LS-DYNA
KEYWORD USER'S MANUAL

VOLUME I

March 2001
Version 960

Mailing Address:
Livermore Software Technology Corporation
2876 Waverley Way
Livermore, California 94550-1740

Support Address:
Livermore Software Technology Corporation
7374 Las Positas Road
Livermore, California 94550

TEL: 925-449-2500
FAX: 925-449-2507
EMAIL: sales@lstc.com

TABLE OF CONTENTS

Volume I

TABLE OF CONTENTS

TABLE OF CONTENTS

TABLE OF CONTENTS

TABLE OF CONTENTS

Volume II

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,
- 2×2 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 hystersis 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 quadrilaterial shell element is available as a type 18 shell. This shell is for extracting normal modes and static analysis.
- A C0 linear 4-node quadrilaterial shell element is implemented as element type 20 with drilling stiffness for normal modes and static analysis.
- An assumed strain linear brick element is avaiable 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.

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.

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 I.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".

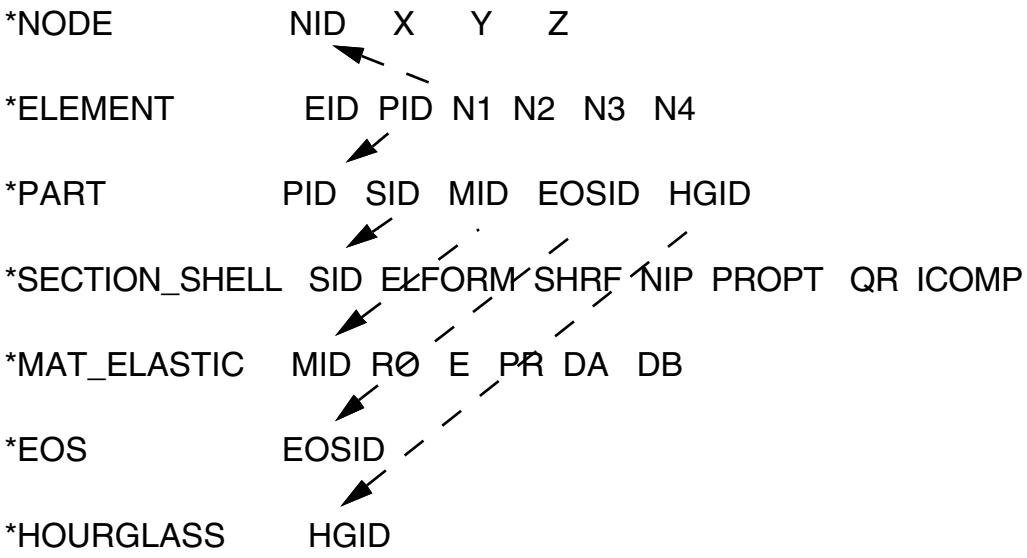


Figure I.1 Organization of the keyword input.

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 of some of the more important 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.

***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.

***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.

***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-DYNA3D.

***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-DYNA3D 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.

***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 *KEYWORD CARD OVERRIDES MEMORY SPECIFIED ON THE EXECUTION LINE.

***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-DYNA3D 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.

***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.

***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.

***SET**
A concept of grouping nodes, elements, materials, etc., in sets is employed throughout the LS-DYNA3D 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 .

***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 I.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 Rigidwalls	Defaults for contacts	*CONTROL_CONTACT
	Definition of contacts	*CONTACT_*Option*
	Definition of rigidwalls	*RIGIDWALL_*Option*

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

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

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 hystersis,

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.2 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.

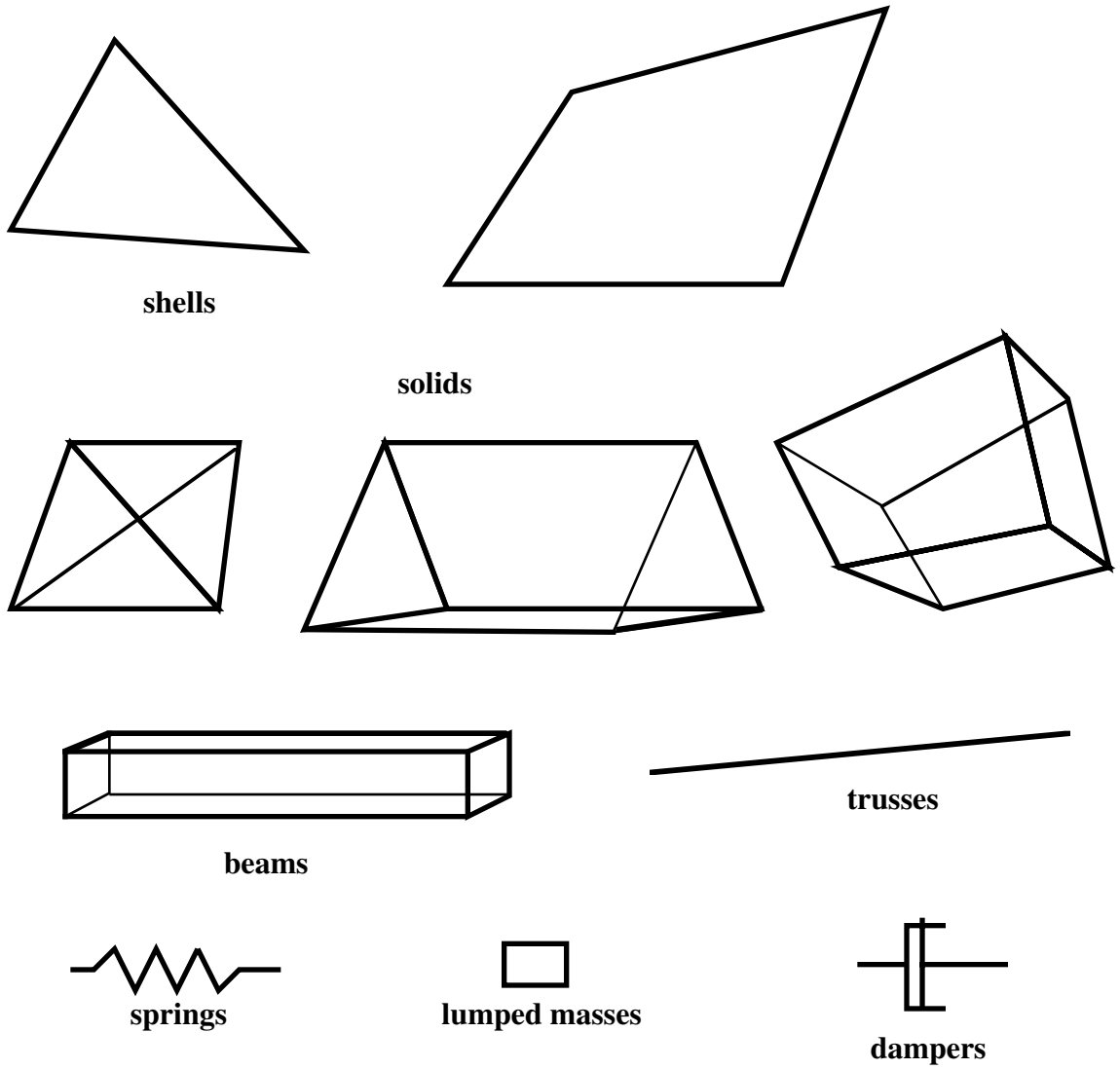


Figure I.2. Elements in LS-DYNA.

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.

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 linear equation solver memory, cpu summary.
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 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 call 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.

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 computers 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 some computer manufacturers.

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** **{KEYWORD}** **{THERMAL}** **{COUPLE}** **{INIT}** MEMORY=**nwds** NCPU= **ncpu** PARA=**para** ENDTIME=**time** NCYLCE=**ncycle**

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 (TOPAZ3D plotfile)
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
nwds = Number of words to be allocated. On engineering workstations a word is usually 32bits. **This number is ignored if memory is 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 *CONTROL_PARALLEL for an explanation of these parameters.
npara = Overrides **PARA** defined in *CONTROL_PARALLEL.
time = Overrides **ENDTIM** defined in *CONTROL_TERMINATION.
ncycle = Overrides **ENDCYC** defined in *CONTROL_TERMINATION.

In order to avoid undesirable or confusing results, each LS-DYNA run should be performed in a separate directory. 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-DYNA3D 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.

File Organization

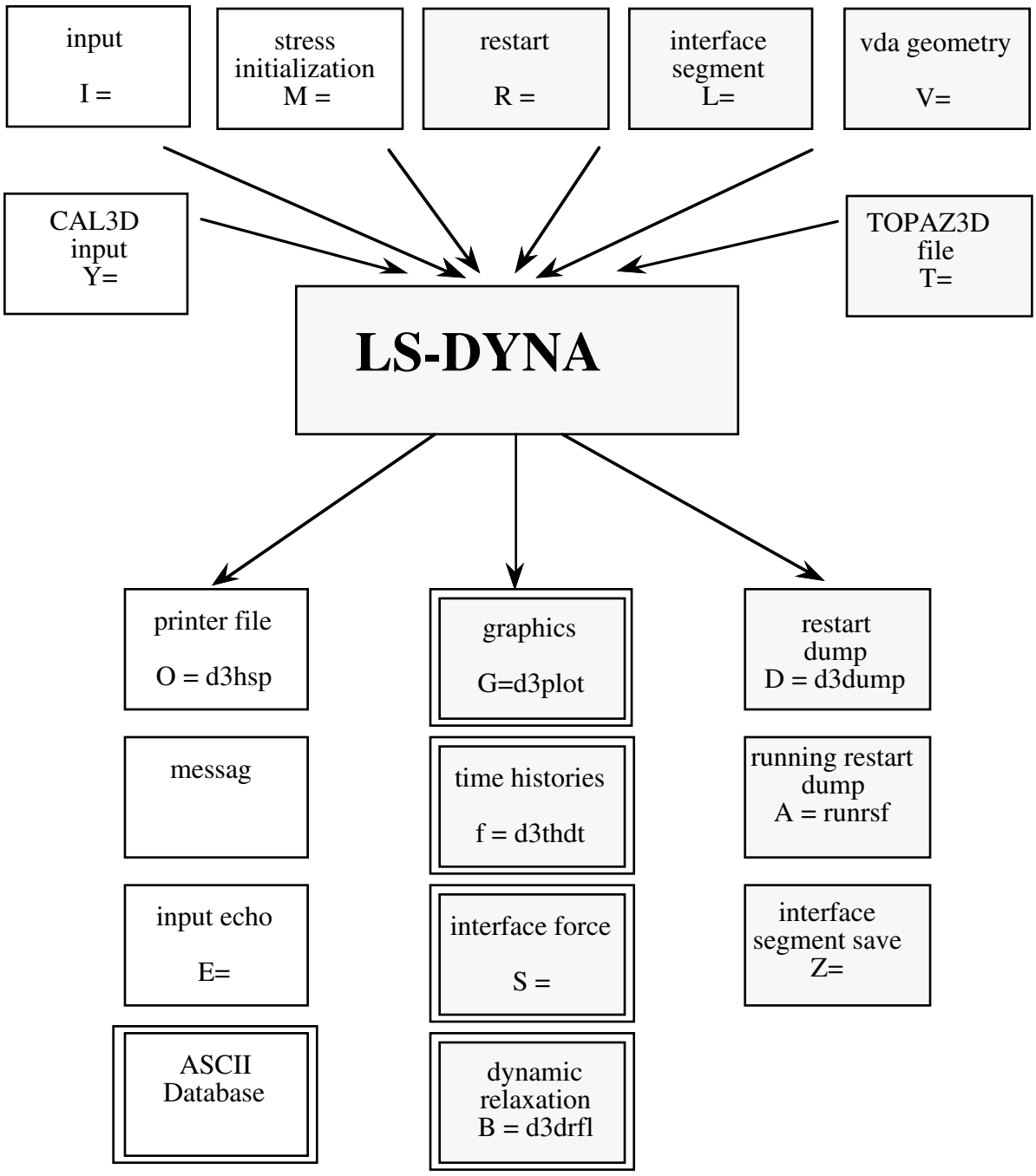


Figure I.3

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* **K E Y W O R D MEMORY=nwds**

where

rtf = restart filename.

Restarting adaptive runs requires that the following parameter be specified on the command line:

LS-DYNA R=**adapt.dump01 .....**

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:

LS-DYNA R=**adapt.dump01** 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 are 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 I.

MESH GENERATION

LS-DYNA is designed to operate with a variety of commercial pre-processing packages. Currently, direct support is available from TRUEGRID[1], PATRAN, FEMB, HYPERMESH, and MEDINA. Several third-party translation programs are available for PATRAN and IDEAS.

Alternately, the pre-processor LS-INGRID [LSTC Report 1019] is available from LSTC and is specialized to LS-DYNA. Some of the capabilities available in LS-INGRID are:

Complete support for all control parameters, loads and material types,

[1] TRUEGRID is a trademark of XYZ Scientific Applications, Inc., PATRAN is a trademark of PDA Engineering, HYPERMESH is a trademark of Altair Engineering, FEMB is a trademark of Engineering Technology Associates, IDEAS is a trademark of Structural Dynamics Research Corporation.

Mass property calculations,
Importing models from other sources (TRUEGRID, PATRAN, IDEAS, IGES and NASTRAN formats),
Interactive viewing and graphical inspection of boundary conditions, etc.,
Model editing,
General purpose mesh generation,
Importing LS-DYNA and DYNA3D models in a variety of older formats,
Complex surface treatment including NURB surfaces,
Parametric modeling.

Capabilities specialized to automotive applications:

Airbag folding and inspection,
Occupant positioning,
Seat belt positioning (both beam and shells),
Merging of occupants, airbags and belts with car models.

LS-POST

LS-POST processes output from LS-DYNA. LS-POST 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-POST can compute a variety of strain measures, reaction forces along constrained boundaries, and momenta. LS-POST is operational on the CRAY, SUN, DEC, IBM RS6000, SGI, HP and PC computers.

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.

Because of the difficulty in handling one large file, an alternative method for obtaining printed output is also available. Many ASCII databases are created at the user's option containing such information as cross-sectional forces, rigidwall forces, nodal point data, element integration point data, global data like total internal and kinetic energy, material energies, nodal interface forces, resultant interface forces, single point constraint forces, as well as files that are compatible with MOVIE.BYU and the Cray Research developed post-processor, MPGS. A SMUG animator database and a NASTRAN BDF file is written for users at General Motors. Each ASCII database is written at its own unique output interval defined in the user input.

