rangle^2$ in Mode II, if S0 < 0 and S1 >0.

2. A linear logarithmic function:

   $T(\dot{\varepsilon}_{eq}) = |T0| + |T1| \left\langle \ln \frac{\dot{\varepsilon}_{eq}}{EDOT\_T} \right\rangle$ in Mode I, if T0 < 0 and T1 <0, and

   $S(\dot{\varepsilon}_{eq}) = |S0| + |S1| \left\langle \ln \frac{\dot{\varepsilon}_{eq}}{EDOT\_S} \right\rangle$ in Mode II, if S0 < 0 and S1 <0.

Alternatively, T and S can be chosen as constant values:

$T(\dot{\varepsilon}_{eq}) = T0$ in Mode I, if T0 > 0, and $S(\dot{\varepsilon}_{eq}) = S0$ in Mode II, if S0 > 0.

The rate-dependency of the fracture energies are given by

$$G_{IC}(\dot{\varepsilon}_{eq}) = |G1C\_0| + \left(G1C\_INF - |G1C\_0|\right)\exp\left(-\frac{EDOT\_G1}{\dot{\varepsilon}_{eq}}\right), \text{ if G1C\_0 < 0, and}$$

$$G_{IIC}(\dot{\varepsilon}_{eq}) = |G2C\_0| + \left(G2C\_INF - |G2C\_0|\right)\exp\left(-\frac{EDOT\_G2}{\dot{\varepsilon}_{eq}}\right), \text{ if G2C\_0 < 0.}$$

If positive values are chosen for G1C_0 or G2C_0, no rate-dependency is considered for this parameter and its value remains constant as specified by the user.

It should be noticed, that the equivalent strain rate $\dot{\varepsilon}_{eq}$ is updated until $\Delta_m > \delta_{m1}$, then the model behavior depends on the equivalent strain rate at yield initiation.

Having defined the parameters describing the single modes, the mixed-mode behavior is formulated by quadratic initiation criteria for both yield stress and damage initiation, while the damage evolution follows a Power-Law.

Due to reasons of readability, the following simplifications are made,

$$T = T(\dot{\varepsilon}_{eq}), S = S(\dot{\varepsilon}_{eq}), G_{IC} = G_{IC}(\dot{\varepsilon}_{eq}) \text{ and } G_{IIC} = G_{IIC}(\dot{\varepsilon}_{eq}).$$

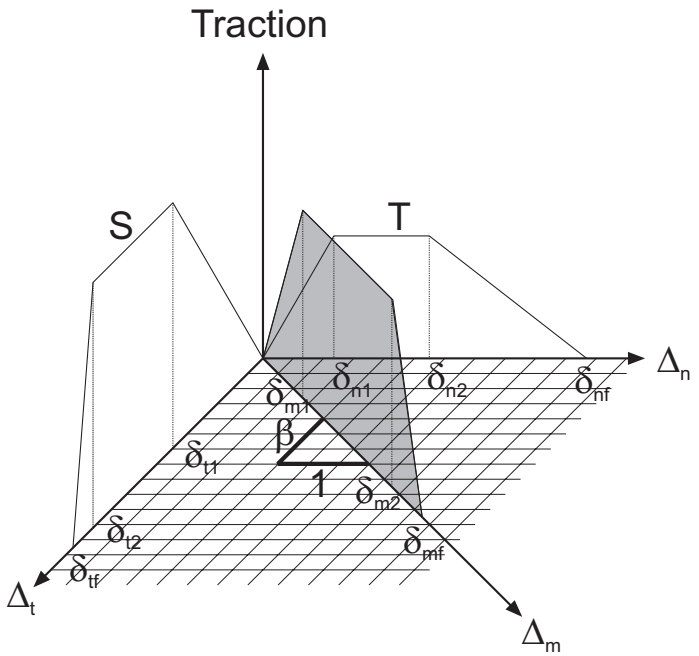


Figure 240.2. Trilinear, mixed-mode traction-separation law

The mixed-mode yield initiation displacement δ_{m1} is defined as

$$\delta_{m1} = \delta_{n1}\delta_{t1}\sqrt{\frac{1+\beta^2}{\delta_{t1}{}^2 + (\beta\delta_{n1})^2}},$$

where $\delta_{n1} = \frac{T}{E_n}$ and $\delta_{t1} = \frac{S}{E_t}$ are the single-mode yield initiation displacements and $\beta = \frac{\delta_{t1}}{\delta_{n1}}$ is the mixed-mode ratio. Analog to the yield initiation, the damage initiation displacement δ_{m2} is defined:

$$\delta_{m2} = \delta_{n2}\delta_{t2}\sqrt{\frac{1+\beta^2}{\delta_{t2}{}^2 + (\beta\delta_{n2})^2}}, \text{ with } \delta_{n2} = \delta_{n1} + \frac{f_{G1}G_{IC}}{T} \text{ and } \delta_{t2} = \delta_{t1} + \frac{f_{G2}G_{IIC}}{S}.$$

With $\gamma = \arccos\left(\frac{\langle u_n \rangle}{\Delta_m}\right)$, the ultimate (failure) displacement δ_{mf} can be written,

$$\delta_{mf} = \frac{\delta_{m1}(\delta_{m1}-\delta_{m2})E_n G_{IIC}\cos^2\gamma + G_{IC}\left(2G_{IIC}+\delta_{m1}(\delta_{m1}-\delta_{m2})E_t\sin^2\gamma\right)}{\delta_{m1}\left(E_n G_{IIC}\cos^2\gamma + E_t G_{IC}\sin^2\gamma\right)}.$$

This formulation describes a power-law damage evolution with an exponent $\eta = 1.0$ (see *MAT_138).

After the shape of the mixed-mode traction-separation law has been determined by δ_{m1}, δ_{m2} and δ_{mf}, the plastic separation in each element direction, $u_{n,P}, u_{t1,P}$ and $u_{t2,P}$ can be calculated. The plastic separation in peel direction is given by

$$u_{n,P} = \max(u_{n,P,\Delta t-1}, u_n - \delta_{m1}\sin\gamma, 0).$$

In shear direction, a shear yield separation $\delta_{t,y}$,

$$\delta_{t,y} = \sqrt{(u_{t1}-u_{t1,P,\Delta t-1})^2 + (u_{t2}-u_{t2,P,\Delta t-1})^2},$$

is defined. If $\delta_{t,y} > \delta_{m1}\sin\gamma$, the plastic shear separations in the element coordinate system are updated,

$$u_{t1,P} = u_{t1,P,\Delta t-1} + u_{t1} - u_{t1,\Delta t-1}, \quad \text{and} \quad u_{t2,P} = u_{t2,P,\Delta t-1} + u_{t2} - u_{t2,\Delta t-1}.$$

In the formulas above, $\Delta t - 1$ indicates the individual value from the last time increment. In case $\Delta_m > \delta_{m2}$, the damage initiation criterion is satisfied and a damage variable D increases monotonically,

$$D = \max\left(\frac{\Delta_m - \delta_{m2}}{\delta_{mf} - \delta_{m2}}, D_{\Delta t-1}, 0\right).$$

When $\Delta_m > \delta_{mf}$, complete damage ($D = 1$) is reached and the element fails in the corresponding integration point.

Finally, the peel and the shear stresses in element directions are calculated,

$$\sigma_{t1} = E_t(1-D)(u_{t1} - u_{t1,P}), \quad \text{and} \quad \sigma_{t2} = E_t(1-D)(u_{t2} - u_{t2P}).$$

In peel direction, no damage under pressure loads is considered,

$$\sigma_n = E_n(1-D)(u_n - u_{n,P}), \text{ if } u_n - u_{n,P} > 0 \quad \text{and} \quad \sigma_n = E_n(u_n - u_{n,P}) \text{ else.}$$

Reference:

S. Marzi, O. Hesebeck, M. Brede and F. Kleiner (2009), A Rate-Dependent, Elasto-Plastic Cohesive Zone Mixed-Mode Model for Crash Analysis of Adhesively Bonded Joints, In Proceeding: *7th European LS-DYNA Conference, Salzburg*

* MAT_PIECEWISE_LINEAR_PLASTIC_THERMAL

This is material type 255, an isotropic elastoplastic material with thermal properties. It can be used for both explicit and implicit analyses. Young's modulus and Poisson's ratio can depend on the temperature by defining two load curves. Moreover, the yield stress in tension and compression are given as load curves for different temperatures by using two tables. The thermal coefficient of expansion can be given as a constant ALPHA or as a load curve, see LALPHA at position 3 on card 2. A positive curve ID for LALPHA models the instantaneous thermal coefficient, whereas a negatives curve ID models the thermal coefficient relative to a reference temperature, TREF. The strain rate effects are modelled with the Cowper-Symonds rate model with the parameters C and P on card 1. Failure can be based on effective plastic strain or using the *MAT_ADD_EROSION keyword.

Card 1

1	2	3	4	5	6	7	8
MID	RO	E	PR	C	P	FAIL	TDEL
A8	F	F	F	F	F	F	F

Card 2

Variable	TABIDC	TABIDT	LALPHA					
Type	I	I	I					

Card 3

Variable	ALPHA	TREF						
Type	F	F						

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
E	Young's modulus: LT.0.0: |E| is the LCID for E versus temperature, GT.0.0: E is constant.