File Organization

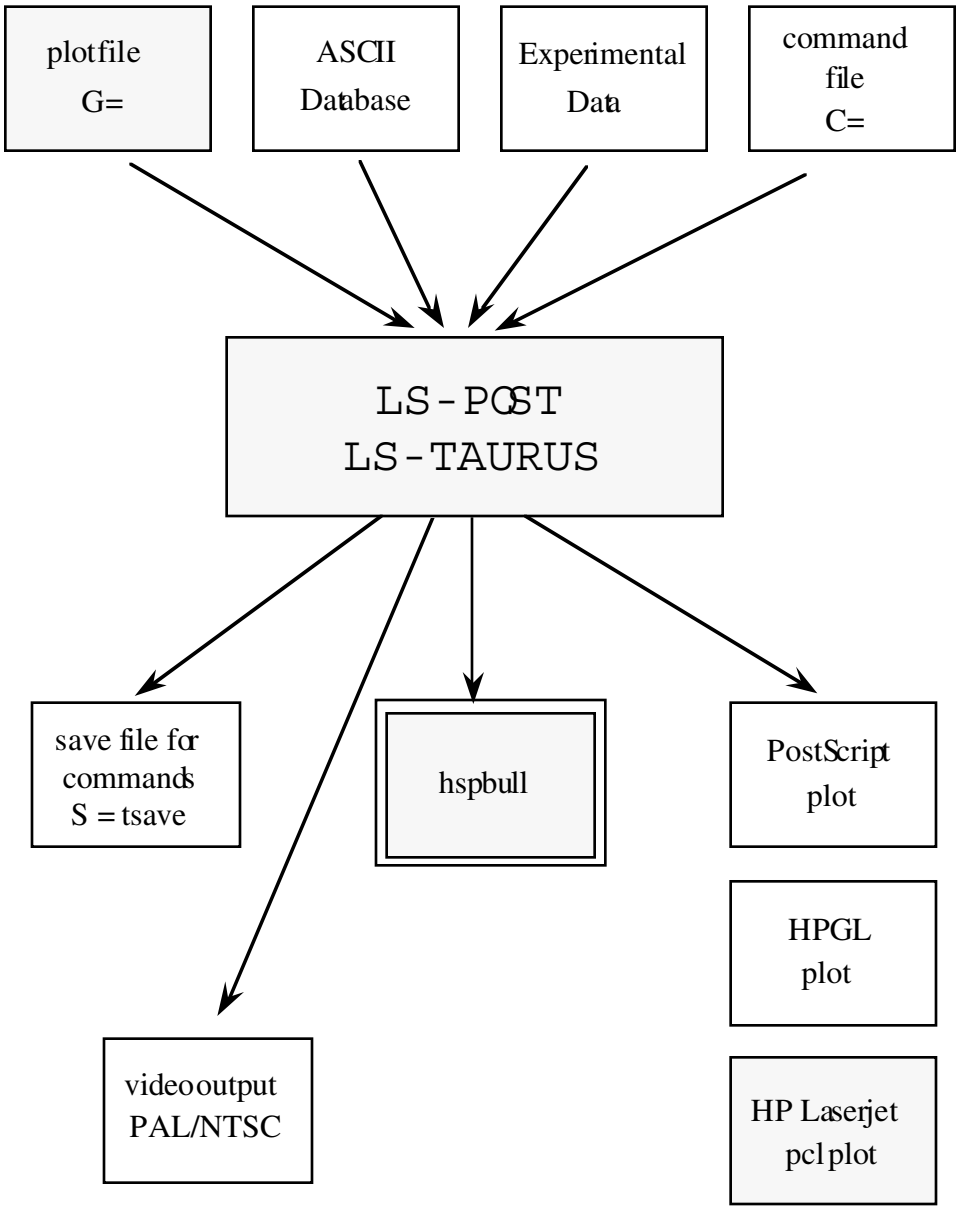


Figure I.4

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 I.5 illustrates the relative cost of the various shell formulations in LS-DYNA3D.

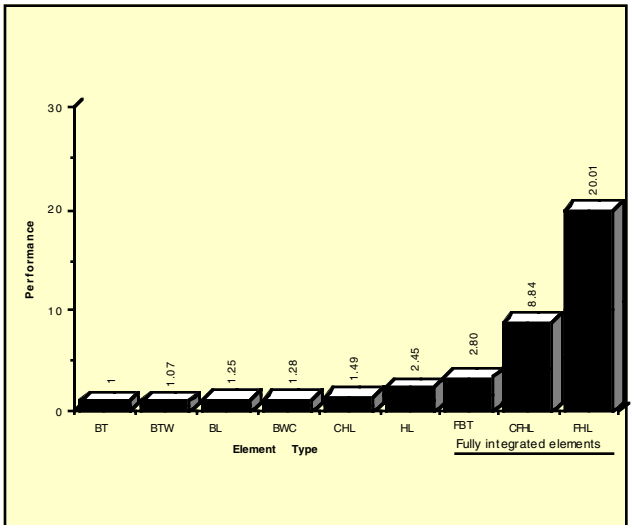


Figure I.5. Relative cost of the four noded shells available in LS-DYNA where BT is the Belytschko-Tsay shell, BTW is the Belytschko-Tsay shell with the warping stiffness taken from the Belytschko-Wong-Chiang, BWC, shell. The BL shell is the Belytschko-Leviathan shell. CHL denotes the Hughes-Liu shell, 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) = $\dfrac{1\,(\text{length unit})}{\left[1\,(\text{time unit})\right]^2}$

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	VEZ		
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 control cards in this section are defined in alphabetical order:

*AIRBAG_*OPTION1*_{*OPTION2*}_{*OPTION3*}_<OPTIONAL NUMERIC ID>
*AIRBAG_INTERACTION
*AIRBAG_REFERENCE_GEOMETRY_*OPTION*_*OPTION*

***AIRBAG**_*OPTION1*_{*OPTION2*}

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

The OPTIONAL_NUMERIC_ID is a unique number used only for identification of the airbag in the definition of airbag interaction via *AIRBAG_INTERACTION. The numeric ID is not used for any other purpose. To define an airbag using the Wang Nefske thermodynamic relationship with an ID of 25 the keyword is *AIRBAG_WANG_NEFSKE_25. For more information on these models the papers by Wang [1988, 1995] and Nusholtz [1991, 1996] are recommended.

*AIRBAG

Card Format

	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 IDs.
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 B.
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:

$$V_{cvolume} = (V_{sca} V_{femodel}) - V_{ini}$$

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

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 B. 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 Format

	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)

	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

	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.			

	1	2	3	4	5	6	7	8
Variable	DVX	DVY	DVZ	DVMAG				
Type	F	F	F	F				
Default	0.	0.	0.	0.				

	1	2	3	4	5	6	7	8
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 cummulative 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

	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:

$$Pressure = \beta \frac{CN}{Relative\ Volume}$$

$$Relative\ Volume = \frac{Current\ Volume}{Initial\ Volume}$$

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 usedto 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
CP	Heat capacity at constant pressure
CV	Heat capacity at constant volume
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
RO	Ambient density, ρ

VARIABLE	DESCRIPTION
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

	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:

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

Additional 4 cards are required for all WANG_NEFSKE models

Card 1

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 <u>if and only if</u> 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			
Type	F	F	F	F	F			
Default	none	none	none	none	none			

Define the following card <u>if and only if</u> 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
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 overides 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	Load curve number defining the vent orifice coefficient which applies to exit hole as a function of time. A nonzero value for C23 overrides LCC23.
A23	Vent orifice area which applies to exit hole. Set 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.
PE	Ambient pressure

VARIABLE	DESCRIPTION
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 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.
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 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/^oK)
B	Second heat capacity coefficient of inflator gas, (e.g., Joules/mole/$^oK^2$)

VARIABLE	DESCRIPTION
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)
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

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

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

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

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

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

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:

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

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:

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

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.

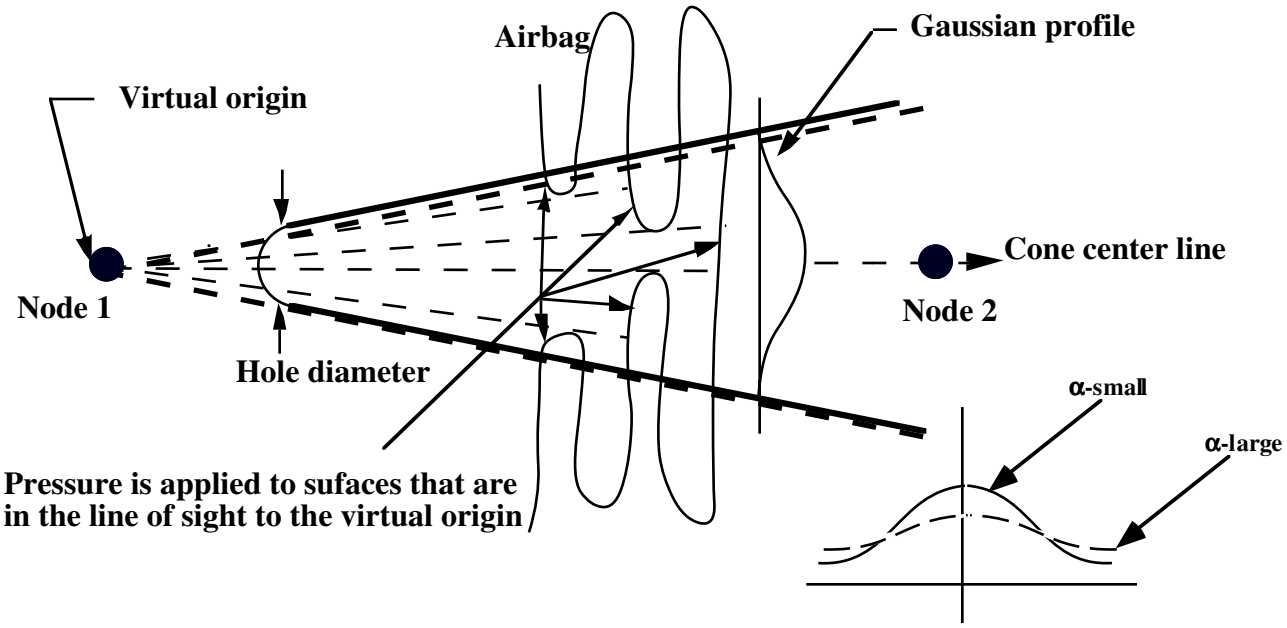
Remarks:

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.

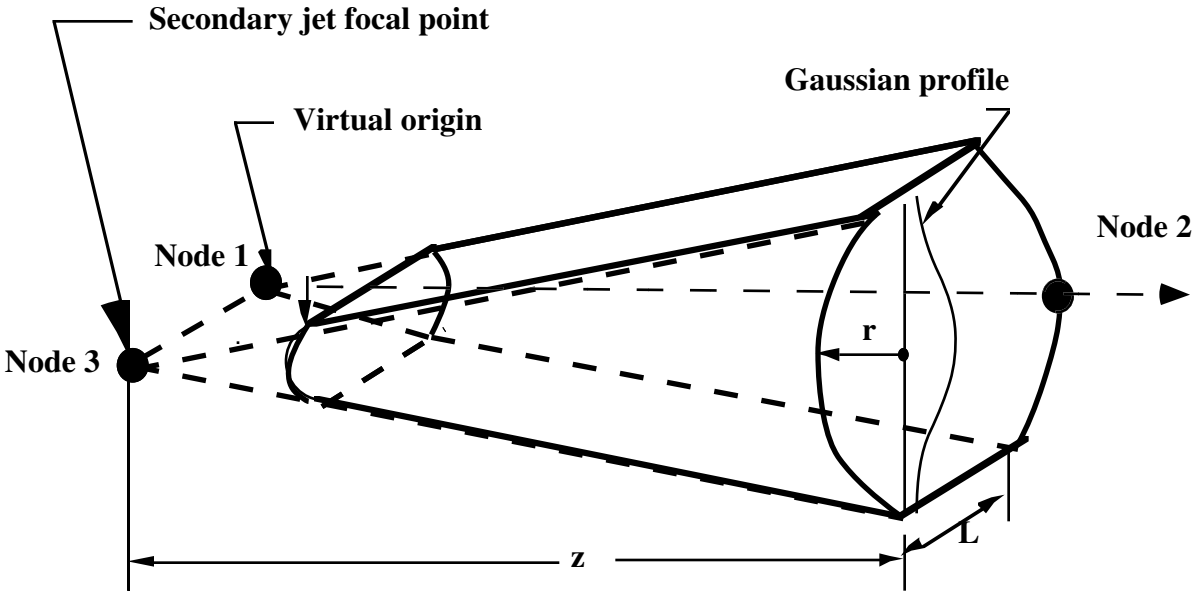
The jetting option provides a simple model to simulate the real pressure distribution in the airbag during the breakout and early unfolding phase. Only the sufaces that are in the line of sight to the virtual origin have an increased pressure applied. With the optional load curve LCRJV, the pressure 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.



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.

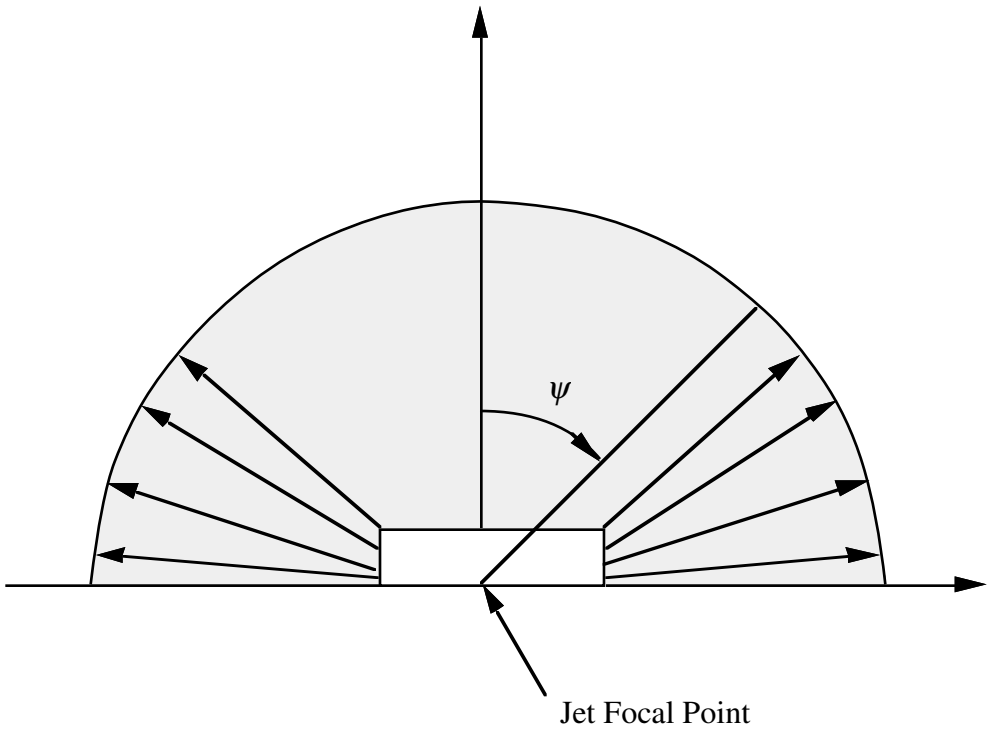


Figure 1.2 Multiple jet model for driver's side airbag.

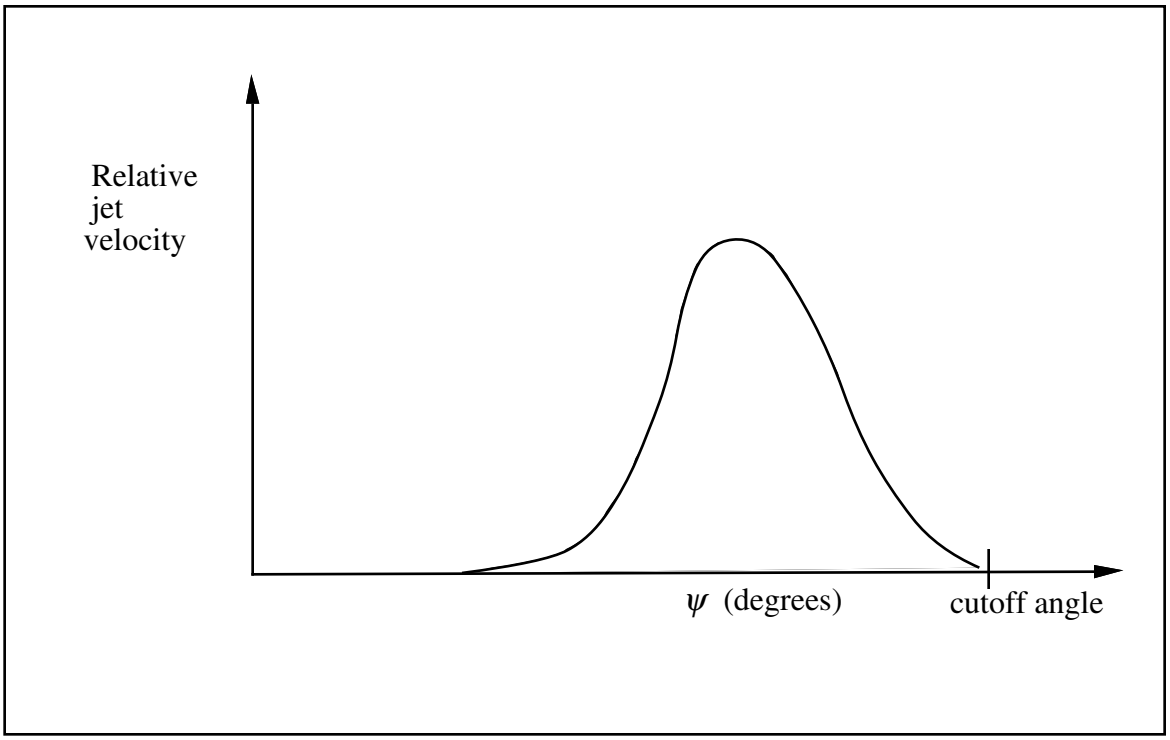


Figure 1.3 Normalized jet velocity versus angle for multiple jet driver's side airbag.

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:

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

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 centreline, $v_{r=0,z}$, 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 calulated 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

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

$$F = dis\tan ce\ between\ jet\ focii\ for\ a\ passenger\ jet$$

Finally, the pressure exerted on an airbag element in view of the jey is given y:

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

By combining the eqations 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 support bracketry to be modelled. The jetting force is written to the ASCII ABSTAT file and the binary XTF file

Additional card required for LINEAR_FLUID option

	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

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.

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,

$V_0(t) = V_0(t) = \frac{M(t)}{\rho}$ 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 the equation for $M(t)$. The mass flow should always be defined as positive since the output flow is substracted.

Additional cards required for HYBRID and HYBRID_JETTING options

	1	2	3	4	5	6	7	8
Variable	ATMOST	ATMOSP	ATMOSD	GC	CC			
Type	F	F	F	F	F			
Default	none	none	none	none	1.0			

	1	2	3	4	5	6	7	8
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

	1	2	3	4	5	6	7	8
Variable	OPT	PVENT	NGAS					
Type	I	F	I					
Default	none	none	none					

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

	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

	1	2	3	4	5	6	7	8
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.
C23	Vent orifice coefficient which applies to exit hole. Set to zero if LCC23 is defined below.
LCC23	Load curve number defining the vent orifice coefficient which applies to exit hole as a function of time. A nonzero value for C23 overrides LCC23.
A23	Vent orifice area which applies to exit hole. Set to zero if LCA23 is defined below.

VARIABLE	DESCRIPTION
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 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.
PVENT	Gauge pressure when venting begins
NGAS	Number of gas inputs to be defined below. (Including initial air)
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

VARIABLE	DESCRIPTION
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 2 cards are required for HYBRID_JETTING 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	1	2	3	4	5	6	7	8
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

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

VARIABLE	DESCRIPTION
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.

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 sufaces 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.

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

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)
NGAS	Number of gas inputs to be defined below. (Including initial air)
ATMT	Atmospheric temperature.
ATMP	Atmospheric pressure
RG	Universal gas constant
HCONV	Convection heat transfer coefficient
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 thermodynamic 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.
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). However, the CHEMKIN database file with file name chemkin, must reside in the same directory that you are running LS-DYNA.

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

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

card 2

	1	2	3	4	5	6	7	8
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

	1	2	3	4	5	6	7	8
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 $$c_p = \frac{\overline{R}}{M}\left(a + bT + cT^2 + dT^3 + eT^4\right)$$

$\bar{R}$ = 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_INTERACTION

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

Define one card for each airbag interaction definition

	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, except that choked flow is currently ignored. This will be added later.

***AIRBAG_REFERENCE_GEOMETRY**_*OPTION_OPTION*

Available options include:

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.

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

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

Card Format (I8,3E16.0)

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 number
X	x coordinate
Y	y coordinate
Z	z coordinate

*ALE

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

*ALE_MULTI-MATERIAL_GROUP

*ALE_REFERENCE_SYSTEM_CURVE

*ALE_REFERENCE_SYSTEM_GROUP

*ALE_REFERENCE_SYSTEM_NODE

*ALE_REFERENCE_SYSTEM_SWITCH

*ALE_SMOOTHING

For other input information related to the ALE capability, see keywords: *CONTROL_ALE, *INITIAL_VOID and *SECTION_SOLID_ALE.

***ALE_MULTI-MATERIAL_GROUP**

Purpose: The following input defines the PART ID's of each multi-material group. Elements containing materials of the same group are treated as single material elements. Currently, this option allows up to three (3) different material goups to be mixed within the same element.

For each group define the following cards.

NOTE: THE TOTAL NUMBER OF GROUPS MUST BE LESS THAN OR EQUAL TO THREE.

Card Format

	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. The multi-material option defined here and void materials, see *INITIAL_VOID, are incompatible and cannot be used together in the same run.

Example

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, oil, water and air, that is, the number of ALE groupls, 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 is used to define a prescribed motion of an ALE mesh, following 12 pre-defined load curves. The command must be combined with *ALE_ REFERENCE_ SYSTEM_GROUP.

Card Format

Card 1

Variable	ID							
Type	I							
Default	none							

Card 2

Variable	LC1	LC2	LC3	LC4	LC5	LC6	LC7	LC8
Type	I	I	I	I	I	I	I	I
Default	none	none	none	none	none	none	none	none

Card 3

Variable	LC9	LC10	LC11	LC12				
Type	I	I	I	I				
Default	none	none	none	none				

VARIABLE	DESCRIPTION
ID	Curve set ID.
LC1...LC12	Load curve ID's.

Remark:

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 LC1 at time t etc.

*ALE_REFERENCE_SYSTEM_GROUP

Purpose: This command is used to assign a specific reference system type (Lagrangian, Eulerian or ALE) to a set of nodes.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	SID	STYPE	PRTYPE	PRID	BCTRAN	BCEXP	BCROT	ICOORD
Type	I	I	I	I	I	I	I	I
Default	none	0	0	0	0	0	0	0

Card 2	1	2	3	4	5	6	7	8
Variable	XC	YC	ZC	EXPLIM				
Type	F	F	F	F				
Default	0.0	0.0	0.0	inf.				

VARIABLE	DESCRIPTION
SID	Set ID.
STYPE	Set type: EQ.0: part set, EQ.1: part, EQ.2: node set, EQ.3: segment set.

VARIABLE	DESCRIPTION
PRTYPE	Reference system type: 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 coordinate system defined by three user defined nodes, see *ALE_REFERENCE_SYSEM_NODE, EQ.6: Switching in time between different reference system types, see *ALE_REFERENCE_SYSEM_SWITCH, EQ.7: Automatic mesh expansion in order to enclose up to twelve user defined nodes, see *ALE_REFERENCE_SYSEM_NODE.
PRID	ID of switch list, node group or curve group (PRTYPE 3, 5, 6 or 7).
BCTRAN	Translational constraints (PRTYPE 3, 4, 5 and 7): 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 ranslation, EQ.6: constrained z and x translation, EQ.7: constrained x, y, and z translation.
BCEXP	Mesh expansion constraints (PRTYPE 3, 4, 5 and 7): 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.
BCROT	Mesh rotation constraints (PRTYPE 3,4 5 and 7): 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.

VARIABLE	DESCRIPTION
ICOORD	Center of mesh expansion and rotation (PRTYPE 3, 4, 5 and 7): EQ.0: at center of gravity, EQ.1: at (XC,YC,ZC).
XC,YC,ZC	Center of mesh expansion.
EXPLIM	Limit ratio for mesh expansion and shrinkage. 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.

*ALE_REFERENCE_SYSTEM_NODE

Purpose: The purpose of this command is to define a group of nodes that control the motion of an ALE mesh.

Card Format

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

Card Format

Card 1	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
Default	none	none	none	none	none	none	none	none

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	NID9	NID10	NID11	NID12				
Type	I	I	I	I				
Default	none	none	none	none				

VARIABLE	DESCRIPTION
ID	Node group ID for PRTYPE 3 or 7, see *ALE_REFERENCE_SYSTEM_GROUP.
NID1...NID12	User specified nodes.

Remark:

For PRTYPE=3 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:

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

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

*ALE_REFERENCE_SYSTEM_SWITCH

Purpose: This command allows for switching between Lagrangian, Eulerian and ALE formulations during the simulation.

Card Format

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

Card Format

Card 2	1	2	3	4	5	6	7	8
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 Format

Card 3	1	2	3	4	5	6	7	8
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 Format

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.
TYPE1...TYPE8	Reference system types: 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 coordinate system defined by three user defined nodes, see *ALE_REFERENCE_SYSEM_NODE, EQ.7: Automatic mesh expansion in order to enclose up to twelve user defined nodes, see *ALE_REFERENCE_SYSEM_NODE.
ID1...ID7	ID of node or curve group (PRTYPE 3, 5 or 7).

Remark:

At time T2 the reference system type is switched from TYPE2 to TYPE3 etc.

*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 Format

	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

Remark:

Abritrary 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.

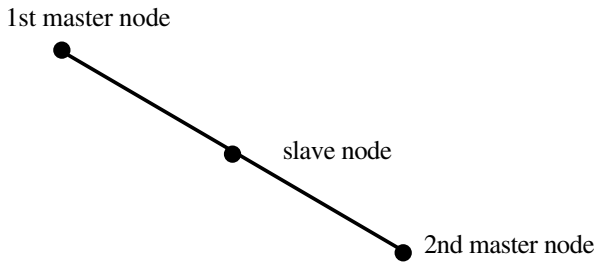


Figure 2.1 This simple constraint, which ensures that a slave node remains on a straight line between two master nodes, is sometimes necessary during ALE smoothing.

*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_AMBIENT_EOS**

BOUNDARY_CONVECTION_*OPTION

***BOUNDARY_CYCLIC**

BOUNDARY_ELEMENT_METHOD_*OPTION

BOUNDARY_FLUX_*OPTION

***BOUNDARY_NON_REFLECTING**

***BOUNDARY_NON_REFLECTING_2D**

BOUNDARY_PRESCRIBED_MOTION_*OPTION

BOUNDARY_PRESSURE_OUTFLOW_*OPTION

BOUNDARY_RADIATION_*OPTION

***BOUNDARY_SLIDING_PLANE**

BOUNDARY_SPC_*OPTION

***BOUNDARY_SYMMETRY_FAILURE**

BOUNDARY_TEMPERATURE_*OPTION

***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 Format

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

VARIABLE	DESCRIPTION
SSID	Segment set ID, see *SET_SEGMENT

Remark:

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_AMBIENT_EOS

Purpose: Define load curve driven internal energy and relative volume for ambient elements.

Card Format

	1	2	3	4	5	6	7	8
Variable	PID	LC1	LC2					
Type	I	I	I					
Default	none	none	none					

VARIABLE	DESCRIPTION
PID	Part ID.
LC1	Load curve ID for specific internal energy.
LC2	Load curve ID for relative volume.

BOUNDARY_CONVECTION_*OPTION

Available options are:

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 Format (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 Format (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 Format (Card 2 of 2)

Card 1	1	2	3	4	5	6	7	8
Variable	HLCID	HMULT	TLCID	TMULT				
Type	I	F	I	F				
Default	none	1.0	none	1.0				

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_∞

Remarks:

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

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

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.

Remark:

Only globally defined axes of rotation are possible.

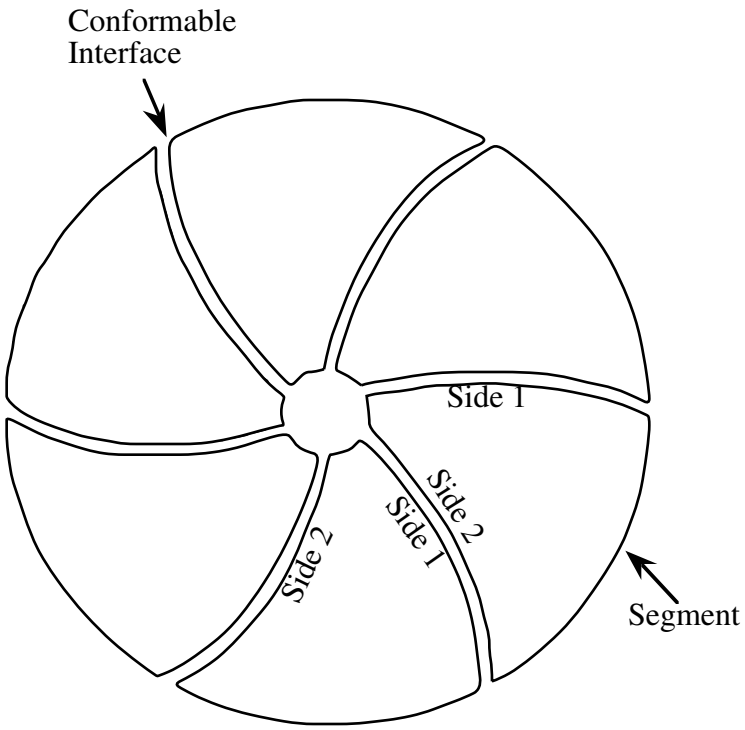


Figure 3.1 With cyclic symmetry only one segment is modeled.

BOUNDARY_ELEMENT_METHOD_*OPTION

Available options are:

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 Theoretical 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.

	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.

VARIABLE	DESCRIPTION
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.

	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 susbsonic 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 Theoretical 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:

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

	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.

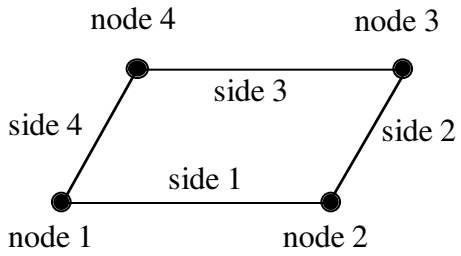


Figure 3.2 Each segment has 4 sides.

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.

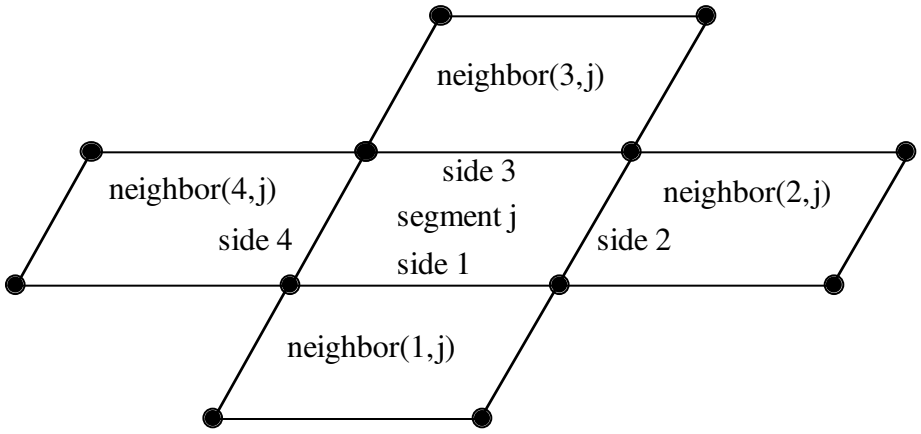


Figure 3.3 Typical neighbor specification.

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.

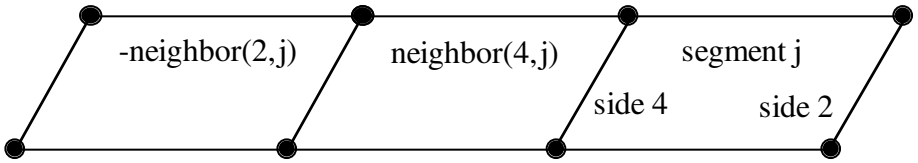


Figure 3.4 If neighbor(2,j) is a negative number it is assumed to lie on the opposite side of neighbor(4,j) as segment j.

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 occaisionally 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).

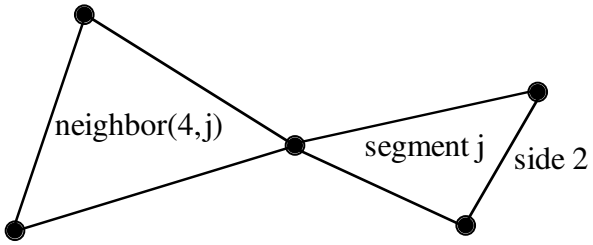


Figure 3.5 Sometimes another triangular boundary element segment can be used as neighbor(4,j).

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 the following card:

Define one 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 delata 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 the following cards:

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

	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 are:

SEGMENT

SET

Purpose: Define flux 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 Format (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 Format (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 Format (Card 2 of 2)

Card 1	1	2	3	4	5	6	7	8
Variable	LCID	MLC1	MLC2	MLC3	MLC4			
Type	I	F	F	F	F			
Default	none	1.0	1.0	1.0	1.0			

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.

Remarks:

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.

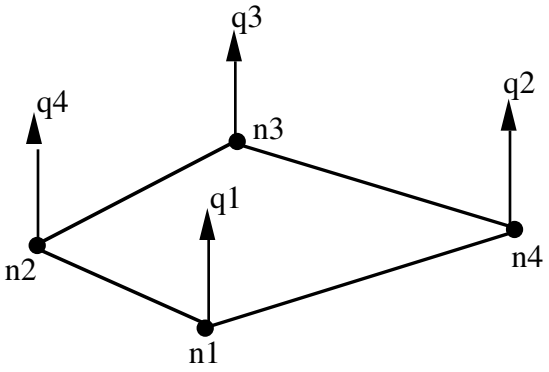


Figure 3.6. Nodal number determines outward normal.