VARIABLE	DESCRIPTION
PR	Poisson's ratio. LT.0.0: |PR| is the LCID for Poisson's ratio versus temperature. GT.0.0: PR is constant
C	Strain rate parameter. See remark 1.
P	Strain rate parameter. See remark 1.
FAIL	Effective plastic strain when the material fails. Note that for solids the *MAT_ADD_EROSION can be used for additional failure criteria.
TDEL	A time step less then TDEL is not allowed. A step size less than TDEL trigger automatic element deletion. This option is ignored for implicit analyses.
TABIDC	Table ID for yield stress in compression, see remark 2.
TABIDT	Table ID for yield stress in tension, see remark 2.
LALPHA	Load curve ID for thermal expansion coefficient as a function of temperature. GT.0.0: the instantaneous thermal expansion coefficient based on the following formula: $d\varepsilon_{ij}^{thermal} = \alpha(T)\,dT\,\delta_{ij}$ LT.0.0: the thermal coefficient is defined relative a reference temperature TREF, such that the total thermal strain is given by: $\varepsilon_{ij}^{thermal} = \alpha(T)\left(T - T_{ref}\right)\delta_{ij}$ With this option active, ALPHA is ignored.
ALPHA	Coefficient of thermal expansion
TREF	Reference temperature, which is required if and only if LALPHA is given with a negative load curve ID.

Remarks:

1: The strain rate effect is modelled by using the Cowper and Symonds model which scales the yield stress according to the factor

$$1+\left(\frac{\dot{\varepsilon}}{C}\right)^{1/P}$$

where $\dot{\varepsilon} = \sqrt{\dot{\varepsilon}_{ij}\dot{\varepsilon}_{ij}}$ is the Euclidean norm of the total strain rate tensor.

2: The yield stresses versus effective plastic strains are given in two tables. One table for yield stresses in compression and another table for yield stresses in tension. The table indices consist of temperatures and at each temperature an unique yield stress curve must be defined. If the same yield stress should be used in both tension and compression, only one table needs to be defined and the same TABID is put in position 1 and 2 on card 2.

3: Two history variables are added to the d3plot file, the Young's modulus and the Poisson's ratio, respectively. They can be requested through the *DATABASE_EXTENT_BINARY keyword.

4. Nodal temperatures must be defined by using a coupled analysis or some other way to define the temperatures, such as **LOAD_THERMAL_VARIABLE* or **LOAD_THERMAL_LOAD_CURVE*.

* MAT_AMORPHOUS_SOLIDS_FINITE_STRAIN

This is material type 256, an isotropic elastic-viscoplastic material model intended to describe the behaviour of amorphous solids such as polymeric glasses. The model accurately captures the hardening-softening-hardening sequence and the Bauschinger effect experimentally observed at tensile loading and unloading respectively. The formulation is based on hyperelasticity and uses the multiplicative split of the deformation gradient F which makes it naturally suitable for both large rotations and large strains. Stress computations are performed in an intermediate configuration and are therefore preceded by a pull-back and followed by a push-forward. The model was originally developed by Anand and Gurtin [2003] and implemented for solid elements by Bonnaud and Faleskog [2008]

Card 1

Variable	1	2	3	4	5	6	7	8
	MID	RO	K	G	MR	LL	NU0	M
Type	A8	F	F	F	F	F	F	F

Card 2

Variable								
	ALPHA	H0	SCV	B	ECV	G0	S0	
Type	F	F	F	F	F	F	F	

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density
K	Bulk modulus

VARIABLE	DESCRIPTION
G	Shear modulus
MR	Kinematic hardening parameter: μ_R (see Eq.1)
LL	Kinematic hardening parameter: λ_L (see Eq.1)
NU0	Creep parameter: v_0 (see Eq.2)
M	Creep parameter: m (see Eq.2)
ALPHA	Creep parameter: α (see Eq.2)
H0	Isotropic hardening parameter: h_0 (see Eq.3-5)
SCV	Isotropic hardening parameter: s_{cv} (see Eq.3-5)
B	Isotropic hardening parameter: b (see Eq.3-5)
ECV	Isotropic hardening parameter: η_{cv} (see Eq.3-5)
G0	Isotropic hardening parameter: g_0 (see Eq.3-5)
S0	Isotropic hardening parameter: s_0 (see Eq.3-5)

Remarks:

1) Kinematic hardening gives rise to the second hardening occurrence in the hardening-softening-hardening sequence. The constants μ_R and λ_L enter the back stress μB (where B is the left Cauchy-Green deformation tensor) through the function μ according to:

$$\mu = \mu_R(\frac{\lambda_L}{3\lambda^p})L^{-1}(\frac{\lambda^p}{\lambda_L}) \qquad \text{Eq.1}$$

where $\lambda^p = \frac{1}{\sqrt{3}}\sqrt{\text{tr}(B^p)}$, B^p is the plastic part of the left Cauchy-Green deformation tensor and where L is the Langevin function defined by $L(X)= \coth(X) - X^{-1}$

2) This material model assumes plastic incompressibility. Nevertheless in order to account for the different behaviours in tension and compression a Drucker-Prager law is included in the creep law according to:

$$v^p = v_0(\frac{\bar{\tau}}{s+\alpha\pi})^{1/m} \qquad \text{Eq.2}$$

where v^p is the equivalent plastic shear strain rate, $\bar{\tau}$ the equivalent shear stress, s the internal variable defined below and $-\pi$ the hydrostatic stress.

3) Isotropic hardening gives rise to the first hardening occurrence in the hardening-softening-hardening sequence. Two coupled internal variables are defined: *s* the resistance to plastic flow and η the local free volume. Their evolution equations read:

$$\dot{s} = h_0\left(1 - \frac{s}{\tilde{s}(\eta)}\right)\nu^{\mathrm{p}} \qquad \text{Eq.3}$$

$$\dot{\eta} = g_0\left(\frac{s}{s_{\mathrm{cv}}} - 1\right)\nu^{\mathrm{p}} \qquad \text{Eq.4}$$

$$\tilde{s}(\eta) = s_{\mathrm{cv}}\left[1 + b(\eta_{\mathrm{cv}} - \eta)\right] \qquad \text{Eq.5}$$

4) Typical material parameters values are given in Ref.1 for Polycarbonate:

G (GPa)	K (GPa)	MR (MPa)	LL (-)	NU0 (s^{-1})	M (-)
0.857	2.24	11.0	1.45	0.0017	0.011

ALPHA (-)	H0 (GPa)	SCV (MPa)	B (-)	ECV (-)	G0 (-)	S0 (MPa)
0.08	2.75	24.0	825	0.001	0.006	20.0

[1] Anand, L., Gurtin, M.E., 2003, "A theory of amorphous solids undergoing large deformations, with application to polymeric glasses," *International Journal of Solids and Structures*, 40, pp. 1465-1487.

*MAT_CHRONOLOGICAL_VISCOELASTIC

This is Material Type 276. This material model provides a general viscoelastic Maxwell model having up to 6 terms in the prony series expansion and is useful for modeling dense continuum rubbers and solid explosives. It is similar to Material Type 76 but allows the incorporation of aging effects on the material properties. Either the coefficients of the prony series expansion or a relaxation curve may be specified to define the viscoelastic deviatoric and bulk behavior.

The material model can also be used with laminated shell. Either an elastic or viscoelastic layer can be defined with the laminated formulation. To activate laminated shell you need the laminated formulation flag on *CONTROL_SHELL. With the laminated option a userdefined integration rule is needed.

Card 1

1	2	3	4	5	6	7	8
Variable	MID	RO	BULK	PCF	EF	TREF	A	B
Type	A8	F	F	F	F	F	F	F

Insert a blank card here if constants are defined on cards 3,4,... below.

If an elastic layer is defined in a laminated shell this card must be blank.

Card 2

1	2	3	4	5	6	7	8
Variable	LCID	NT	BSTART	TRAMP	LCIDK	NTK	BSTARTK	TRAMPK
Type	F	I	F	F	F	I	F	F

Card Format for viscoelastic constants. Up to 6 cards may be input. A keyword card (with a "*" in column 1) terminates this input if less than 6 cards are used. These cards are not needed if relaxation data is defined. The number of terms for the shear behavior may differ from that for the bulk behavior: simply insert zero if a term is not included.

If an elastic layer is defined you only need to define GI and KI (note in an elastic layer only one card is needed)

Optional Cards	1	2	3	4	5	6	7	8
Variable	GI	BETAI	KI	BETAKI				
Type	F	F	F	F				

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Mass density.
BULK	Elastic bulk modulus.
PCF	Tensile pressure elimination flag for solid elements only. If set to unity tensile pressures are set to zero.
EF	Elastic flag (if equal 1, the layer is elastic. If 0 the layer is viscoelastic).
TREF	Reference temperature for shift function (must be greater than zero).
A	Chronological coefficient $\alpha(t_a)$. See Remarks below.
B	Chronological coefficient $\beta(t_a)$. See Remarks below.
LCID	Load curve ID for deviatoric behavior if constants, G_i, and β_i are determined via a least squares fit. This relaxation curve is shown below.
NT	Number of terms in shear fit. If zero the default is 6. Fewer than NT terms will be used if the fit produces one or more negative shear moduli. Currently, the maximum number is set to 6.
BSTART	In the fit, β_1 is set to zero, β_2 is set to BSTART, β_3 is 10 times β_2, β_4 is 100 times greater than β_3, and so on. If zero, BSTART is determined by an iterative trial and error scheme.
TRAMP	Optional ramp time for loading.
LCIDK	Load curve ID for bulk behavior if constants, K_i, and $\beta\kappa_i$ are determined via a least squares fit. This relaxation curve is shown below.
NTK	Number of terms desired in bulk fit. If zero the default is 6. Currently, the maximum number is set to 6.