*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 Format

	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	2	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

VARIABLE	DESCRIPTION
TSUBC	Time interval for MCOL subcycling. EQ. 0.0: no subcycling
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-POST, 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 Format

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.

*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 Format

	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.

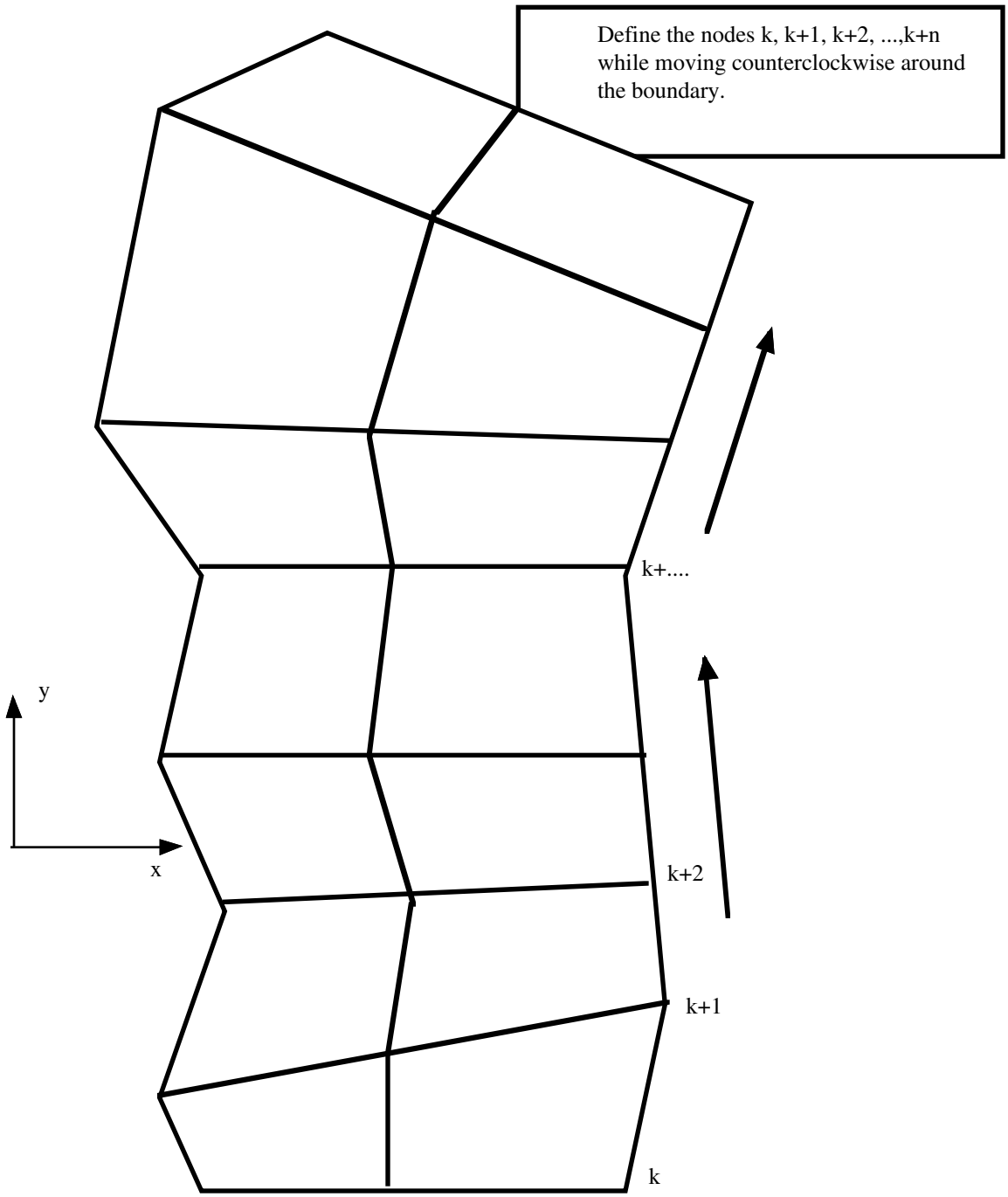


Figure 3.7 When defining a transmitting boundary in 2D define the node numbers in the node set in consecutive order while moving counterclockwise around the boundary.

BOUNDARY_OUTFLOW_CFD_*OPTION

Available options include:

SEGMENT

SET

Purpose: Define "passive" outflow boundary conditions for the incompressible flow solvers. These boundary conditions are active only when SOLN=4 or SOLN=5 on the *CONTROL_SOLUTION keyword.

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

Card Format

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 Format

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.

Remarks:

In the incompressible flow solver, the role of the outflow boundary conditions is to provide a computational boundary that is passive - particularly in the presence of strong vortical flow structures. Typically, this boundary condition is applied at boundaries that have been artificially imposed to emulate far-field conditions in a large physical domain.

BOUNDARY_PRESCRIBED_CFD_*OPTION

Available options include:

NODE

SET

Purpose: Define an imposed nodal variable (velocity, temperature, species, etc.) on a node or a set of nodes for the Navier-Stokes flow solver. Do not use the NODE option in r-adaptive problems since the node ID's may change during the adaptive step.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	*type*ID	DOF		LCID	SF			
Type	I	I		I	F			
Default	none	none		none	1.			

VARIABLE	DESCRIPTION
*type*ID	Node ID (NID) or nodal set ID (NSID), see *SET_NODE.
DOF	Applicable degrees-of-freedom: EQ. 101: x-velocity, EQ. 102: y-velocity, EQ. 103: z-velocity, EQ. 104: Temperature, EQ. 107: turbulent kinetic energy, EQ. 110: turbulent eddy viscosity, EQ. 121: Species mass fraction-1, EQ. 122: Species mass fraction-2, EQ. 123: Species mass fraction-3, EQ. 124: Species mass fraction-4, EQ. 125: Species mass fraction-5, EQ. 126: Species mass fraction-6, EQ. 127: Species mass fraction-7, EQ. 128: Species mass fraction-8, EQ. 129: Species mass fraction-9, EQ. 130: Species mass fraction-10, EQ. 201: x-, y-, z-velocity, EQ. 202: x-, y-velocity, EQ. 203: x-, z-velocity, EQ. 204: y-, z-velocity,

EQ. 301: All species.

LCID	Load curve ID to describe motion value versus time, see *DEFINE_CURVE.
SF	Load curve scale factor. (default=1.0)

Remarks:

The prescription of all nodal variables in the incompressible flow solver are defined using this keyword. It is similar in function to the *BOUNDARY_PRESCRIBED_MOTION_*OPTION* keyword, but permits the specification of boundary conditions for velocities, mass concentration of each species, temperature, turbulent kinetic energy, etc. Additionally, the keyword optionally provides for prescribing all velocities, for example on a no-slip and no-penetration surface. Multiple instances of the keyword permit individual nodal variables to be prescribed using independent load curves and scale factors.

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$  *BOUNDARY_PRESCRIBED_CFD_SET
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  A set of nodes is given a prescribed velocity in the
$  x-direction according to a specified vel-time curve (which is scaled).
$
*BOUNDARY_PRESCRIBED_CFD_SET
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$     nsid      dof                 lcid       sf
         4      101                    8      2.0
$
$     nsid = 4    nodal set ID number, requires a *SET_NODE_option
$      dof = 101  x-velocity is prescribed
$     lcid = 8    velocity follows load curve 8, requires a *DEFINE_CURVE
$       sf = 2.0  velocity specified by load curve is scaled by 2.0
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

BOUNDARY_PRESCRIBED_MOTION_*OPTION

Available options include:

NODE

SET

RIGID

RIGID_LOCAL

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.

Card Format

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
NID, NSID, PID	Node ID (NID), nodal set ID (NSID), SEE *SET_NODE, or part ID (PID), see *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 the vector given by the VID. Rotation about the normal axes is permitted. EQ.-8: rotational motion about the vector given by the 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. 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. 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. 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. 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. 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.
VAD	Velocity/Acceleration/Displacement flag: EQ.0: velocity (rigid bodies and nodes), EQ.1: acceleration (nodes only), EQ.2: displacement (rigid bodies and nodes). EQ.3: velocity versus displacement (rigid bodies) EQ.4: relative displacement (rigid bodies only)
LCID	Load curve ID to describe motion value versus time, see *DEFINE_CURVE.
SF	Load curve scale factor. (default=1.0)

VARIABLE	DESCRIPTION
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.
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:

Arbitrary translations and rotations are possible. Rotations around local axis can be defined either by setting DOF=8 or by using the offset option of DOF>8. 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.

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$   *BOUNDARY_PRESCRIBED_MOTION_SET
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  A set of nodes is given a prescribed translational velocity in the
$  x-direction according to a specified vel-time curve (which is scaled).
$
*BOUNDARY_PRESCRIBED_MOTION_SET
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$     nsid       dof       vad      lcid        sf       vid     death
         4         1         0         8       2.0
$
$     nsid = 4    nodal set ID number, requires a *SET_NODE_option
$      dof = 1    motion is in x-translation
$      vad = 0    motion prescribed is velocity
$     lcid = 8    velocity follows load curve 8, requires a *DEFINE_CURVE
$       sf = 2.0  velocity specified by load curve is scaled by 2.0
$      vid        not used in this example
$    death        use default (essentially no death time for the motion)
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$

$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$   *BOUNDARY_PRESCRIBED_MOTION_RIGID
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  A rigid body is given a prescribed rotational displacement about the
$  z-axis according to a specified displacement-time curve.
$
*BOUNDARY_PRESCRIBED_MOTION_RIGID
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$      pid       dof       vad      lcid        sf       vid     death
        84         7         2         9                          14.0
$
$      pid = 84   apply motion to part number 84
$      dof = 7    rotation is prescribed about the z-axis
$      vad = 2    the prescribed motion is displacement (angular)
$     lcid = 9    rotation follows load curve 9, requires a *DEFINE_CURVE
$                     (rotation should be in radians)
$       sf        use default (sf = 1.0)
$      vid        not used in this example
$    death = 14   prescribed motion is removed at 14 ms (assuming time is in ms)
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

*BOUNDARY_PRESSURE_CFD_SET

Purpose: Apply a pressure load over each segment in a segment set for the incompressible flow solver. The pressure convention follows Figure 3.8.

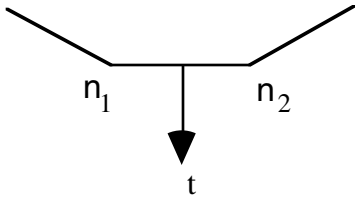
Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	SSID	LCID	P					
Type	I	I	F					
Default	none	none	none					
Remarks		1	1					

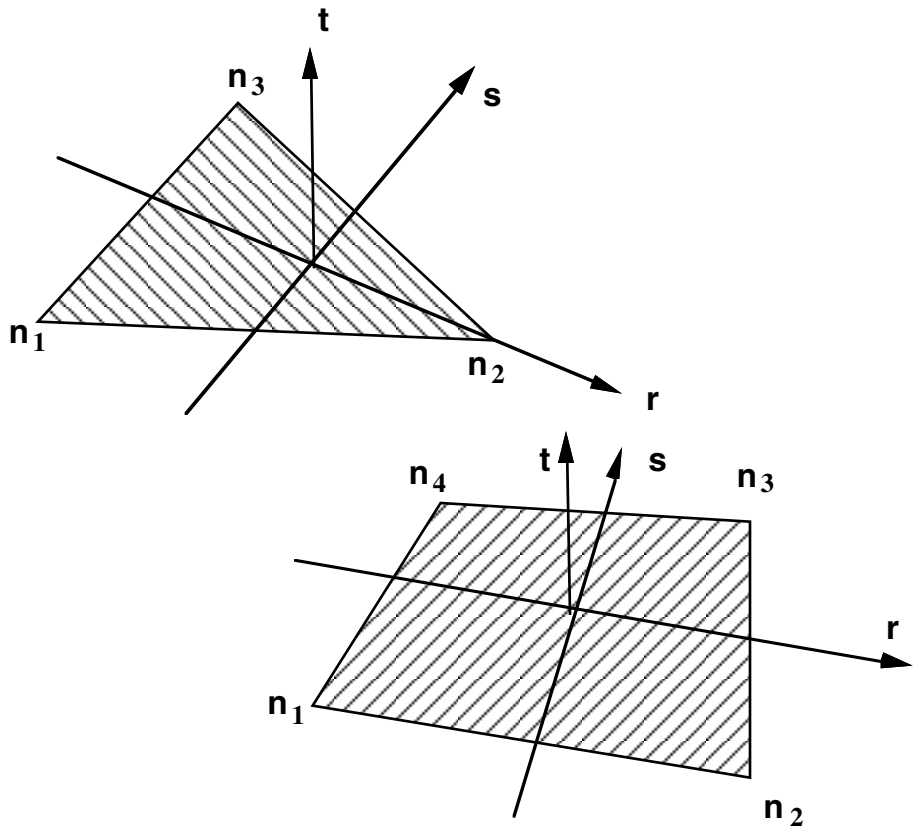
VARIABLE	DESCRIPTION
SSID	Segment set ID, see *SET_SEGMENT
LCID	Load curve ID, see *DEFINE_CURVE
P	Pressure to be applied.

Remarks:

1. The load curve multipliers may be used to increase or decrease the pressure amplitude. The time value is not scaled.



a) 2-Dimensional definition for pressure boundary segments.



b) 3-Dimensional definition for pressure boundary segments

Figure 3.8. Nodal numbering for pressure boundary segments. 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.

BOUNDARY_PRESSURE_OUTFLOW_*OPTION

Available options are

SEGMENT

SET

Purpose: Define pressure outflow boundary conditions. These boundary conditions are attached to solid elements using the Eulerian ambient formulation (7) and defined to be pressure outflow ambient elements (3). See *SECTION_SOLID_*OPTION*.

For the SET option define the following card

Card Format

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 Format

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_RADIATION_*OPTION

Available options are:

SEGMENT

SET

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

There are two types of radiation boundary conditions that can be specified.

1. The first type, models radiation exchange between a finite element surface segment and the environment at temperature T_∞. The view factor between the finite element surface segment and the environment is 1.

2. The second type, models the radiation exchange between all the finite element segments that define a completely closed volume. The view factors between all the finite element segments defining the enclosure must be calculated and stored in a file named **viewfl**. With the _SET option multiple independent boundary radiation enclosures may be defined.

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

Card Format (Card 1 of 2)

Card 1	1	2	3	4	5	6	7	8
Variable	SSID	TYPE	RAD_GRP	FILE_NO				
Type	I	I	I	I				
Default	none	1	0	0				

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

Card Format (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			

Define the following card boundary radiation type 1 :

Card Format (Card 2 of 2)

	1	2	3	4	5	6	7	8
Variable	RFLCID	RFMULT	TILCID	TIMULT				
Type	I	F	I	F				
Default	none	1.0	none	1.0				

Define the following card boundary radiation type 2 :

Card Format (Card 2 of 2)

	1	2	3	4	5	6	7	8
Variable	SELCID	SEMULT						
Type	I	F						
Default	none	1.0						

VARIABLE	DESCRIPTION
SSID	Segment set ID, see *SET_SEGMENT
N1,N2...	Node ID's defining segment
TYPE	Radiation type: EQ.1: Radiation boundary to environment (default), EQ.2: Radiation in enclosure. The following two parameters are used for enclosure radiation definitions defined using the _SET option.
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.

VARIABLE	DESCRIPTION
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.
RFLCID	Load curve ID for radiation factor *f*, see *DEFINE_CURVE: GT.0: function versus time, EQ.0: use constant multiplier value, RFMULT, LT.0: function versus temperature.
RFMULT	Curve multiplier for *f*, see *DEFINE_CURVE
TILCID	Load curve ID for T_∞ versus time, see *DEFINE_CURVE: EQ.0: use constant multiplier, TIMULT.
TIMULT	Curve multiplier for T_∞, see *DEFINE_CURVE
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

Remarks:

A radiation boundary condition is calculated using a radiant-heat-transfer coefficient. Set $\dot{q}'' = h_r\ (T - T_\infty)$, where h_r is a radiant-heat-transfer coefficient defined as

$$h_r = f(T + T_\infty)(T^2 + T^2{}_\infty)$$

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.