VARIABLE	DESCRIPTION
BSTARTK	In the fit, $\beta\kappa_1$ is set to zero, $\beta\kappa_2$ is set to BSTARTK, $\beta\kappa_3$ is 10 times $\beta\kappa_2$, $\beta\kappa_4$ is 100 times greater than $\beta\kappa_3$, and so on. If zero, BSTARTK is determined by an iterative trial and error scheme.
TRAMPK	Optional ramp time for bulk loading.
GI	Optional shear relaxation modulus for the ith term
BETAI	Optional shear decay constant for the ith term
KI	Optional bulk relaxation modulus for the ith term
BETAKI	Optional bulk decay constant for the ith term

Remarks:

The Cauchy stress, σ_{ij}, is related to the strain rate by

$$\sigma_{ij}(t) = -p\delta_{ij} + \int_0^t g_{ijkl}(t-\tau)\frac{\partial \varepsilon_{kl}(\tau)}{\partial \tau}d\tau \ . \qquad (1)$$

For this model, it is postulated that the mathematical form is preserved in the constitutive equation for aging; however two new material functions, $g_0'(t_a)$ and $g_1'(t_a,t)$ are introduced to replace g_0 and $g_1(t)$, which is expressed in terms of a Prony series as in material model 76, *MAT_GENERAL_VISCOELASTIC. The aging time is denoted by t_a.

$$\sigma_{ij}(t_a,t) = -p\delta_{ij} + \int_0^t g_{ijkl}'(t_a,t-\tau)\frac{\partial \varepsilon_{kl}(\tau)}{\partial \tau}d\tau \qquad (2)$$

where

$$g_{ijkl}'(t_a,t) = \alpha(t_a)g_{ijkl}\left[\beta(t_a)t\right]. \qquad (3)$$

where $\alpha(t_a)$ and $\beta(t_a)$ are two new material properties that are functions of the aging time t_a. The material properties functions $\alpha(t_a)$ and $\beta(t_a)$ will be determined with the experimental results. For determination of $\alpha(t_a)$ and $\beta(t_a)$, Eq. (2) can be written in the following form

$$\log\left(\sigma_{ij} - p\delta_{ij}\right)_{t_a,t} = \log\alpha(t_a) + \log\left(\sigma_{ij} - p\delta_{ij}\right)_{t_a=0,t\to\xi}$$
$$\log\xi = \log\beta(t_a) + \log t \qquad (4)$$

Therefore, if one plots the stress versus time on log-log scales, with the vertical axis being the stress and the horizontal axis being the time, then the stress-relaxation curve for any aged time history can be obtained directly from the stress-relaxation curve at $t_a = 0$ by imposing a vertical shift and a horizontal shift on the stress-relaxation curves. The vertical shift and the horizontal shift are $\log \alpha(t_a)$ and $\log \beta(t_a)$ respectively.

***MAT_ALE_VISCOUS**

This may also be referred to as MAT_ALE_02. This "fluid-like" material model is very similar to Material Type 9 (*MAT_NULL). It allows the modeling of non-viscous fluids with constant or variable viscosity. The variable viscosity is a function of an equivalent deviatoric strain rate. If inviscid material is modeled, the deviatoric or viscous stresses are zero, and the equation of state supplies the pressures (or diagonal components of the stress tensor). All *MAT_ALE_ cards apply only to ALE element formulation.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	PC	MULO	MUHI	RK	Not used	RN
Type	I	F	F	F	F	F		F
Defaults	none	none	0.0	0.0	0.0	0.0		0.0

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
PC	Pressure cutoff (≤ 0.0), (See remark 4).
MULO	There are 4 possible cases (See remark 1): 1) If MULO=0.0, then inviscid fluid is assumed. 2) If MULO > 0.0, and MUHI=0.0 or is not defined, then this is the traditional constant dynamic viscosity coefficient μ. 3) If MULO > 0.0, and MUHI > 0.0, then MULO and MUHI are lower and upper viscosity limit values for a power-law-like variable viscosity model. 4) If MULO is negative (for example, MULO = -1), then a user-input data load curve (with LCID=1) defining dynamic viscosity as a function of equivalent strain rate is used.
MUHI	Upper dynamic viscosity limit (default=0.0). This is defined only if RK and RN are defined for the variable viscosity case.
RK	Variable dynamic viscosity multiplier (See remark 6).
RN	Variable dynamic viscosity exponent (See remark 6).

Remarks:

1. The null material must be used with an equation-of-state. Pressure cutoff is negative in tension. A (deviatoric) viscous stress of the form

$$\sigma'_{ij} = 2\mu\dot{\varepsilon}'_{ij}$$
$$\left[\frac{N}{m^2}\right] \sim \left[\frac{N}{m^2}s\right]\left[\frac{1}{s}\right]$$

   is computed for nonzero μ where $\dot{\varepsilon}'_{ij}$ is the deviatoric strain rate. μ is the dynamic viscosity. For example, in SI unit system, μ has a unit of [Pa*s].

2. The null material has no shear stiffness and hourglass control must be used with care. In some applications, the default hourglass coefficient might lead to significant energy losses. In general for fluid(s), the hourglass coefficient QM should be small (in the range 1.0E-4 to 1.0E-6 for the standard default IHQ choice).

3. Null material has no yield strength and behaves in a fluid-like manner.

4. The pressure cut-off, PC, must be defined to allow for a material to "numerically" cavitate. In other words, when a material undergoes dilatation above certain magnitude, it should no longer be able to resist this dilatation. Since dilatation stress or pressure is negative, setting PC limit to a very small negative number would allow for the material to cavitate once the pressure in the material goes below this negative value.

5. If the viscosity exponent is less than 1.0, $RN < 1.0$, then $RN - 1.0 < 0.0$. In this case, at very low equivalent strain rate, the viscosity can be artificially very high. MULO is then used as the viscosity value.

6. The empirical variable dynamic viscosity is typically modeled as a function of *equivalent shear rate* based on experimental data.

$$\mu(\dot{\bar{\gamma}}) = RK \bullet \dot{\bar{\gamma}}^{(RN-1)}$$

   For an incompressible fluid, this may be written equivalently as

$$\mu(\dot{\bar{\varepsilon}}) = RK \bullet \dot{\bar{\varepsilon}}^{(RN-1)}$$

   The "overbar" denotes a scalar equivalence. The "dot" denotes a time derivative or rate effect. And the "prime" symbol denotes deviatoric or volume preserving components. The *equivalent shear rate* components may be related to the basic definition of (small-strain) strain rate components as follows:

$$\dot{\varepsilon}_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) \Rightarrow \dot{\varepsilon}'_{ij} = \dot{\varepsilon}_{ij} - \delta_{ij}\left(\frac{\dot{\varepsilon}_{kk}}{3}\right)$$

$$\dot{\gamma}_{ij} = 2\dot{\varepsilon}_{ij}$$

Typically, the 2nd invariant of the deviatoric strain rate tensor is defined as:

$$I_{2\dot{\varepsilon}'} = \frac{1}{2}\left[\dot{\varepsilon}'_{ij}\dot{\varepsilon}'_{ij}\right]$$

The *equivalent (small-strain) deviatoric strain rate* is defined as:

$$\dot{\bar{\varepsilon}}' \equiv 2\sqrt{I_{2\dot{\varepsilon}'}} = \sqrt{2\left[\dot{\varepsilon}'_{ij}\dot{\varepsilon}'_{ij}\right]} = \sqrt{4\left[\dot{\varepsilon}'^2_{12} + \dot{\varepsilon}'^2_{23} + \dot{\varepsilon}'^2_{31}\right] + 2\left[\dot{\varepsilon}'^2_{11} + \dot{\varepsilon}'^2_{22} + \dot{\varepsilon}'^2_{33}\right]}$$

In non-Newtonian literatures, the *equivalent shear rate* is sometimes defined as

$$\dot{\bar{\gamma}} \equiv \sqrt{\frac{\dot{\gamma}_{ij}\dot{\gamma}_{ij}}{2}} = \sqrt{2\dot{\varepsilon}_{ij}\dot{\varepsilon}_{ij}} = \sqrt{4\left[\dot{\varepsilon}^2_{12} + \dot{\varepsilon}^2_{23} + \dot{\varepsilon}^2_{31}\right] + 2\left[\dot{\varepsilon}^2_{11} + \dot{\varepsilon}^2_{22} + \dot{\varepsilon}^2_{33}\right]}$$

It turns out that, (a) for incompressible materials ($\dot{\varepsilon}_{kk} = 0$), and (b) the shear terms are equivalent when $i \neq j$ ➔ $\dot{\varepsilon}_{ij} = \dot{\varepsilon}'_{ij}$, the *equivalent shear rate* is algebraically equivalent to the *equivalent (small-strain) deviatoric strain rate*.

$$\dot{\bar{\varepsilon}}' = \dot{\bar{\gamma}}$$

***MAT_ALE_GAS_MIXTURE**

This may also be referred to as *MAT_ALE_03. This model is used to simulate thermally equilibrated ideal gas mixtures. This only works with the multi-material ALE formulation (ELFORM=11 in *SECTION_SOLID). This keyword needs to be used together with *INITIAL_GAS_MIXTURE for the initialization of gas densities and temperatures. When applied in the context of ALE airbag modeling, the injection of inflator gas is done with a *SECTION_POINT_SOURCE_MIXTURE command which controls the injection process. This is an identical material model to the *MAT_GAS_MIXTURE model.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	IADIAB	RUNIV					
Type	A8	I	F					
Default	none	0	0.0					
Remark		5	1					

Card 2: Method (A) RUNIV=BLANK or 0.0 ➔ Per-mass unit is used

Card 2	1	2	3	4	5	6	7	8
Variable	CVmass1	CVmass2	CVmass3	CVmass4	CVmass5	CVmass6	CVmass7	Cvmass8
Type	F	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	none	None
Remark								

Card 3: Method (A) RUNIV=BLANK or 0.0 ➔ Per-mass unit is used

Card 3	1	2	3	4	5	6	7	8
Variable	CPmass1	CPmass2	CPmass3	CPmass4	CPmass5	CPmass6	CPmass7	Cpmass8
Type	F	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	none	None
Remark								

Card 2: Method (B) RUNIV is nonzero

Card 2	1	2	3	4	5	6	7	8
Variable	MOLWT1	MOLWT2	MOLWT3	MOLWT4	MOLWT5	MOLWT6	MOLWT7	MOLWT8
Type	F	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	none	None
Remark	2							

Card 3: Method (B) RUNIV is nonzero ➔ Per-mole unit is used

Card 3	1	2	3	4	5	6	7	8
Variable	CPmole1	CPmole2	CPmole3	CPmole4	CPmole5	CPmole6	Cpmole7	CPmole8
Type	F	F	F	F	F	F	F	F
Default	none	none	none	none	None	none	none	None
Remark	2							

Card 4: Method (B) RUNIV is nonzero

Card 4	1	2	3	4	5	6	7	8
Variable	B1	B2	B3	B4	B5	B6	B7	B8
Type	F	F	F	F	F	F	F	F
Default	none	none	none	none	None	none	none	None
Remark	2							

Card 5: Method (B) RUNIV is nonzero

Card 5	1	2	3	4	5	6	7	8
Variable	C1	C2	C3	C4	C5	C6	C7	C8
Type	F	F	F	F	F	F	F	F
Default	none	none	none	none	None	none	none	None
Remark	2							

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
IADIAB	This flag (default=0) is used to turn ON/OFF adiabatic compression logics for an ideal gas (remark 5). EQ.0: OFF (default) EQ.1: ON
RUNIV	Universal gas constant in per-mole unit (8.31447 J/(mole*K)).
CVmass1-CVmass8	If RUNIV is BLANK or zero (method A): Heat capacity at constant volume for up to eight different gases in per-mass unit.

VARIABLE	DESCRIPTION
CPmass1-CPmass8	If RUNIV is BLANK or zero (method A): Heat capacity at constant pressure for up to eight different gases in per-mass unit.
MOLWT1-MOLWT8	If RUNIV is nonzero (method B): Molecular weight of each ideal gas in the mixture (mass-unit/mole).
CPmole1-CPmole8	If RUNIV is nonzero (method B): Heat capacity at constant pressure for up to eight different gases in per-mole unit. These are nominal heat capacity values typically at STP. These are denoted by the variable "A" in the equation in remark 2.
B1-B8	If RUNIV is nonzero (method B): First order coefficient for a temperature dependent heat capacity at constant pressure for up to eight different gases. These are denoted by the variable "B" in the equation in remark 2.
C1-C8	If RUNIV is nonzero (method B): Second order coefficient for a temperature dependent heat capacity at constant pressure for up to eight different gases. These are denoted by the variable "C" in the equation in remark 2.

Remarks:

1. There are 2 methods of defining the gas properties for the mixture. If RUNIV is BLANK or ZERO ➔ Method (A) is used to define constant heat capacities where per-mass unit values of C_v and C_p are input. Only cards 2 and 3 are required for this method. Method (B) is used to define constant or temperature dependent heat capacities where per-mole unit values of C_p are input. Cards 2-5 are required for this method.

2. The per-mass-unit, temperature-dependent, constant-pressure heat capacity is

$$C_p(T) = \frac{[A + B*T + C*T^2]}{MW} \sim \frac{J}{kg*K} \qquad B \sim J/(mole*K^2)$$

$$A = \tilde{C}_{P0} \sim J/(mole*K) \qquad C \sim J/(mole*K^3)$$

   The units shown are only for demonstration of the equation.

3. The initial temperature and the density of the gas species present in a mesh or part at time zero is specified by the keyword *INITIAL_GAS_MIXTURE.

4. The ideal gas mixture is assumed to be thermal equilibrium, that is, all species are at the same temperature (T). The gases in the mixture are also assumed to follow Dalton's Partial Pressure Law, $P = \sum_{i}^{ngas} P_i$. The partial pressure of each gas is then $P_i = \rho_i R_{gas_i} T$

where $R_{gasi} = \frac{R_{univ}}{MW}$. The individual gas species temperature equals the mixture temperature. The temperature is computed from the internal energy where the *mixture internal energy per unit volume* is used,

$$e_V = \sum_i^{ngas} \rho_i C_{V_i} T_i = \sum_i^{ngas} \rho_i C_{V_i} T$$

$$T = T_i = \frac{e_V}{\sum_i^{ngas} \rho_i C_{V_i}}$$

In general, the advection step conserves momentum and internal energy, but not kinetic energy. This can result in energy lost in the system and lead to a pressure drop. In *MAT_GAS_MIXTURE the dissipated kinetic energy is automatically stored in the internal energy. Thus in effect the total energy is conserved instead of conserving just the internal energy. This numerical scheme has been shown to improve accuracy in some cases. However, the user should always be vigilant and check the physics of the problem closely.

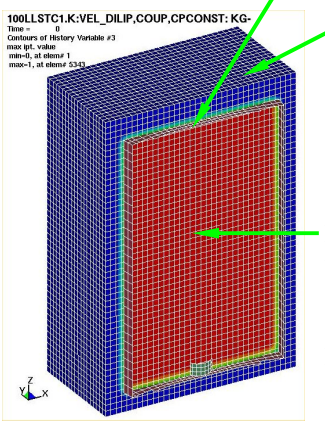
5. As an example consider an airbag surrounded by ambient air. As the inflator gas flows into the bag, the ALE elements cut by the airbag fabric shell elements will contain some inflator gas inside and some ambient air outside. The multi-material element treatment is not perfect. Consequently the temperature of the outside air may, occasionally, be made artificially high after the multi-material element treatment. To prevent the outside ambient air from getting artificially high T, set IDIAB=1 for the ambient air outside. Simple adiabatic compression equation is then assumed for the outside air. The use of this flag may be needed, but only when that outside air is modeled by the *MAT_GAS_MIXTURE card.

Example:

Consider a tank test model where the Lagrangian tank (Part S1) is surrounded by an ALE air mesh (Part H4=AMMGID 1). There are 2 ALE parts which are defined but initially have no corresponding mesh: part 5 (H5=AMMGID 2) is the resident gas inside the tank at t = 0, and part 6 (H6=AMMGID 2) is the inflator gas(es) which is injected into the tank when t > 0. AMMGID stands for ALE Multi-Material Group ID. Please see figure and input below. The *MAT_GAS_MIXTURE (MGM) card defines the gas properties of ALE parts H5 & H6. The MGM card input for both method (A) and (B) are shown.

The *INITIAL_GAS_MIXTURE card is also shown. It basically specifies that "AMMGID 2 may be present in part or mesh H4 at t=0, and the initial density of this gas is defined in the rho1 position which corresponds to the 1st material in the mixture (or H5, the resident gas)."

Example configuration:



Sample input:

```
$---------------------------------------------------------------------------------
*PART
H5 = initial gas inside the tank
$      PID     SECID       MID     EOSID      HGID      GRAV    ADPOPT      TMID
         5         5         5         0         5         0         0
*SECTION_SOLID
         5        11         0
$---------------------------------------------------------------------------------
$ Example 1:  Constant heat capacities using per-mass unit.
$*MAT_GAS_MIXTURE
$      MID    IADIAB     R_univ
$        5         0         0
$  Cv1_mas   Cv2_mas   Cv3_mas   Cv4_mas   Cv5_mas   Cv6_mas   Cv7_mas   Cv8_mas
$718.7828911237.56228
$  Cp1_mas   Cp2_mas   Cp3_mas   Cp4_mas   Cp5_mas   Cp6_mas   Cp7_mas   Cp8_mas
$1007.00058 1606.1117
$---------------------------------------------------------------------------------
$ Example 2:  Variable heat capacities using per-mole unit.
*MAT_GAS_MIXTURE
$      MID    IADIAB     R_univ
         5         0  8.314470
$      MW1       MW2       MW3       MW4       MW5       MW6       MW7       MW8
 0.0288479   0.02256
$  Cp1_mol   Cp2_mol   Cp3_mol   Cp4_mol   Cp5_mol   Cp6_mol   Cp7_mol   Cp8_mol
 29.049852  36.23388
$       B1        B2        B3        B4        B5        B6        B7        B8
  7.056E-3  0.132E-1
$       C1        C2        C3        C4        C5        C6        C7        C8
 -1.225E-6 -0.190E-5
$---------------------------------------------------------------------------------
$ One card is defined for each AMMG that will occupy some elements of a mesh set
*INITIAL_GAS_MIXTURE
$      SID     STYPE     MMGID        T0
         4         1         1    298.15
$     RHO1      RHO2      RHO3      RHO4      RHO5      RHO6      RHO7      RHO8
1.17913E-9
*INITIAL_GAS_MIXTURE
$      SID     STYPE     MMGID        T0
         4         1         2    298.15
$     RHO1      RHO2      RHO3      RHO4      RHO5      RHO6      RHO7      RHO8
1.17913E-9
$---------------------------------------------------------------------------------
```

***MAT_SPRING_ELASTIC**

This is Material Type 1 for discrete springs and dampers. This provides a translational or rotational elastic spring located between two nodes. Only one degree of freedom is connected.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	K						
Type	A8	F						

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
K	Elastic stiffness (force/displacement) or (moment/rotation).

*MAT_DAMPER_VISCOUS

This is Material Type 2 for discrete springs and dampers. This material provides a linear translational or rotational damper located between two nodes. Only one degree of freedom is then connected.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	DC						
Type	A8	F						

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
DC	Damping constant (force/displacement rate) or (moment/rotation rate).