The Stefan Boltzmann constant must be defined for radiation in enclosure (type 2). See *CONTROL_THERMAL_SOLVER.

A file, with the name **viewfl** or **viewfl_****, containing the surface-to-surface area*view factor products (i.e., A_iF_{ij}) must be defined. The A_iF_{ij} products must be stored in this file by row and formatted as 8E10.0.

row 1	A_1F_{11}	A_1F_{12}	••••	A_1F_{1n}
row 2	A_2F_{21}	A_1F_{22}	••••	A_2F_{2n}
•	•	•	••••	•
row n	A_nF_{n1}	A_nF_{n2}	••••	A_nF_{nn}

The order of segments in the view factor file follow the order the sets are assigned to the boundary radiation definition. For example with the following definition,

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$   *BOUNDARY_RADIATION_SET
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Make boundary enclosure radiation groups 8 and 9.
$
*BOUNDARY_RADIATION_SET
         15          2          9         10
        1.0        1.0
*BOUNDARY_RADIATION_SET
         12          2          9         10
        1.0        1.0
*BOUNDARY_RADIATION_SET
         13          2          8         21
        1.0        1.0
$
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

Enclousre 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 file view_10. Enclosure radiation group 8 is composed of the segment in SEGMENT_SET 13 and reads the view factors from viewfl_21.

For the zero group definition the order is segments defined by
*BOUNDARY_RADIATION_SEGMENT followed by segments defined by
*BOUNDARY_RADIATION_SET.

*BOUNDARY_SLIDING_PLANE

Purpose: Define a sliding symmetry plane. This option applies to continuum domains modeled with solid elements.

Card Format

	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_*OPTION

Available options include:

NODE

SET

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.

Card Format

	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

VARIABLE	DESCRIPTION
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.

Remark:

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_SYMMETRY_FAILURE

Purpose: Define a symmetry plane with a failure criterion. This option applies to continuum domains modeled with solid elements.

Card Format

	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 are:

NODE

SET

Purpose: Define temperature boundary conditions for a thermal or coupled thermal/structural analysis.

Card Format

	1	2	3	4	5	6	7	8
Variable	NID/SID	LCID	CMULT					
Type	I	I	F					
Default	none	0	1.0					

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

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 workpiece is moving.

Card 1 Format

	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

	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)

	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	reward 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 (J. Goldak, "A New Finite Element Model for Welding Heat Sources", Metallurgical Transactions B, Volume 15B, June 1984, pp 299-305). Heat is generated in a 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 Format

	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 strutural 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.

*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

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 positoning file is essential with this feature, see Appendix K for further details.

OPTION specifies the human subject type. The male and female type represent adults while the child is genderless.

MALE

FEMALE

CHILD

Card Format (Card 1 of 2)

	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 Format (Card 2 of 2)

Card 1	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 K 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.

	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.

VARIABLE	DESCRIPTION
SCFi	Scale factor applied to the load curve of the i-th joint degree of freedom.

Card 2 - Required.

	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.

	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.

	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

*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. 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. The dummy interacts with the finite element structure through contact interfaces.

Card Format (Card 1 of 2)

	1	2	3	4	5	6	7	8
Variable	DID	SIZE	UNITS	DEFRM	VX	VY	VZ	
Type	I	F	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 percemtile adult EQ.3: 95th percentile adult
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.2: deformable abdomen/headskin EQ.7: deformable abdomen/jacket/headskin
VX,VY,VZ	Initial velocity of the dummy in the global x, y and z directions.

Card Format (Card 2 of 2)

Card 1	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.

The following options are available.

LUMBAR

LOWER_NECK

UPPER_NECK

LEFT_SHOULDER

RIGHT_SHOULDER

LEFT_ELBOW

RIGHT_ELBOW

LEFT_WRIST

RIGHT_WRIST

LEFT_HIP

RIGHT_HIP

LEFT_KNEE

RIGHT_KNEE

LEFT_ANKLE

RIGHT_ANKLE

RIBCAGE

Card 1 - Required.

	1	2	3	4	5	6	7	8
Variable	DID	Q1	Q2	Q3	FRIC			
Type	F	F	F	F	F			

Card 2 - Required.

	1	2	3	4	5	6	7	8
Variable	C1	ALO1	BLO1	AHI1	BHI1	QLO1	QHI1	
Type	F	F	F	F	F	F	F	

Card 3 - Required. Left blank if joint has only one degree of freedom.

	1	2	3	4	5	6	7	8
Variable	C2	ALO2	BLO2	AHI2	BHI2	QLO2	QHI2	
Type	F	F	F	F	F	F	F	

Card 4 - Required. Left blank if the joint has only two degrees of freedom.

	1	2	3	4	5	6	7	8
Variable	C3	ALO3	BLO3	AHI3	BHI3	QLO3	QHI3	
Type	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 K 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 coeffient for the low regime spring of the joint's i-th DOF.
AHIi	Linear coeffient for the high regime spring of the joint's i-th DOF.
BHIi	Cubic coeffient for the high regime spring of the joint's i-th DOF.
QLOi	Value for which the low regime spring definition becomes active.
QHIi	Value for which the high regime spring definition becomes active.

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$  *COMPONENT_HYBRIDIII_JOINT_LEFT_ANKLE
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  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
```

*COMPONENT

*CONSTRAINED

The keyword *CONSTRAINED provides a way of constraining degrees of freedom to move together in some way. The keyword control cards in this section are defined in alphabetical order:

***CONSTRAINED_ADAPTIVITY**

CONSTRAINED_EXTRA_NODES_*OPTION

CONSTRAINED_GENERALIZED_WELD_*OPTION

***CONSTRAINED_GLOBAL**

***CONSTRAINED_INTERPOLATION**

CONSTRAINED_JOINT_*OPTION

CONSTRAINED_JOINT_STIFFNESS_*OPTION

***CONSTRAINED_LAGRANGE_IN_SOLID**

***CONSTRAINED_LINEAR**

***CONSTRAINED_NODAL_RIGID_BODY_{*OPTION*}**

***CONSTRAINED_NODE_SET**

***CONSTRAINED_POINTS**

***CONSTRAINED_RIGID_BODIES**

***CONSTRAINED_RIGID_BODY_STOPPERS**

***CONSTRAINED_RIVET**

***CONSTRAINED_SHELL_TO_SOLID**

***CONSTRAINED_SPOTWELD_{*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 Format

	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_EXTRA_NODES_*OPTION

Available options include:

NODE

SET

Purpose: Define extra nodes for rigid body.

Card Format

	1	2	3	4	5	6	7	8
Variable	PID	NID/NSID						
Type	I	I						
Default	none	none						

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.

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 nodes 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

Then the following options are available:

SPOT

FILLET

BUTT

CROSS_FILLET

COMBINED

Purpose: Define spot and fillet 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.

Card 1 Format. This card is required for all weld options.

	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
NSID	Nodal set ID, see *SET_NODE_*OPTION*.
CID	Coordinate system ID for output of data in local system, see *DEFINE_ COORDINATE_*OPTION*. CID is not required for spotwelds if the nodes are not conincident.
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.

VARIABLE	DESCRIPTION
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. and general welds.
NPRT	Print option in file RBDOUT. EQ.0: default from Control Card is used 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 spotwelds.
SS	S_s, shear force at failure, only for the brittle failure of spotwelds.
N	n, exponent for normal force, only for the brittle failure of spotwelds.
M	m, exponent for shear force, only for the brittle failure of spotwelds.

Remarks:

Spotweld 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 in slightly increased run times.

Brittle failure of the spotwelds occurs when:

$$\left(\frac{\max(f_n,0)}{S_n}\right)^n+\left(\frac{|f_s|}{S_s}\right)^m\geq 1$$

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 5.1 the ordering of the nodes is shown for the 2 node and 3 node spotwelds. This order is with respect to the local coordinate system where the local z axis determines the tensile direction. The nodes in the spotweld may coincide. The failure of the 3 node spotweld may occur gradually with first one node failing and later the second node may fail. For n noded spotwelds 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.

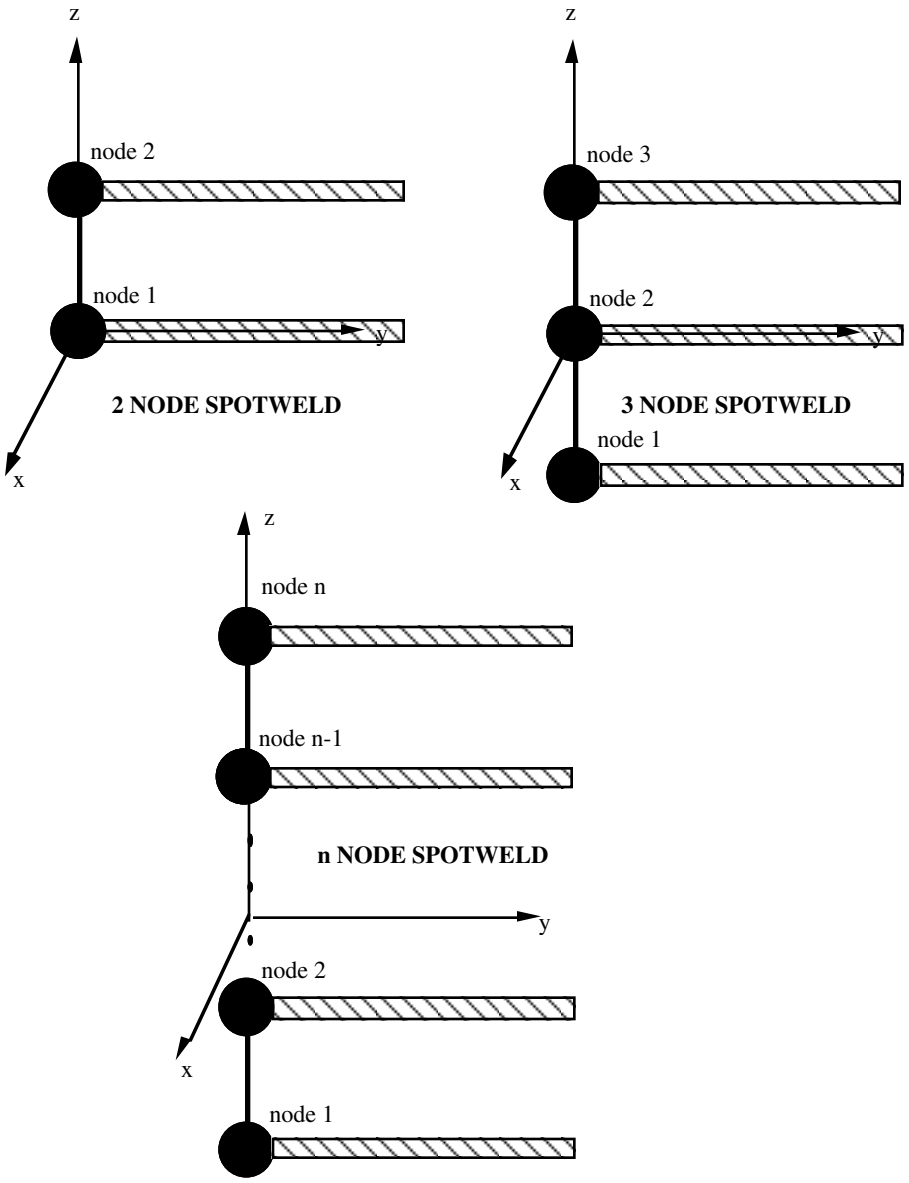


Figure 5.1. Nodal ordering and orientation of the local coordinate system is important for determining spotweld failure.

Additional Card required for the FILLET option:

Card 2	1	2	3	4	5	6	7	8
Variable	TFAIL	EPSF	SIGY	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.
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 5.2 and 5.3).
W	w, width of flange (see Figure 5.2).
A	a, width of fillet weld (see Figure 5.2).
ALPHA	α, weld angle (see Figure 5.2) in degrees.

Remarks:

Ductile fillet weld failure, due to plastic straining, is treated identically to spotweld 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 5.2 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.

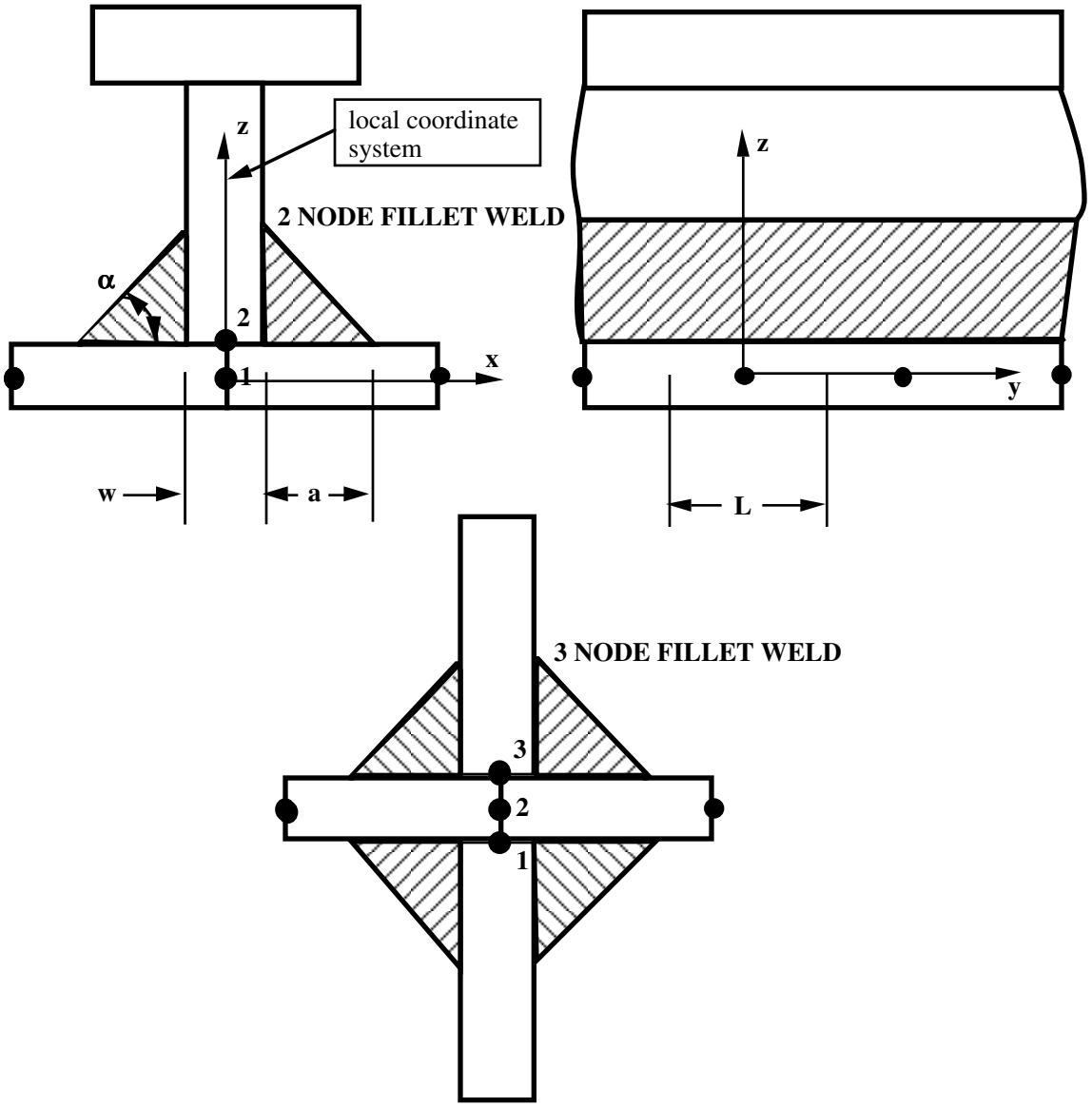


Figure 5.2. Nodal ordering and orientation of the local coordinate system is shown for fillet weld failure. The angle is defined in degrees.

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 5.2 and 5.3).
D	d, thickness of butt weld (see Figure 5.3).
LT	L_t, transverse length of butt weld (see Figure 5.3).

Remarks:

Ductile butt weld failure, due to plastic straining, is treated identically to spotweld 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.

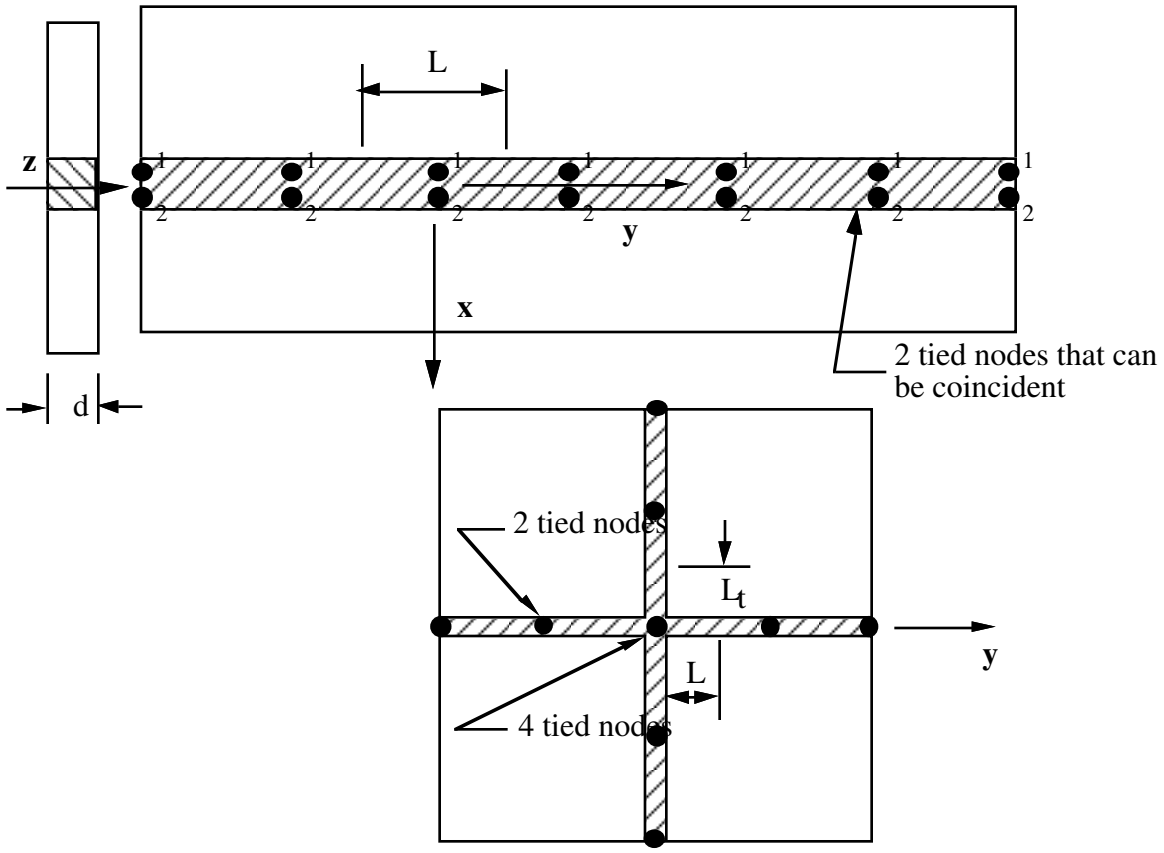


Figure 5.3. Orientation of the local coordinate system and nodal ordering is shown for butt weld failure.

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$  *CONSTRAINED_GENERALIZED_WELD_BUTT
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Weld two plates that butt up against each other at three nodal pair
$  locations.  The nodal pairs are 32-33, 34-35 and 36-37.
$
$  This requires 3 separate *CONSTRAINED_GENERALIZED_WELD_BUTT definitions,
$  one for each nodal pair.  Each weld is to have a length (L) = 10,
$  thickness (D) = 2, and a transverse length (Lt) = 1.
$
$  Failure is defined two ways:
$    Ductile failure if effective plastic strain exceeds 0.3
$    Brittle failure if the stress failure criteria exceeds 0.25
$       - scale the brittle failure criteria by beta = 0.9.
$    Note: beta < 1 weakens weld          beta > 1 strengthens weld
$
*CONSTRAINED_GENERALIZED_WELD_BUTT
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$     nsid       cid
        21
$    tfail      epsf      sigy      beta         L         D        Lt
                0.3     0.250       0.9      10.0       2.0       1.0
$
$
*CONSTRAINED_GENERALIZED_WELD_BUTT
$     nsid       cid
        23
$    tfail      epsf      sigy      beta         L         D        Lt
                0.3     0.250       0.9      10.0       2.0       1.0
$
$
*CONSTRAINED_GENERALIZED_WELD_BUTT
$     nsid       cid
        25
$    tfail      epsf      sigy      beta         L         D        Lt
                0.3     0.250       0.9      10.0       2.0       1.0
$
$
*SET_NODE_LIST
$      sid
        21
$     nid1      nid2
        32        33
*SET_NODE_LIST
        23
        34        35
*SET_NODE_LIST
        25
        36        37
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

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 5.2 and 5.3).
W	w, width of flange (see Figure 5.2).
A	a, width of fillet weld (see Figure 5.2).
ALPHA	α, weld angle (see Figure 5.2) in degrees.
NODEA	Node ID, A, in weld pair (CROSS or COMBINED option only). See Figure 5.4.
NODEB	Node ID, B, in weld pair (CROSS orCOMBINED option only).
NCID	Local coordinate system ID (CROSS or COMBINED option only).