*MAT_SPRING_ELASTOPLASTIC

This is Material Type 3 for discrete springs and dampers. This material provides an elastoplastic translational or rotational spring with isotropic hardening located between two nodes. Only one degree of freedom is connected.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	K	KT	FY				
Type	A8	F	F	F				

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
K	Elastic stiffness (force/displacement) or (moment/rotation).
KT	Tangent stiffness (force/displacement) or (moment/rotation).
FY	Yield (force) or (moment).

*MAT_SPRING_NONLINEAR_ELASTIC

This is Material Type 4 for discrete springs and dampers. This material provides a nonlinear elastic translational and rotational spring with arbitrary force versus displacement and moment versus rotation, respectively. Optionally, strain rate effects can be considered through a velocity dependent scale factor. With the spring located between two nodes, only one degree of freedom is connected.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	LCD	LCR					
Type	A8	I	I					

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
LCD	Load curve ID describing force versus displacement or moment versus rotation relationship
LCR	Optional load curve describing scale factor on force or moment as a function of relative velocity or. rotational velocity, respectively. The load curve must define the response in the negative and positive quadrants and pass through point (0,0).

*MAT_DAMPER_NONLINEAR_VISCOUS

This is Material Type 5 for discrete springs and dampers. This material provides a viscous translational damper with an arbitrary force versus velocity dependency, or a rotational damper with an arbitrary moment versus rotational velocity dependency. With the damper located between two nodes, only one degree of freedom is connected.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	LCDR						
Type	A8	I						

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
LCDR	Load curve identification describing force versus rate-of-displacement relationship or a moment versus rate-of-rotation relationship. The load curve must define the response in the negative and positive quadrants and pass through point (0,0).

*MAT_SPRING_GENERAL_NONLINEAR

This is Material Type 6 for discrete springs and dampers. This material provides a general nonlinear translational or rotational spring with arbitrary loading and unloading definitions. Optionally, hardening or softening can be defined. With the spring located between two nodes, only one degree of freedom is connected.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	LCDL	LCDU	BETA	TYI	CYI		
Type	A8	I	I	F	F	F		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
LCDL	Load curve identification describing force/torque versus displacement/rotation relationship for loading, see Figure 126.1.
LCDU	Load curve identification describing force/torque versus displacement/rotation relationship for unloading, see Figure 119.1.
BETA	Hardening parameter, β: EQ.0.0: tensile and compressive yield with strain softening (negative or zero slope allowed in the force versus displacement. load curves), NE.0.0: kinematic **hardening without strain softening,** EQ.1.0: isotropic **hardening without strain softening.**
TYI	Initial yield force in tension (> 0)
CYI	Initial yield force in compression (< 0)

Remarks:

Load curve points are in the format (displacement, force or rotation, moment). The points must be in order starting with the most negative (compressive) displacement or rotation and ending with the most positive (tensile) value. The curves need not be symmetrical.

The displacement origin of the "unloading" curve is arbitrary, since it will be shifted as necessary as the element extends and contracts. On reverse yielding the "loading" curve will also be shifted along the displacement re or. rotation axis. The initial tensile and compressive yield forces (TYI and CYI) define a range within which the element remains elastic (i.e. the

"loading" curve is used for both loading and unloading). If at any time the force in the element exceeds this range, the element is deemed to have yielded, and at all subsequent times the "unloading" curve is used for unloading.

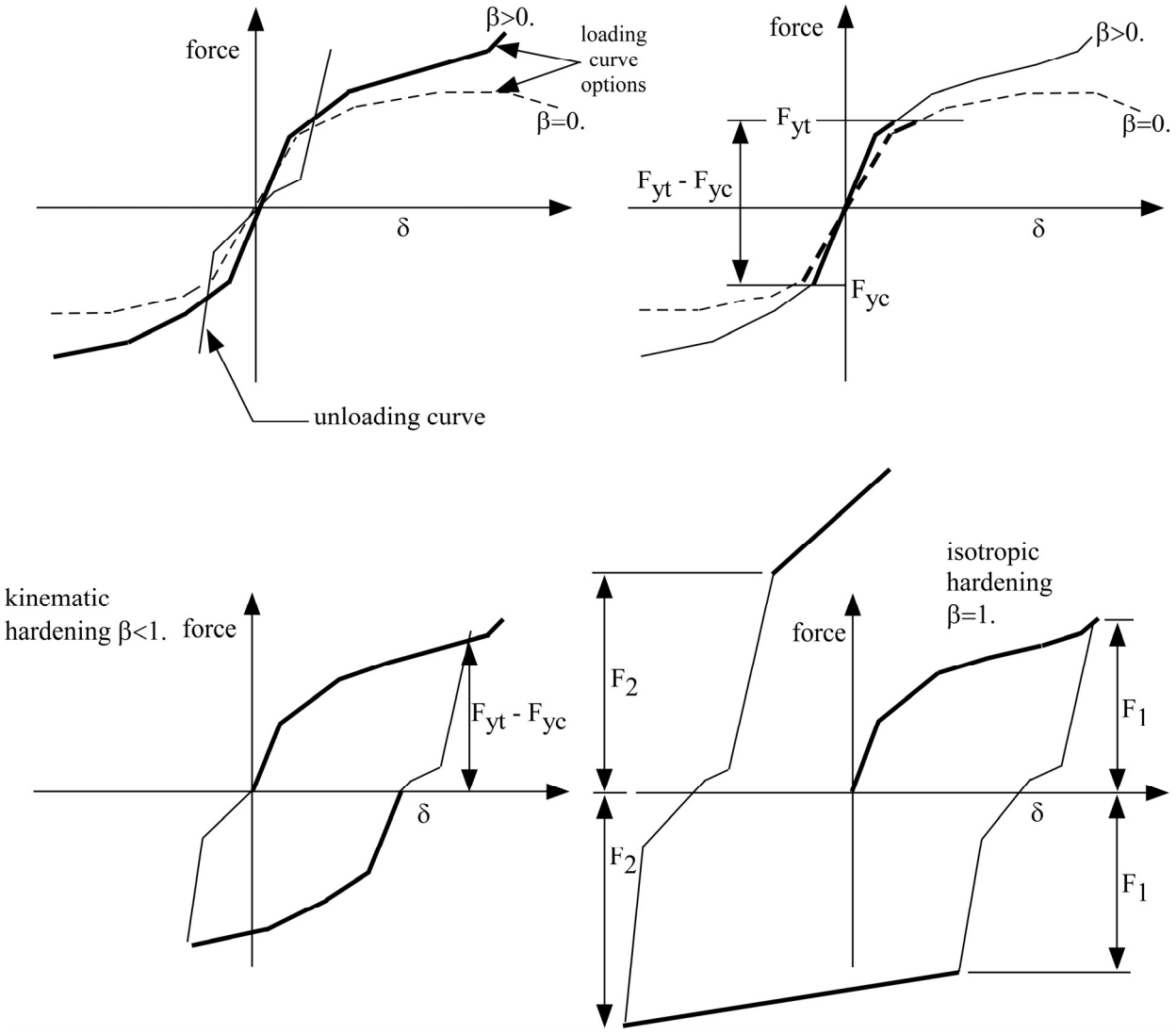


Figure S06.1. General nonlinear material for discrete elements.

*MAT_SPRING_MAXWELL

This is Material Type 7 for discrete springs and dampers. This material provides a three Parameter Maxwell Viscoelastic translational or rotational spring. Optionally, a cutoff time with a remaining constant force/moment can be defined.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	K0	KI	BETA	TC	FC	COPT	
Type	A8	F	F	F	F	F	F	
Default	---	---	---	---	10^{20}	0	0	

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
K0	K_0, short time stiffness
KI	K_∞, long time stiffness
BETA	Decay parameter.
TC	Cut off time. After this time a constant force/moment is transmitted.
FC	Force/moment after cutoff time
COPT	Time implementation option: EQ.0: incremental time change, NE.0: continuous time change.

Remarks:

The time varying stiffness K(t) may be described in terms of the input parameters as

$$K(t) = K_\infty + (K_0 - K_\infty)e^{-\beta t}.$$

This equation was implemented by Schwer [1991] as either a continuous function of time or incrementally following the approach of Herrmann and Peterson [1968]. The continuous function of time implementation has the disadvantage of the energy absorber's resistance decaying with increasing time even without deformation. The advantage of the incremental

implementation is that an energy absorber must undergo some deformation before its resistance decays, i.e., there is no decay until impact, even in delayed impacts. The disadvantage of the incremental implementation is that very rapid decreases in resistance cannot be easily matched.

*MAT_SPRING_INELASTIC

This is Material Type 8 for discrete springs and dampers. This material provides an inelastic tension or compression only, translational or rotational spring. Optionally, a user-specified unloading stiffness can be taken instead of the maximum loading stiffness.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	LCFD	KU	CTF				
Type	A8	I	F	F				

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
LCFD	Load curve identification describing arbitrary force/torque versus displacement/rotation relationship. This curve must be defined in the positive force-displacement quadrant regardless of whether the spring acts in tension or compression.
KU	Unloading stiffness (optional). The maximum of KU and the maximum loading stiffness in the force/displacement or the moment/rotation curve is used for unloading.
CTF	Flag for compression/tension: EQ.-1.0: tension only, EQ.0.0: default is set to 1.0, EQ.1.0: compression only.

*MAT_SPRING_TRILINEAR_DEGRADING

This is Material Type 13 for discrete springs and dampers. This material allows concrete shearwalls to be modeled as discrete elements under applied seismic loading. It represents cracking of the concrete, yield of the reinforcement and overall failure. Under cyclic loading, the stiffness of the spring degrades but the strength does not.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	DEFL1	F1	DEFL2	F2	DEFL3	F3	FFLAG
Type	A8	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
DEFL1	Deflection at point where concrete cracking occurs.
F1	Force corresponding to DEFL1
DEFL2	Deflection at point where reinforcement yields
F2	Force corresponding to DEFL2
DEFL3	Deflection at complete failure
F3	Force corresponding to DEFL3
FFLAG	Failure flag.

*MAT_SPRING_SQUAT_SHEARWALL

This is Material Type 14 for discrete springs and dampers. This material allows squat shear walls to be modeled using discrete elements. The behavior model captures concrete cracking, reinforcement yield, ultimate strength followed by degradation of strength finally leading to collapse.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	A14	B14	C14	D14	E14	LCID	FSD
Type	A8	F	F	F	F	F	I	F

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
A14	Material coefficient A
B14	Material coefficient B
C14	Material coefficient C
D14	Material coefficient D
E14	Material coefficient E
LCID	Load curve ID referencing the maximum strength envelope curve
FSD	Sustained strength reduction factor

Material coefficients A, B, C and D are empirically defined constants used to define the shape of the polynomial curves which govern the cyclic behavior of the discrete element. A different polynomial relationship is used to define the loading and unloading paths allowing energy absorption through hysteresis. Coefficient E is used in the definition of the path used to 'jump' from the loading path to the unloading path (or vice versa) where a full hysteresis loop is not completed. The load curve referenced is used to define the force displacement characteristics of the shear wall under monotonic loading. This curve is the basis to which the polynomials defining the cyclic behavior refer to. Finally, on the second and subsequent loading / unloading cycles, the shear wall will have reduced strength. The variable FSD is the sustained strength reduction factor.

***MAT_SPRING_MUSCLE**

This is Material Type 15 for discrete springs and dampers. This material is a Hill-type muscle model with activation. It is for use with discrete elements. The LS-DYNA implementation is due to Dr. J.A. Weiss.

Card 1

Variable	MID	L0	VMAX	SV	A	FMAX	TL	TV
Type	A8	F	F	F	F	F	F	F
Default		1.0		1.0			1.0	1.0

Card 2

Variable	FPE	LMAX	KSH					
Type	F	F	F					
Default	0.0							

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
L0	Initial muscle length, *Lo.*
VMAX	Maximum CE shortening velocity, *Vmax*.
SV	Scale factor, *Sv,* for *Vmax* vs. active state. LT.0: absolute value gives load curve ID GE.0: constant value of 1.0 is used
A	Activation level vs. time function. LT.0: absolute value gives load curve ID GE.0: constant value of *A* is used
FMAX	Peak isometric force, *Fmax*.

VARIABLE	DESCRIPTION
TL	Active tension vs. length function. LT.0: absolute value gives load curve ID GE.0: constant value of 1.0 is used
TV	Active tension vs. velocity function. LT.0: absolute value gives load curve ID GE.0: constant value of 1.0 is used
FPE	Force vs. length function, *Fpe*, for parallel elastic element. LT.0: absolute value gives load curve ID EQ.0: exponential function is used (see below) GT.0: constant value of 0.0 is used
LMAX	Relative length when *Fpe* reaches *Fmax*. Required if *Fpe*=0 above.
KSH	Constant, *Ksh*, governing the exponential rise of *Fpe*. Required if *Fpe*=0 above.

Remarks:

The material behavior of the muscle model is adapted from the original model proposed by Hill [1938]. Reviews of this model and extensions can be found in Winters [1990] and Zajac [1989]. The most basic Hill-type muscle model consists of a contractile element (CE) and a parallel elastic element (PE) (Figure 139.1). An additional series elastic element (SEE) can be added to represent tendon compliance. The main assumptions of the Hill model are that the contractile element is entirely stress free and freely distensible in the resting state, and is described exactly by Hill's equation (or some variation). When the muscle is activated, the series and parallel elements are elastic, and the whole muscle is a simple combination of identical sarcomeres in series and parallel. The main criticism of Hill's model is that the division of forces between the parallel elements and the division of extensions between the series elements is arbitrary, and cannot be made without introducing auxiliary hypotheses. However, these criticisms apply to *any* discrete element model. Despite these limitations, the Hill model has become extremely useful for modeling musculoskeletal dynamics, as illustrated by its widespread use today.

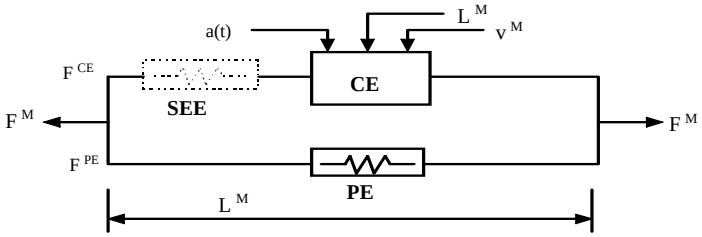


Figure S15.1. Discrete model for muscle contraction dynamics, based on a Hill-type representation. The total force is the sum of passive force F^{PE} and active force F^{CE}. The passive element (PE) represents energy storage from muscle elasticity, while the contractile element (CE) represents force generation by the muscle. The series elastic element (SEE), shown in dashed lines, is often neglected when a series tendon compliance is included. Here, a(t) is the activation level, L^M is the length of the muscle, and v^M is the shortening velocity of the muscle.

When the contractile element (CE) of the Hill model is inactive, the entire resistance to elongation is provided by the PE element and the tendon load-elongation behavior. As activation is increased, force then passes through the CE side of the parallel Hill model, providing the contractile dynamics. The original Hill model accommodated only full activation - this limitation is circumvented in the present implementation by using the modification suggested by Winters (1990). The main features of his approach were to realize that the CE force-velocity input force equals the CE tension-length output force. This yields a three-dimensional curve to describe the force-velocity-length relationship of the CE. If the force-velocity y-intercept scales with activation, then given the activation, length and velocity, the CE force can be determined.

Without the SEE, the total force in the muscle FM is the sum of the force in the CE and the PE because they are in parallel:

$$F^M = F^{PE} + F^{CE}$$

The relationships defining the force generated by the CE and PE as a function of L^M, V^M and a(t) are often scaled by F_{max}, the peak isometric force (p. 80, Winters 1990), L_0, the initial length of the muscle (p. 81, Winters 1990), and V_{max}, the maximum unloaded CE shortening velocity (p. 80, Winters 1990). From these, dimensionless length and velocity can be defined:

$$L = \frac{L^M}{L_o},$$

$$V = \frac{V^M}{V_{max} * S_V(a(t))}$$

Here, S_V scales the maximum CE shortening velocity V_{max} and changes with activation level a(t). This has been suggested by several researchers, i.e. Winters and Stark [1985]. The activation level specifies the level of muscle stimulation as a function of time. Both have values between 0 and 1. The functions $S_V(a(t))$ and a(t) are specified via load curves in LS-DYNA, or default values of S_V=1 and a(t)=0 are used. Note that L is always positive and that V is positive for lengthening and negative for shortening.

The relationship between F^{CE}, V and L was proposed by Bahler et al. [1967]. A three-dimensional relationship between these quantities is now considered standard for computer implementations of Hill-type muscle models [Winters 1990]. It can be written in dimensionless form as:

$$F^{CE} = a(t)*F_{max}*f_{TL}(L)*f_{TV}(V)$$

Here, f_{TL} and f_{TV} are the tension-length and tension-velocity functions for active skeletal muscle. Thus, if current values of L^M, V^M, and a(t) are known, then F^{CE} can be determined (Figure 139.1).

The force in the parallel elastic element F^{PE} is determined directly from the current length of the muscle using an exponential relationship [Winters 1990]:

$$f_{PE} = \frac{F^{PE}}{F_{MAX}} = 0, \quad L \le 1$$

$$f_{PE} = \frac{F^{PE}}{F_{MAX}} = \frac{1}{\exp(K_{sh}) - 1}\left[\exp\left(\frac{K_{sh}}{L_{max}}(L-1)\right) - 1\right], \quad L > 1$$

Here, L_{max} is the relative length at which the force F_{max} occurs, and K_{sh} is a dimensionless shape parameter controlling the rate of rise of the exponential. Alternatively, the user can define a custom f_{PE} curve giving tabular values of normalized force versus dimensionless length as a load curve.

For computation of the total force developed in the muscle F^M, the functions for the tension-length f_{TL} and force-velocity f_{TV} relationships used in the Hill element must be defined. These relationships have been available for over 50 years, but have been refined to allow for behavior such as active lengthening. The active tension-length curve f_{TL} describes the fact that isometric muscle force development is a function of length, with the maximum force occurring at an optimal length. According to Winters, this optimal length is typically around L=1.05, and the force drops off for shorter or longer lengths, approaching zero force for L=0.4 and L=1.5. Thus the curve has a bell-shape. Because of the variability in this curve between muscles, the user must specify the function f_{TL} via a load curve, specifying pairs of points representing the normalized force (with values between 0 and 1) and normalized length L (Figure 163.1).