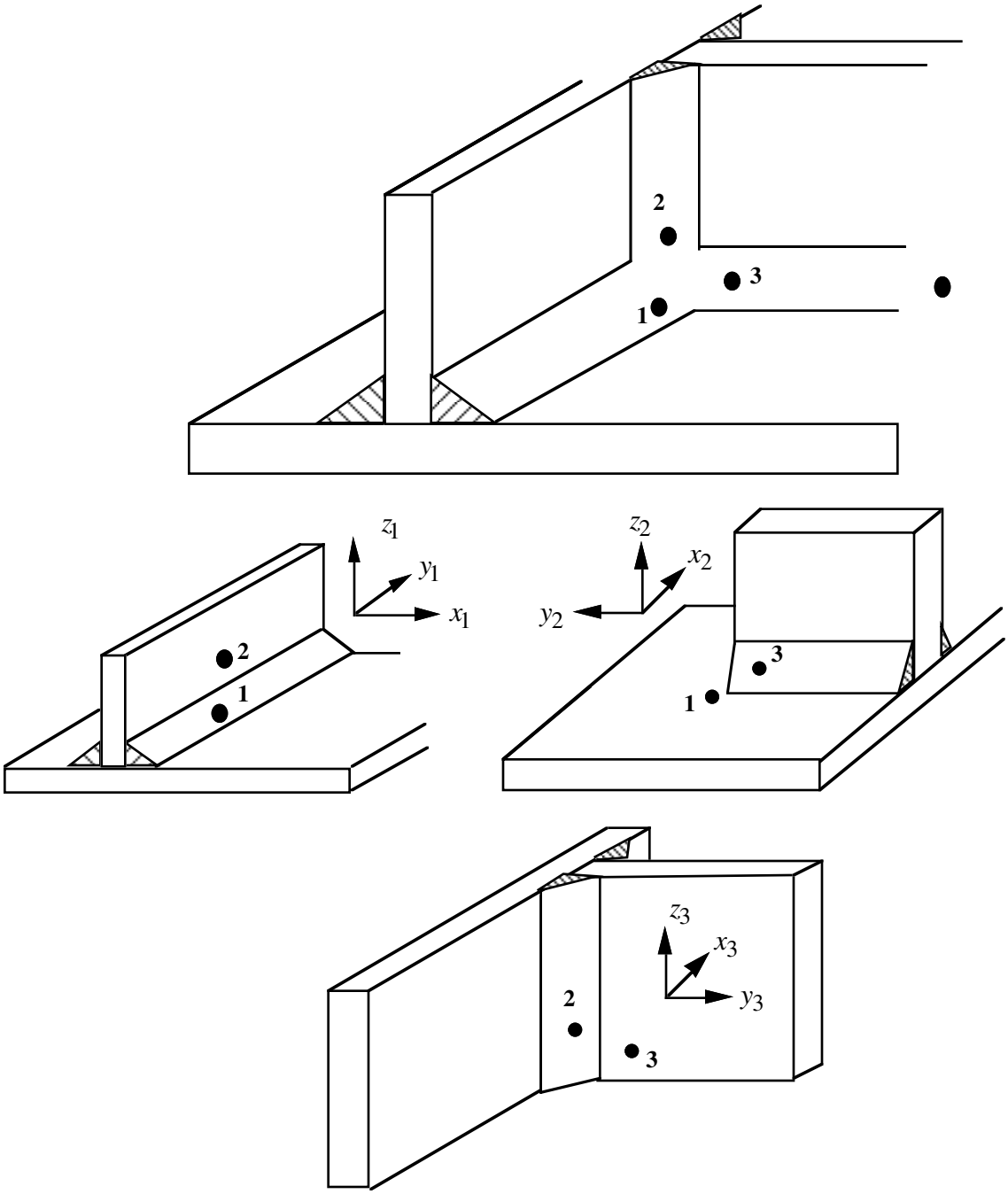


Figure 5.4. A simple cross fillet weld illustrates the required input. Here NFW=3 with nodal pairs (A=2, B=1), (A=3, B=1), and (A=3, B=2). The local coordinate axes are shown. These axes are fixed in the rigid body and are referenced to the local rigid body coordinate system which tracks the rigid body rotation.

*CONSTRAINED

Additional NPR Card Sets required for the COMBINED option. Repeat cards 2 and 3 below NPR times:

Card 2

Variable	TFAIL	EPSF	SIGY	BETA	L	W	A	ALPHA
Type	F	F	F	F	F	F	F	F

Card 3

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 5.2 and 5.3).
W	w, width of flange (see Figure 5.2).
A	a, width of fillet weld (see Figure 5.2).
ALPHA	α, weld angle (see Figure 5.2) 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 5.5. EQ.0: fillet weld EQ.1: butt weld

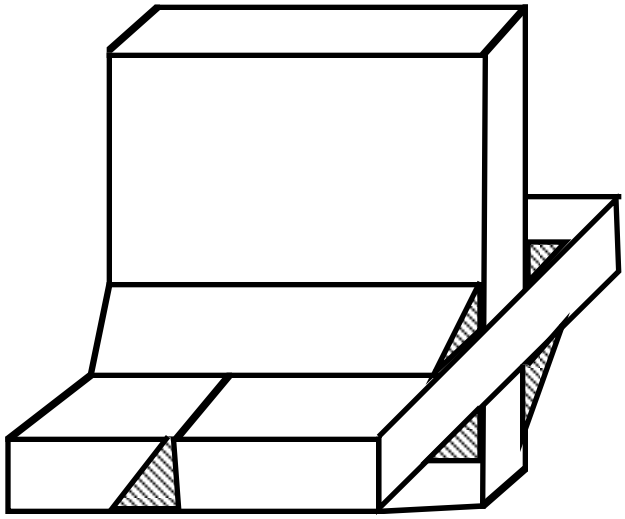


Figure 5.5. A combined weld is a mixture of fillet and butt welds.

*CONSTRAINED_GLOBAL

Purpose: Define a global boundary constraint plane.

Card Format

	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 x.
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

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. One *CONSTRAINED_ INTERPOLATION card is required for each constraint definition. The input list of independent nodes is terminated when the next "*" card is found.

Card Format

	1	2	3	4	5	6	7	8
Variable	ICID	DNID	DDOF					
Type	I	I	I					
Default	0	0	123456					

Cards 2, 3, 4, etc. Define one card per independent node. Input is terminated when a "*" card is found.

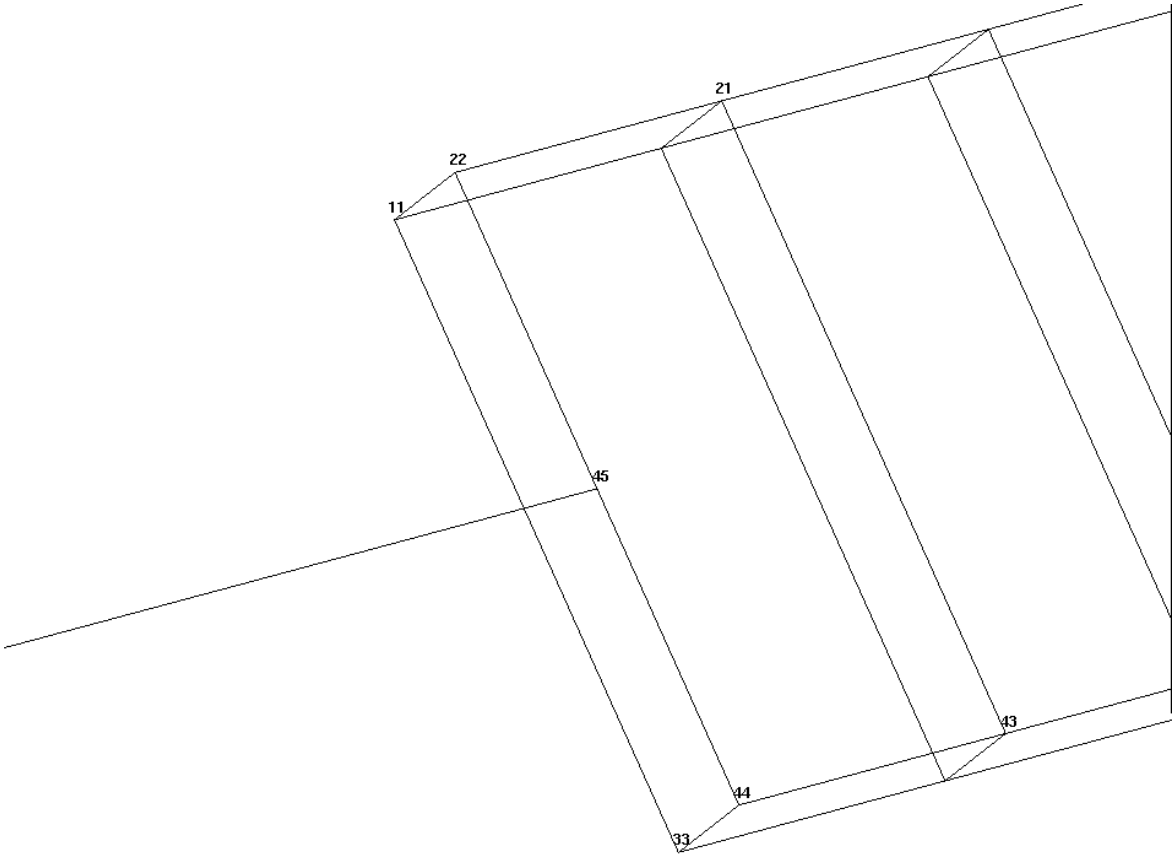
	1	2	3	4	5	6	7	8
Variable	INID	IDOF	TWGHTX	TWGHTY	TWGHTZ	RWGHTX	RWGHTY	RWGHTZ
Type	I	I	F	F	F	F	F	F
Default	0	123456	1.0	TWGHTX	TWGHTX	TWGHTX	TWGHTX	TWGHTX

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:

VARIABLE	DESCRIPTION
	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
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.
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.

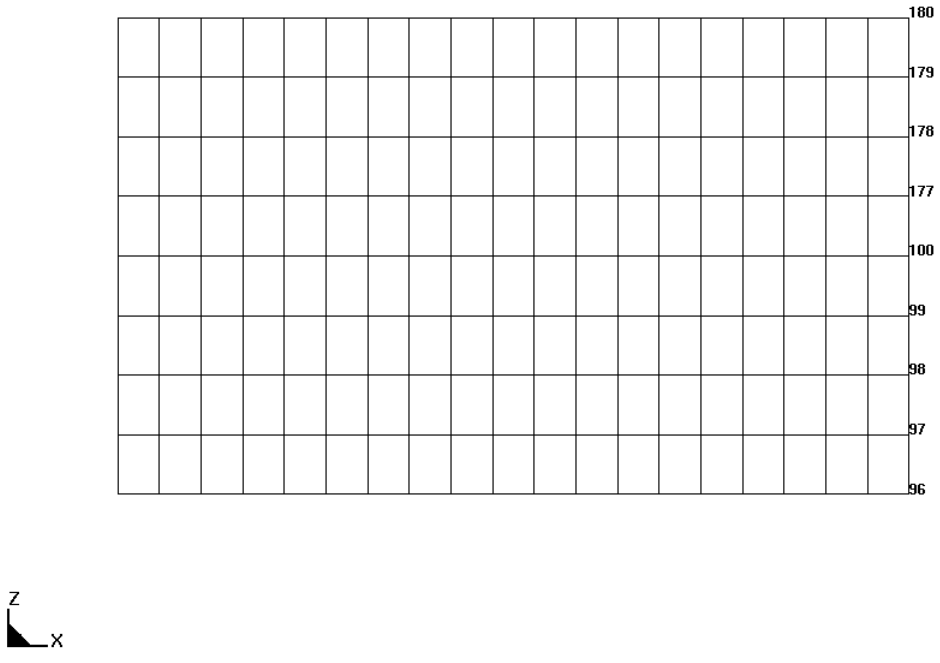
```
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$  *CONSTRAINED_INTERPOLATION  (Beam to solid coupling)
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Tie a beam element to a solid element.
$
$  The node of the beam to be tied 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
$
*.........
$
```



*CONSTRAINED

***CONSTRAINED_JOINT**_*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 then an additional option is available:

LOCAL

Purpose: Define a joint between two rigid bodies, see Figure 5.6.

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 4.4) 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.

The geometry of joints is defined in Figure 5.4. 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).

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.
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

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 Card Format: 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

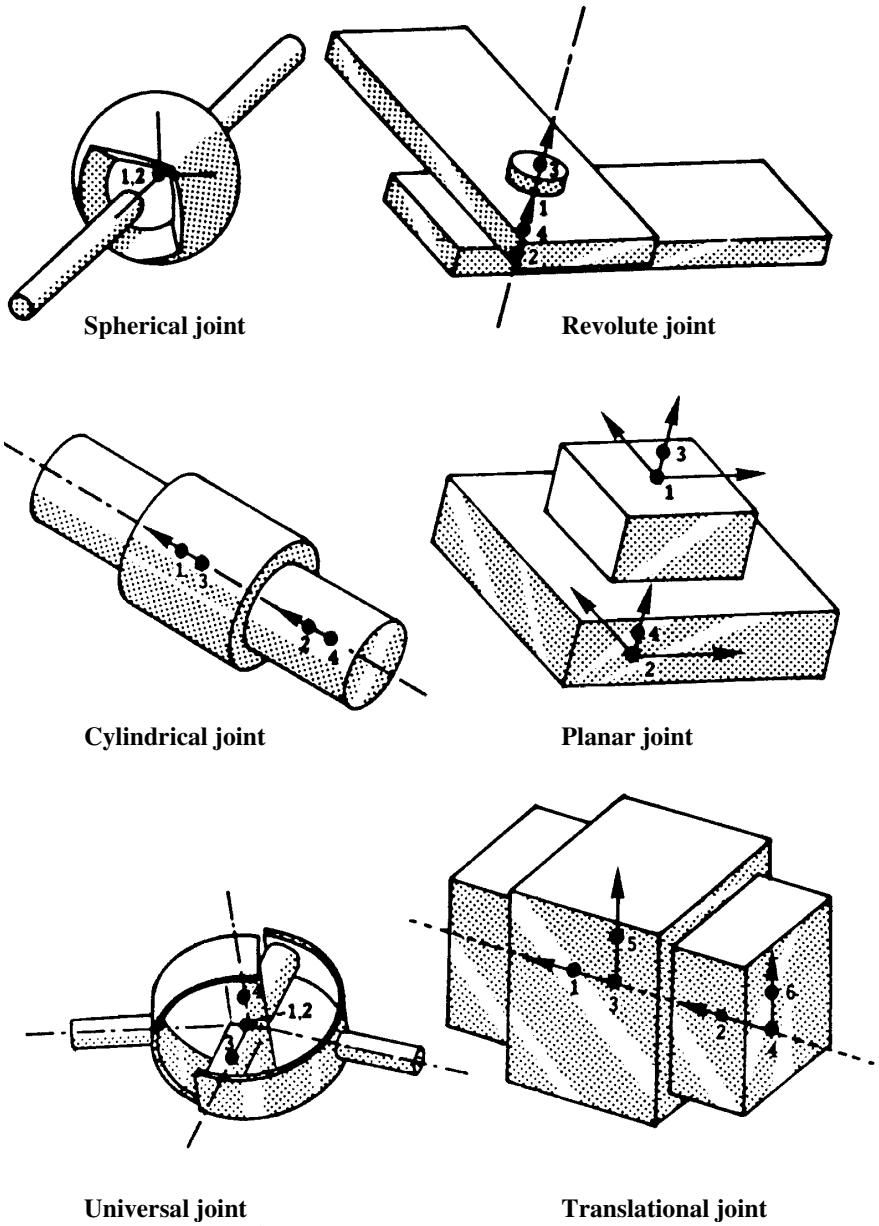


Figure 5.6. Joint definitions 1-6

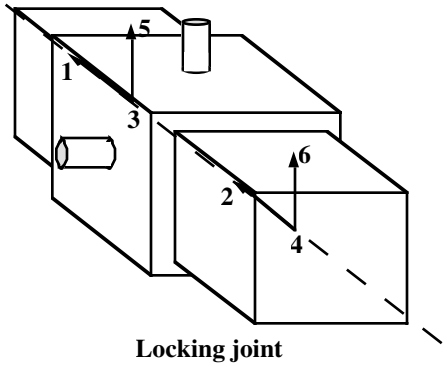


Figure 5.7. Locking joint.

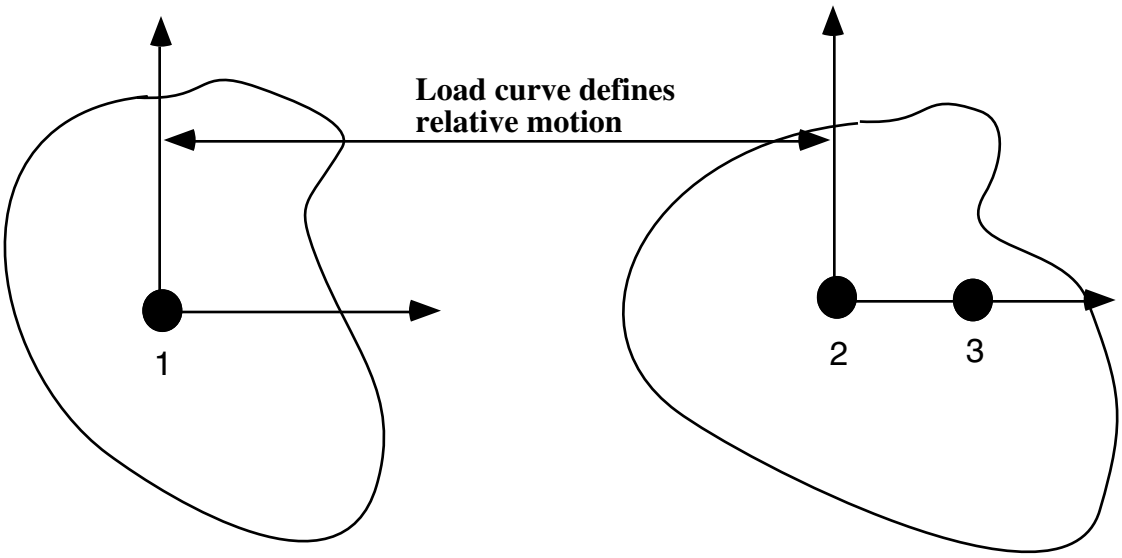


Figure 5.8. Translational motor joint. This joint can be used in combination with the translational or the cylindrical joint.

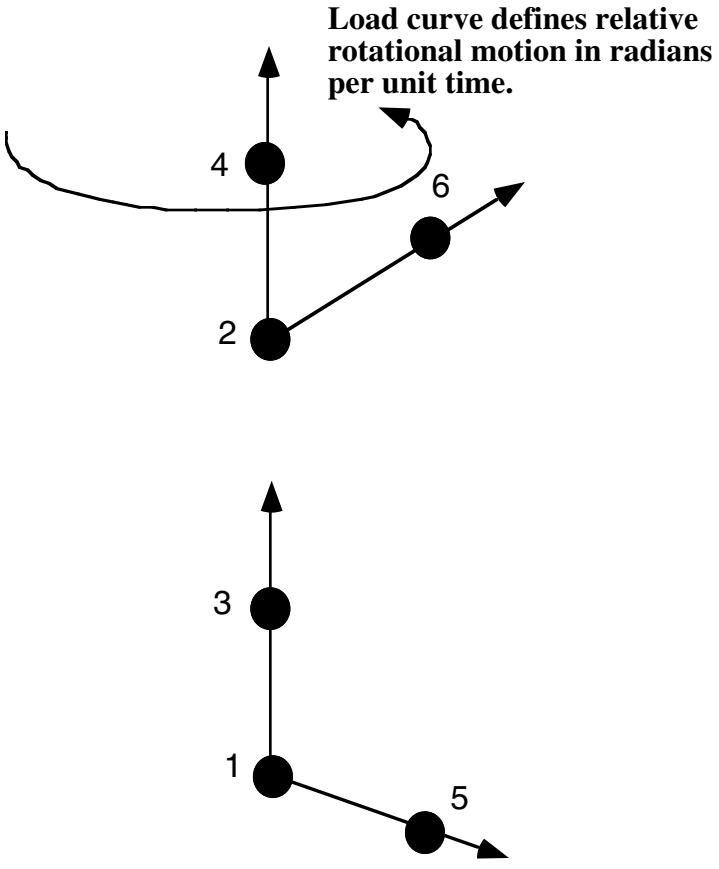


Figure 5.9. Rotational motor joint. This joint can be used in combination with other joints such as the revolute or cylindrical joints.

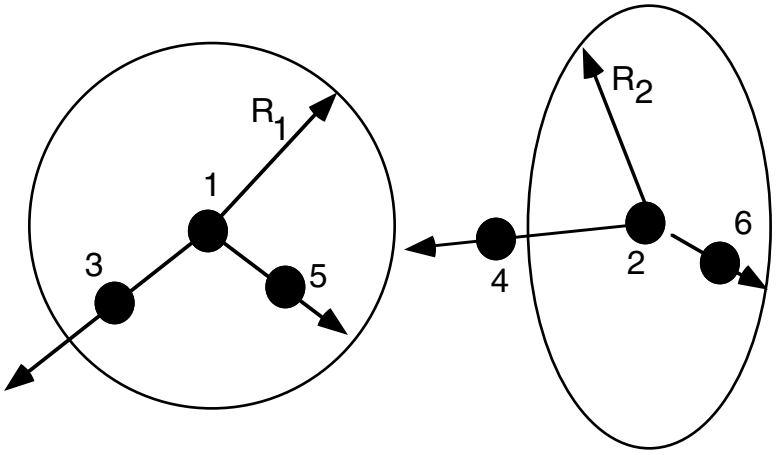


Figure 5.10. Gear joint. Nodal pairs (1,3) and (2,4) define axes that are orthogonal to the gears. Nodal pairs (1,5) and (2,6) define vectors in the plane of the gears. The ratio $\frac{R_2}{R_1}$ is specified.

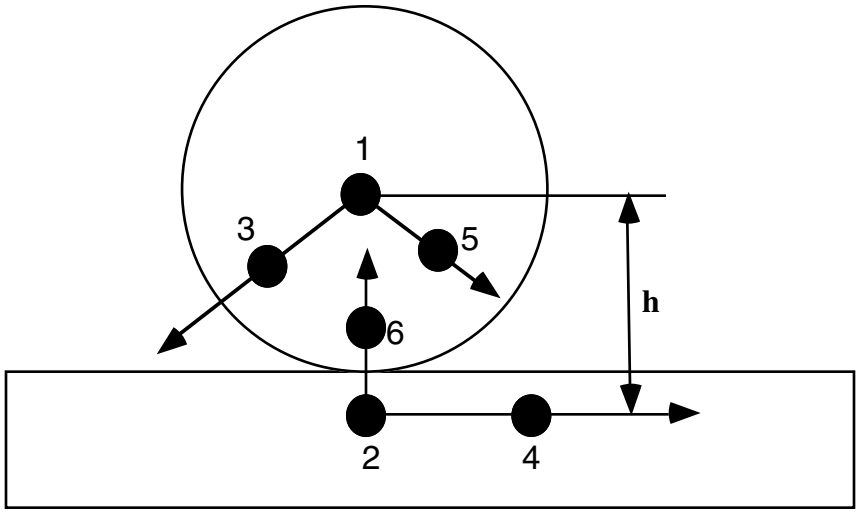


Figure 5.11. Rack and pinion joint. Nodal pair (1,3) defines a vector that is orthogonal to the plane of the gear. Nodal pair (1,5) is a vector in the plane of the gear. Nodal pair (2,4) defines the direction of travel for the second body. The value h is specified.

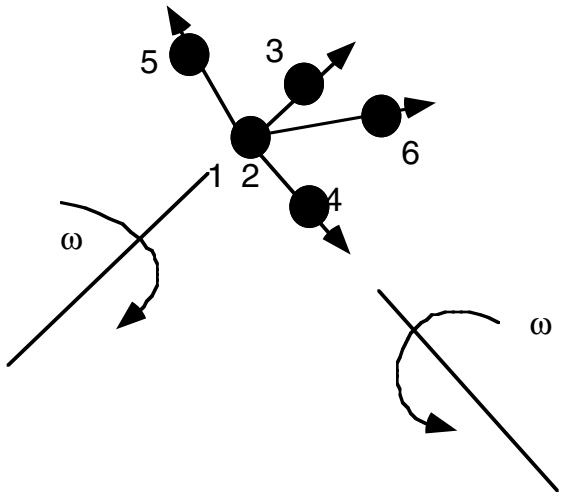


Figure 5.12. Constant velocity joint. Nodal pairs (1,3) and (2,4) define an axes for the constant angular velocity, and nodal pairs (1,5) are orthogonal vectors. Here nodal points 1 and 2 must be coincident.

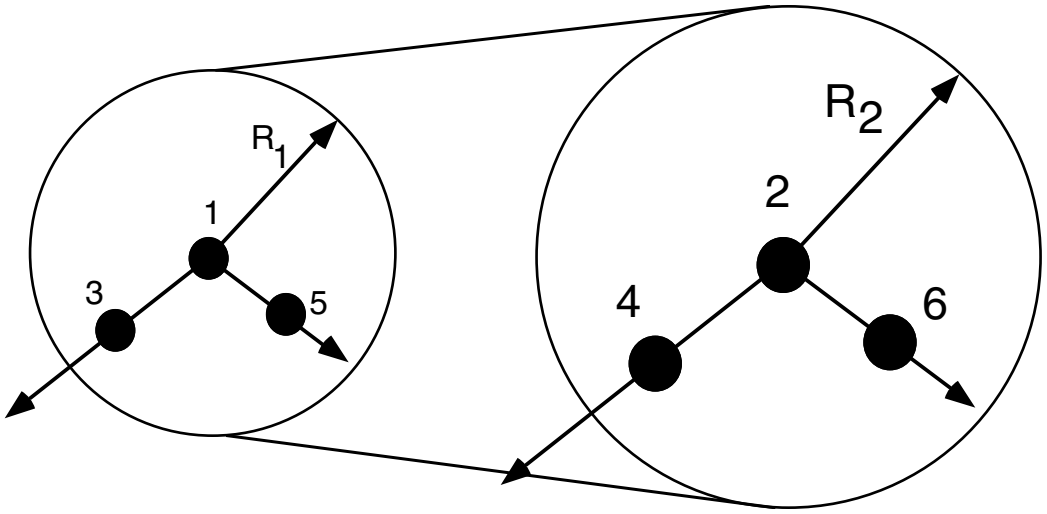


Figure 5.13. Pulley joint. Nodal pairs (1,3) and (2,4) define axes that are orthogonal to the pulleys. Nodal pairs (1,5) and (2,6) define vectors in the plane of the pulleys. The ratio $\frac{R_2}{R_1}$ is specified.

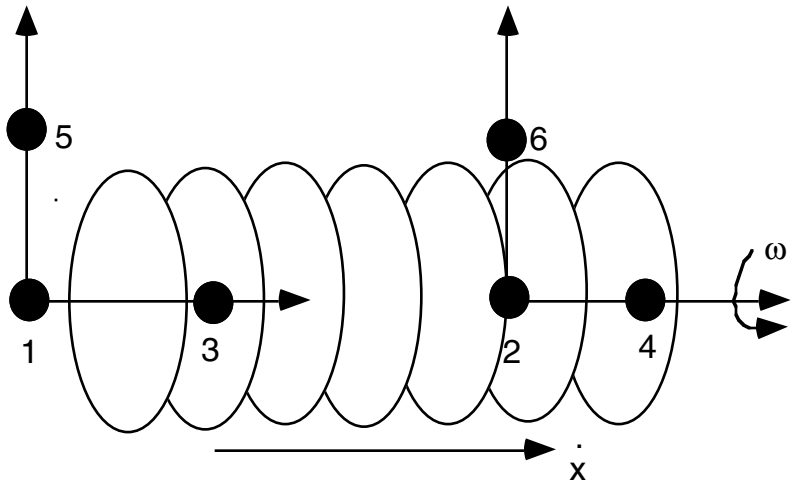


Figure 5.14. Screw joint. The second body translates in response to the spin of the first body. Nodal pairs (1,3) and (2,4) lie along the same axis and nodal pairs (1,5) and (2,6) are orthogonal vectors. The helix ratio, $\frac{\dot{x}}{\omega}$, is specified.

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$  *CONSTRAINED_JOINT_PLANAR
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  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_STIFFNESS_*OPTION

Options include:

GENERALIZED

FLEXION-TORSION

Purpose: Define optional rotational joint stiffnesses 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 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 Format:

Card 1 is common to all joint stiffness types.

Cards 2 to 4 are unique for each stiffness type.

Card 1 - Required for all joint stiffness types.

Card 1	1	2	3	4	5			
Variable	JSID	PIDA	PIDB	CIDA	CIDB			
Type	I	I	I	I	I			
Default	none	none	none	none	CIDA			

VARIABLE	DESCRIPTION
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*.
CIDB	Coordinate ID for rigid body B. If zero, the coordinate ID for rigid body A is used, see *DEFINE_COORDINATE_*OPTION*.

Card 2 of 4 - Required for GENERALIZED stiffness.

Card 2	1	2	3	4	5	6		
Variable	LCIDPH	LCIDT	LCIDPS	DLCIDPH	DLCIDT	DLCIDPS		
Type	I	I	I	I	I	I		
Default	none	none	none	none	none	none		

VARIABLE	DESCRIPTION
LCIDPH	Load curve ID for ϕ–moment versus rotation in radians. See Figure 5.15. 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.
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.

Card 3 of 4 - Required for GENERALIZED stiffness.

Card 3	1	2	3	4	5	6		
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		

VARIABLE	DESCRIPTION
ESPH	Elastic stiffness per unit radian for friction and stop angles for ϕ rotation. See Figure 5.16. 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 ID defining the yield moment versus ϕ rotation. See Figure 5.8.