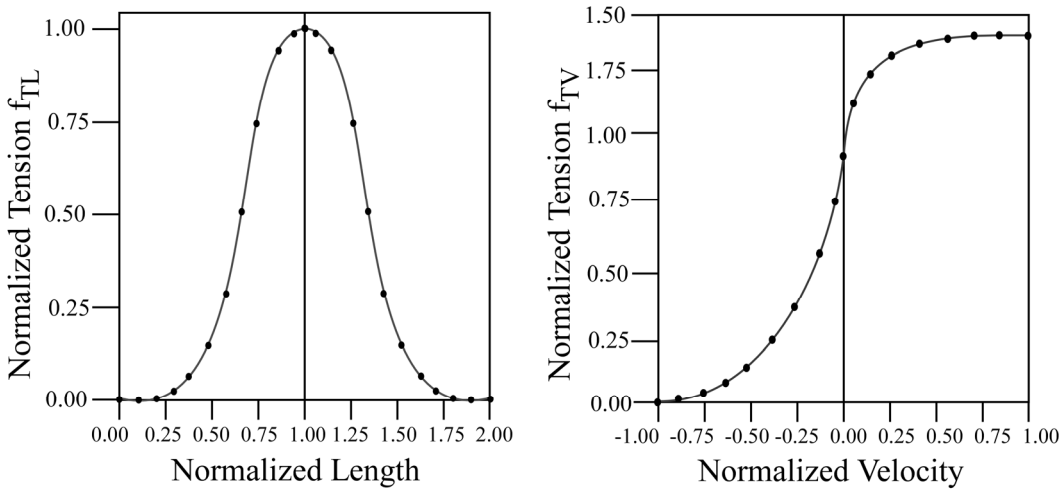


Figure S15.2. Typical normalized tension-length (TL) and tension-velocity (TV) curves for skeletal muscle.

The active tension-velocity relationship f_{TV} used in the muscle model is mainly due to the original work of Hill. Note that the dimensionless velocity V is used. When V=0, the normalized tension is typically chosen to have a value of 1.0. When V is greater than or equal to 0, muscle lengthening occurs. As V increases, the function is typically designed so that the force increases from a value of 1.0 and asymptotes towards a value near 1.4. When V is less than zero, muscle shortening occurs and the classic Hill equation hyperbola is used to drop the normalized tension to 0 (Figure 163.1). The user must specify the function f_{TV} via a load curve, specifying pairs of points representing the normalized tension (with values between 0 and 1) and normalized velocity V.

*MAT_SEATBELT

Purpose: Define a seat belt material. See notes below.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	MPUL	LLCID	ULCID	LMIN			
Type	A8	F	I	I	F			
Default	0	0.	0	0	0.0			

VARIABLE	DESCRIPTION
MID	Belt material number. A unique number or label not exceeding 8 characters must be specified.
MPUL	Mass per unit length
LLCID	Load curve identification for loading (force vs. engineering strain).
ULCID	Load curve identification for unloading (force vs. engineering strain).
LMIN	Minimum length (for elements connected to slip rings and retractors), see notes below.
CSE	Optional compressive stress elimination option which applies to shell elements only (default 0.0): EQ.0.0: eliminate compressive stresses in shell fabric EQ.1.0: don't eliminate compressive stresses. This option should not be used if retractors and sliprings are present in the model.
DAMP	Optional Rayleigh damping coefficient, which applies to shell elements only. A coefficient value of 0.10 is the default corresponding to 10% of critical damping. Sometimes smaller or larger values work better.

Remarks:

Each belt material defines stretch characteristics and mass properties for a set of belt elements. The user enters a load curve for loading, the points of which are (Strain, Force). Strain is defined as engineering strain, i.e.

$$Strain = \frac{current\ \ length}{initial\ length} - 1.$$

Another similar curve is entered to describe the unloading behavior. Both load curves should start at the origin (0,0) and contain positive force and strain values only. The belt material is tension only with zero forces being generated whenever the strain becomes negative. The first non-zero point on the loading curve defines the initial yield point of the material. On unloading, the unloading curve is shifted along the strain axis until it crosses the loading curve at the 'yield' point from which unloading commences. If the initial yield has not yet been exceeded or if the origin of the (shifted) unloading curve is at negative strain, the original loading curves will be used for both loading and unloading. If the strain is less than the strain at the origin of the unloading curve, the belt is slack and no force is generated. Otherwise, forces will then be determined by the unloading curve for unloading and reloading until the strain again exceeds yield after which the loading curves will again be used.

A small amount of damping is automatically included. This reduces high frequency oscillation, but, with realistic force-strain input characteristics and loading rates, does not significantly alter the overall forces-strain performance. The damping forced opposes the relative motion of the nodes and is limited by stability:

$$D = \frac{.1 \times mass \times relativevelocity}{timestepsize}$$

In addition, the magnitude of the damping force is limited to one-tenth of the force calculated from the force-strain relationship and is zero when the belt is slack. Damping forces are not applied to elements attached to sliprings and retractors.

The user inputs a mass per unit length that is used to calculate nodal masses on initialization.

A 'minimum length' is also input. This controls the shortest length allowed in any element and determines when an element passes through sliprings or are absorbed into the retractors. One tenth of a typical initial element length is usually a good choice.

MAT_THERMAL_*OPTION

Available options include:

ISOTROPIC

ORTHOTROPIC

ISOTROPIC_TD

ORTHOTROPIC_TD

ISOTROPIC_PHASE_CHANGE

ISOTROPIC_TD_LC

The *MAT_THERMAL_ cards allow thermal properties to be defined in coupled structural/thermal and thermal only analyses, see *CONTROL_SOLUTION. Thermal properties must be defined for all solid and shell elements in such analyses. Thermal properties need not be defined for beam or discrete elements as these elements are not accounted for in the thermal phase of the calculation. However dummy thermal properties will be echoed for these elements in the D3HSP file.

Thermal material properties are specified by a thermal material ID number (TMID), this number is independent of the material ID number (MID) defined on all other *MAT_.. property cards. In the same analysis identical TMID and MID numbers may exist. The TMID and MID numbers are related through the *PART card.

*MAT_THERMAL_ISOTROPIC

This is thermal material property type 1. It allows isotropic thermal properties to be defined.

Card (1 of 2)

Card 1	1	2	3	4	5	6	7	8
Variable	TMID	TRO	TGRLC	TGMULT	TLAT	HLAT		
Type	A8	F	F	F	F	F		

Card (2 of 2)

Card 2	1	2	3	4	5	6	7	8
Variable	HC	TC						
Type	F	F						

VARIABLE	DESCRIPTION
TMID	Thermal material identification. A unique number or label not exceeding 8 characters must be specified.
TRO	Thermal density: EQ 0.0 default to structural density.
TGRLC	Thermal generation rate curve number, see *DEFINE_CURVE: GT.0: function versus time, EQ.0: use constant multiplier value, TGMULT, LT.0: function versus temperature.
TGMULT	Thermal generation rate multiplier: EQ.0.0: no heat generation.
TLAT	Phase change temperature
HLAT	Latent heat
HC	Heat capacity
TC	Thermal conductivity

*MAT_THERMAL_ORTHOTROPIC

This is thermal material property type 2. It allows orthotropic thermal properties to be defined.

Card (1 of 4)

Card 1	1	2	3	4	5	6	7	8
Variable	TMID	TRO	TGRLC	TGMULT	AOPT	TLAT	HLAT	
Type	A8	F	F	F	F	F	F	

Card (2 of 4)

Card 2	1	2	3	4	5	6	7	8
Variable	HC	K1	K2	K3				
Type	F	F	F	F				

Card (3 of 4)

Card 3	1	2	3	4	5	6	7	8
Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card (4 of 4)

Card 4	1	2	3	4	5	6	7	8
Variable	D1	D2	D3					
Type	F	F	F					

VARIABLE	DESCRIPTION
TMID	Thermal material identification. A unique number or label not exceeding 8 characters must be specified.
TRO	Thermal density: EQ 0.0 default to structural density.
TGRLC	Thermal generation rate curve number, see *DEFINE_CURVE: GT.0: function versus time, EQ.0: use constant multiplier value, TGMULT, LT.0: function versus temperature.
TGMULT	Thermal generation rate multiplier: EQ.0.0: no heat generation.
AOPT	Material axes definition: EQ.0.0: locally orthotropic with material axes by element nodes N_1, N_2 and N_4, EQ.1.0: locally orthotropic with material axes determined by a point in space and global location of element center, EQ.2.0: globally orthotropic with material axes determined by vectors.
TLAT	Phase change temperature
HLAT	Latent heat
HC	Heat capacity
K_1	Thermal conductivity K_1 in local x-direction
K_2	Thermal conductivity K_2 in local y-direction
K_3	Thermal conductivity K_3 in local z-direction
XP, YP, ZP	Define coordinate of point **p** for AOPT = 1
A1, A2, A3	Define components of vector **a** for AOPT = 2
D1, D2, D3	Define components of vector **v** for AOPT = 2

*MAT_THERMAL_ISOTROPIC_TD

This is thermal material property type 3. It allows temperature dependent isotropic properties to be defined. The temperature dependency is defined by specifying a minimum of two and a maximum of eight data points. The properties must be defined for the temperature range that the material will see in the analysis.

Card (1 of 4)

Card 1	1	2	3	4	5	6	7	8
Variable	TMID	TRO	TGRLC	TGMULT	TLAT	HLAT		
Type	A8	F	F	F	F	F		

Card (2 of 4)

Card 2	1	2	3	4	5	6	7	8
Variable	T1	T2	T3	T4	T5	T6	T7	T8
Type	F	F	F	F	F	F	F	F

Card (3 of 4)

Card 3	1	2	3	4	5	6	7	8
Variable	C1	C2	C3	C4	C5	C6	C7	C8
Type	F	F	F	F	F	F	F	F

Card (4 of 4)

Card 4	1	2	3	4	5	6	7	8
Variable	K1	K2	K3	K4	K5	K6	K7	K8
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
TMID	Thermal material identification. A unique number or label not exceeding 8 characters must be specified.
TRO	Thermal density: EQ 0.0 default to structural density.
TGRLC	Thermal generation rate curve number, see *DEFINE_CURVE: GT.0: function versus time, EQ.0: use constant multiplier value, TGMULT, LT.0: function versus temperature.
TGMULT	Thermal generation rate multiplier: EQ.0.0: no heat generation.
TLAT	Phase change temperature
HLAT	Latent heat
T1 ... T8	Temperatures (T1 ... T8)
C1 ... C8	Heat capacity at T1 ... T8
K1 ... K8	Thermal conductivity at T1 ... T8

*MAT_THERMAL_ORTHOTROPIC_TD

This is thermal material property type 4. It allows temperature dependent orthotropic properties to be defined. The temperature dependency is defined by specifying a minimum of two and a maximum of eight data points. The properties must be defined for the temperature range that the material will see in the analysis.

Card (1 of 8)

Card 1	1	2	3	4	5	6	7	8
Variable	TMID	TRO	TGRLC	TGMULT	AOPT	TLAT	HLAT	
Type	A8	F	F	F	F	F	F	

Card (2 of 8)

Card 2	1	2	3	4	5	6	7	8
Variable	T1	T2	T3	T4	T5	T6	T7	T8
Type	F	F	F	F	F	F	F	F

Card (3 of 8)

Card 3	1	2	3	4	5	6	7	8
Variable	C1	C2	C3	C4	C5	C6	C7	C8
Type	F	F	F	F	F	F	F	F

Card (4 of 8)

Card 4	1	2	3	4	5	6	7	8
Variable	(K1)$_1$	(K1)$_2$	(K1)$_3$	(K1)$_4$	(K1)$_5$	(K1)$_6$	(K1)$_7$	(K1)$_8$
Type	F	F	F	F	F	F	F	F

Card (5 of 8)

Card 5	1	2	3	4	5	6	7	8
Variable	(K2)1	(K2)2	(K2)3	(K2)4	(K2)5	(K2)6	(K2)7	(K2)8
Type	F	F	F	F	F	F	F	F

Card (6 of 8)

Card 6	1	2	3	4	5	6	7	8
Variable	(K3)1	(K3)2	(K3)3	(K3)4	(K3)5	(K3)6	(K3)7	(K3)8
Type	F	F	F	F	F	F	F	F

Card (7 of 8)

Card 7	1	2	3	4	5	6	7	8
Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card (8 of 8)

Card 8	1	2	3	4	5	6	7	8
Variable	D1	D2	D3					
Type	F	F	F					

VARIABLE	DESCRIPTION
TMID	Thermal material identification. A unique number or label not exceeding 8 characters must be specified.
TRO	Thermal density: EQ 0.0 default to structural density.

VARIABLE	DESCRIPTION
TGRLC	Thermal generation rate curve number, see *DEFINE_CURVE: GT.0: function versus time, EQ.0: use constant multiplier value, TGMULT, LT.0: function versus temperature.
TGMULT	Thermal generation rate multiplier: EQ.0.0: no heat generation.
AOPT	Material axes definition: (see Mat_*OPTION* TROPIC_ELASTIC for a more complete description): EQ.0.0: locally orthotropic with material axes by element nodes N_1, N_2 and N_4, EQ.1.0: locally orthotropic with material axes determined by a point in space and global location of element center, EQ.2.0: globally orthotropic with material axes determined by vectors.
TLAT	Phase change temperature
HLAT	Latent heat
T1 ... T8	Temperatures (T1 ... T8)
C1 ... C8	Heat capacity at T1 ... T8
$(K1)_1$ ... $(K1)_8$	Thermal conductivity K_1 in local x-direction at T1 ... T8
$(K2)_1$ ... $(K2)_8$	Thermal conductivity K_2 in local y-direction at T1 ... T8
$(K3)_1$ ... $(K3)_8$	Thermal conductivity K_3 in local z-direction at T1 ... T8
XP, YP, ZP	Define coordinate of point **p** for AOPT = 1
A1, A2, A3	Define components of vector **a** for AOPT = 2
D1, D2, D3	Define components of vector **d** for AOPT = 2

*MAT_THERMAL_ISOTROPIC_PHASE_CHANGE

This is thermal material property type 5. It allows temperature dependent isotropic properties with phase change to be defined. The latent heat of the material is defined together with the solid and liquid temperatures. The temperature dependency is defined by specifying a minimum of two and a maximum of eight data points. The properties must be defined for the temperature range that the material will see in the analysis.

Card (1 of 5)

Card 1	1	2	3	4	5	6	7	8
Variable	TMID	TRO	TGRLC	TGMULT				
Type	A8	F	F	F				

Card (2 of 5)

Card 2	1	2	3	4	5	6	7	8
Variable	T1	T2	T3	T4	T5	T6	T7	T8
Type	F	F	F	F	F	F	F	F

Card (3 of 5)

Card 3	1	2	3	4	5	6	7	8
Variable	C1	C2	C3	C4	C5	C6	C7	C8
Type	F	F	F	F	F	F	F	F

Card (4 of 5)

Card 4	1	2	3	4	5	6	7	8
Variable	K1	K2	K3	K4	K5	K6	K7	K8
Type	F	F	F	F	F	F	F	F

Card (5 of 5)

Card 5	1	2	3	4	5	6	7	8
Variable	SOLT	LIQT	LH					
Type	F	F	F					

VARIABLE	DESCRIPTION
TMID	Thermal material identification. A unique number or label not exceeding 8 characters must be specified.
TRO	Thermal density: EQ 0.0 default to structural density.
TGRLC	Thermal generation rate curve number, see *DEFINE_CURVE: GT.0: function versus time, EQ.0: use constant multiplier value, TGMULT, LT.0: function versus temperature.
TGMULT	Thermal generation rate multiplier: EQ.0.0: no heat generation.
T1 ... T8	Temperatures (T1 ... T8)
C1 ... C8	Heat capacity at T1 ... T8
K1 ... K8	Thermal conductivity at T1 ... T8
SOLT	Solid temperature, T_S (must be < T_L)
LIQT	Liquid temperature, T_L (must be > T_S)
LH	Latent heat

Remarks:

During phase change, that is between the solid and liquid temperatures, the heat capacity of the material will be enhanced to account for the latent heat as follows:

$$c(t) = m\left[1 - \cos 2\pi\left(\frac{T - T_S}{T_L - T_S}\right)\right] \qquad T_S < T < T_L$$

Where

T_L = liquid temperature

T_S = solid temperature

T = temperature

m = multiplier such that $\lambda = \int_{T_S}^{T_L} C(T)dT$

λ = latent heat

c = heat capacity

*MAT_THERMAL_ISOTROPIC_TD_LC

This is thermal material property type 6. It allows isotropic thermal properties that are temperature dependent specified by load curves to be defined. The properties must be defined for the temperature range that the material will see in the analysis.

Card (1 of 2)

Card 1	1	2	3	4	5	6	7	8
Variable	TMID	TRO	TGRLC	TGMULT				
Type	A8	F	F	F				

Card (2 of 2)

Card 2	1	2	3	4	5	6	7	8
Variable	HCLC	TCLC						
Type	F	F						

VARIABLE	DESCRIPTION
TMID	Thermal material identification. A unique number or label not exceeding 8 characters must be specified.
TRO	Thermal density: EQ 0.0 default to structural density.
TGRLC	Thermal generation rate curve number, see *DEFINE_CURVE: GT.0: function versus time, EQ.0: use constant multiplier value, TGMULT, LT.0: function versus temperature.
TGMULT	Thermal generation rate multiplier: EQ.0.0: no heat generation.
HCLC	Load curve ID specifying heat capacity vs. temperature.
TCLC	Load curve ID specifying thermal conductivity vs. temperature.

***MAT_THERMAL_USER_DEFINED**

These are Thermal Material Types 11-15. The user can supply his own subroutines. Please consult Appendix H for more information.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	MT	LMC	NVH	AOPT	IORTHO	IHVE
Type	A8	F	F	F	F	F	F	F

Define the following two cards if and only if IORTHO=1

Card 2	1	2	3	4	5	6	7	8
Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card 3

Variable	D1	D2	D3					
Type	F	F	F					

Define LMC material parameters using 8 parameters per card.

Card 4	1	2	3	4	5	6	7	8
Variable	P1	P2	P3	P4	P5	P6	P7	P8
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not exceeding 8 characters must be specified.
RO	Thermal mass density.
MT	User material type (11-15 inclusive).
LMC	Length of material constants array. LMC must not be greater than 32.
NVH	Number of history variables.
AOPT	Material axes option of orthotropic materials. Use if IORTHO=1.0. EQ.0.0: locally orthotropic with material axes by element nodes N_1, N_2 and N_4, EQ.1.0: locally orthotropic with material axes determined by a point in space and global location of element center, EQ.2.0: globally orthotropic with material axes determined by vectors. LT.0.0: the absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_NODES, *DEFINE_COORDINATE_SYSTEM or *DEFINE_COORDINATE_VECTOR). Available in R3 version of 971 and later.
IORTHO	Set to 1.0 if the material is orthotropic.
IHVE	Set to 1.0 to activate exchange of history variables between mechanical and thermal user material models.
XP-D3	Material axes orientation of orthotropic materials. Use if IORTHO=1.0 See *MAT_THERMAL_ORTHOTROPIC for a description.
P1	First material parameter.
.	.
.	.
PLMC	LMCth material parameter.

Remarks:

1. The IHVE=1 option makes it possible for a thermal user material subroutine to read the history variables of a mechanical user material subroutine defined for the same part and vice versa. If the integration points for the thermal and mechanical elements are not coincident then extrapolation/interpolation is used to calculate the value when reading history variables.
2. Option _TITLE is supported
3. *INCLUDE_TRANSFORM: Transformation of units is only supported for RO field and vectors on card 2 and 3.