EST	Elastic stiffness per unit radian for friction and stop angles for θ rotation. See Figure 5.16. 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 ID defining the yield moment versus θ rotation. See Figure 5.16.
ESPS	Elastic stiffness per unit radian for friction and stop angles for ψ rotation. See Figure 5.16. 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 ID defining the yield moment versus ψ rotation. See Figure 5.16.

Card 4 of 4 - Required for GENERALIZED stiffness.

Card 4	1	2	3	4	5	6		
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
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.

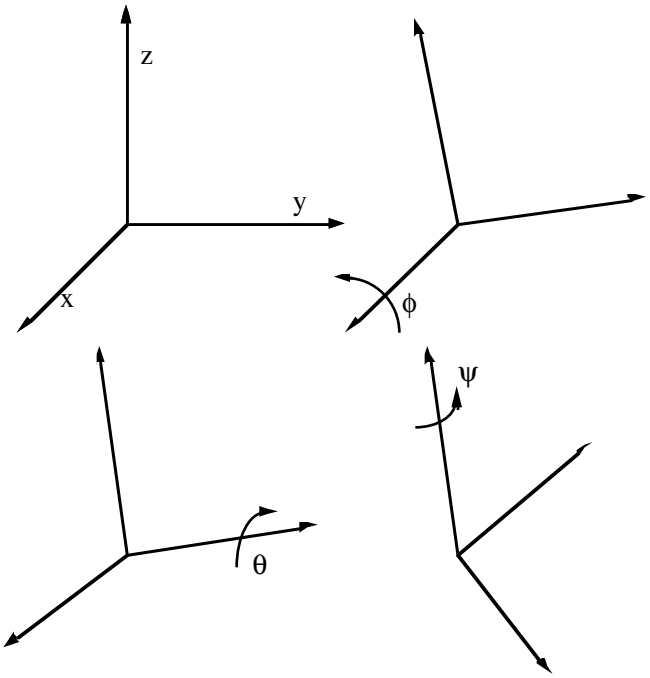


Figure 5.15. Definition of angles for the generalized joint stiffness. The magnitude of the angular rotations are limited by the stop angles defined on Card 4. If the initial local coordinate axes do not coincide, the angles, ϕ, θ, and ψ, will be initialized and torques will develop instantaneously based on the defined load curves.

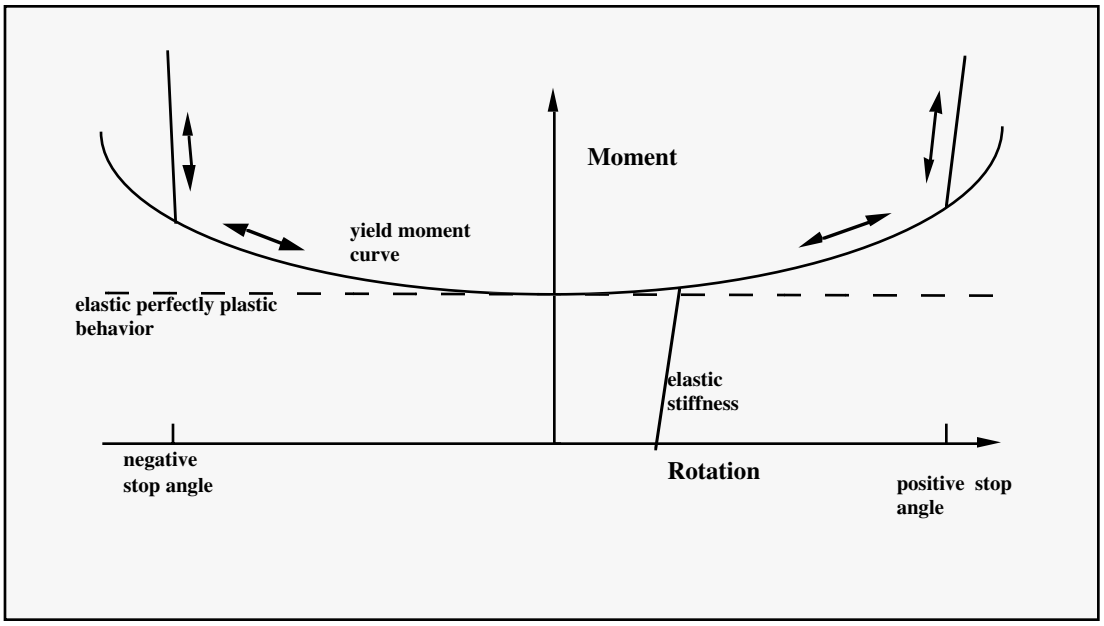


Figure 5.16. Frictional behavior is modeled by a plasticity model. Elastic behavior is obtained once the stop angles are reached. The same elastic stiffness is used to simulate sticking situations.

Card 2 of 4 - Required for FLEXION-TORSION stiffness.

Card 2	1	2	3	4	5	6		
Variable	LCIDAL	LCIDG	LCIDBT	DLCIDAL	DLCIDG	DLCIDBT		
Type	I	I	I	I	I	I		
Default	none	1.0	none	none	1.0	none		

VARIABLE	DESCRIPTION
LCIDAL	Load curve ID for α–moment versus rotation in radians. See Figure 5.9 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.
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.

Card 3 of 4 - Required for FLEXION-TORSION stiffness.

Card 3	1	2	3	4				
Variable	ESAL	FMAL	ESBT	FMBT				
Type	F	F	F	F				
Default	0.0	0.0	0.0	0.0				

VARIABLE	DESCRIPTION
ESAL	Elastic stiffness per unit radian for friction and stop angles for α rotation, see Figure 5.17. 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 ID defining the yield moment versus α rotation, see Figure 5.17.
ESBT	Elastic stiffness per unit radian for friction and stop angles for β twist, see Figure 5.17. 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 ID defining the yield moment versus β rotation, see Figure 5.17.

Card 4 of 4 - Required for FLEXION-TORSION stiffness.

Card 4	1	2	3					
Variable	SAAL	NSABT	PSABT					
Type	F	F	F					
Default	not used	not used	not used					

VARIABLE	DESCRIPTION
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.

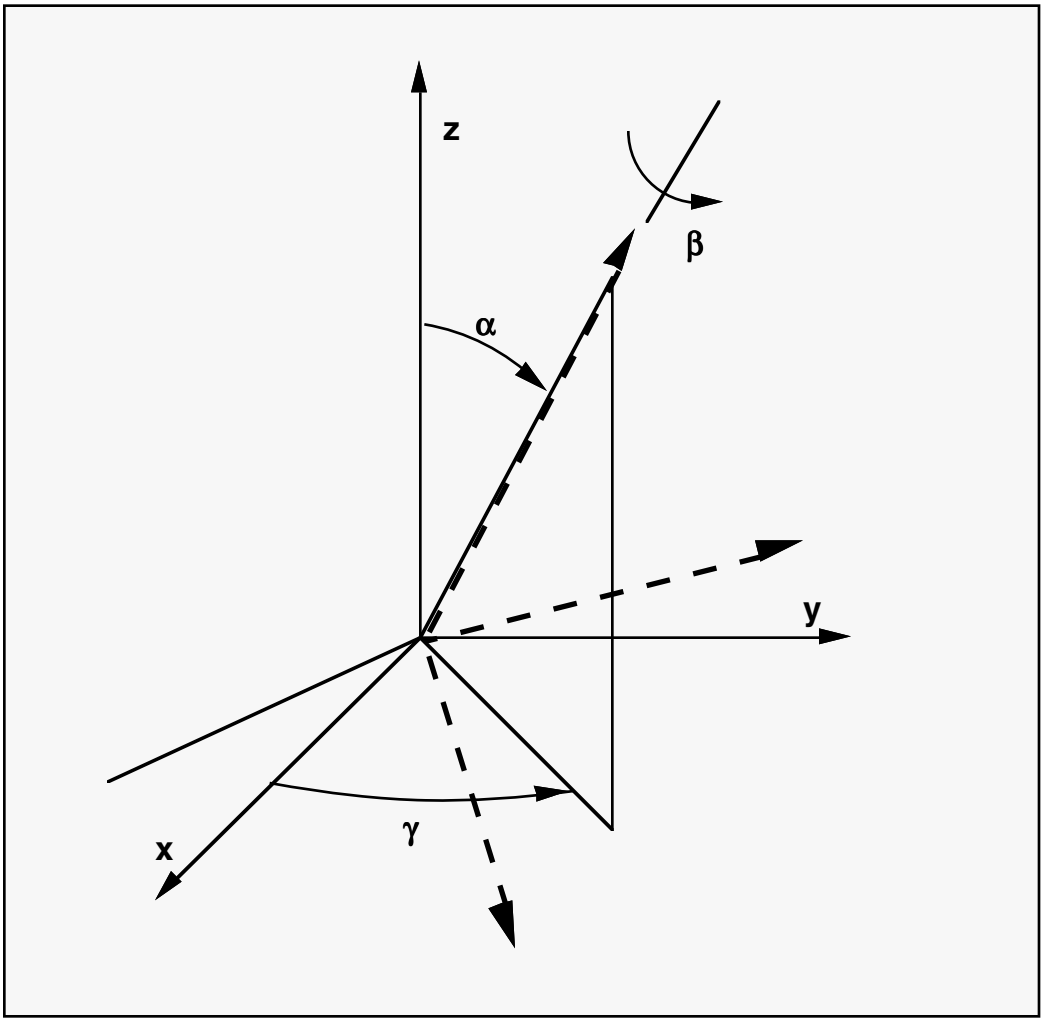


Figure 5.17. Flexion-torsion joint angles. If the initial positions of the local coordinate axes of the two rigid bodies connected by the joint do not coincide, the angles, α and γ, are initialized and torques will develop instantaneously based on the defined load curves. The angle β is also initialized but no torque will develop about the local axis on which β is measured. Rather, β will be measured relative to the computed offset.

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$  *CONSTRAINED_JOINT_STIFFNESS_GENERALIZED
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  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 futher 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_LAGRANGE_IN_SOLID

Purpose: Couple a Lagrangian mesh (slave) of shells, solids or beams to the material points of an Eulerian mesh (master). This option may also be used to model rebar in concrete or tire cords in rubber. The slave part or slave part set is coupled to the master part or master part set.

Note: For RIGID slave PARTS a penalty coupling method must be used, see option CTYPE below.

Card Format

Card 1

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

Variable	START	END	PFAC	FRIC	FRCMIN	NORM		
Type	F	F	F	F	F	I		
Default	0	1.0E10	0.1	0.0	0.5	0		

Card 3

Variable	CQ	HMIN	HMAX					
Type	F	F	F					
Default	0.0	none	none					

VARIABLE	DESCRIPTION
SLAVE	Part, part set ID or Segment set ID of slaves see *PART , *SET_PART or *SET_SEGMENT.
MASTER	Part or part set ID of master solid elements, see *PART or *SET_PART.
SSTYP	Slave type: EQ.0: part set ID, EQ.1: part ID. EQ.2: segment set ID
MSTYP	Master type: EQ.0: part set ID, EQ.1: part ID.
NQUAD	Quadratue rule for coupling slaves to solids. EQ.0: at nodes only, EQ.n: use a rectangular grid of n*n points, EQ.-n: at nodes and a rectangular grid of n*n points.
CTYPE	Coupling type EQ.1: constrained acceleration, EQ.2: constrained acceleration and velocity (default), EQ.3: constrained acceleration and velocity, normal direction only, EQ.4: penalty coupling (Shell Elements), EQ.5: penalty coupling with erosion (Solid Elements).
DIREC	Coupling direction (CTYPE 4 and 5). EQ.1: normal direction, compression and tension (default), EQ.2: normal direction, compression only, EQ.3: all directions.
MCOUP	Multi-material option (CTYPE 4 and 5). EQ.0: couple with all multi-material groups, EQ.1: couple with material with highest density.
START	Start time for coupling.
END	End time for coupling.
PFAC	Penalty factor (CTYPE 4 and 5 only).
FRIC	Coefficient of friction (DIREC 2 only).
FRCMIN	Minimum volume fraction to activate coupling (MCOUP=1)

VARIABLE	DESCRIPTION
NORM	Shell and segment normal orientation: EQ.0: right hand rule (default), EQ.1: left hand rule.
CQ	Heat transfer coefficeint, C_q.
HMIN	Minimum air gap in heat transfer, $h_{\min}$.
PFAC	Maximum air gap in heat transfer, $h_{\max}$. There is no heat transfer above this value.

Remark:

The heat flux per unit area, q,is defined as:

$$q = \frac{C_q \Delta T}{\max(h_{\min}, h)}$$

where ΔT is the temperature difference between the master and slave sides and where h is the actual air gap

*CONSTRAINED_LINEAR

Purpose: Define linear constraint equations between displacements/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	NUM							
Type	I							
Default	none							

Card 2 - Define NUM cards below, one card for each nodal point.

Card 2	1	2	3	4	5	6	7	8
Variable	NID	DOFX	DOFY	DOFZ	DOFRX	DOFRY	DOFRZ	COEF
Type	I	I	I	I	I	I	I	F
Default	none	0	0	0	0	0	0	0
Remark	1							

VARIABLE	DESCRIPTION
NUM	Number of nodes in equation
NID	Node ID

Define only one nonzero number for parameters DOFX, DOFY, DOFZ, DOFRX, DOFRY, and DOFRZ.

DOFX	Insert 1 (0) for (no) translational constraint in x-direction.
DOFY	Insert 1 (0) for (no) translational constraint in y-direction.
DOFZ	Insert 1 (0) for (no) translational constraint in z-direction.
DOFRX	Insert 1 (0) for (no) rotational constraint about x-axis.

VARIABLE	DESCRIPTION
DOFRY	Insert 1 (0) for (no) rotational constraint about y-axis.
DOFRZ	Insert 1 (0) for (no) rotational constraint about 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.

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$  *CONSTRAINED_LINEAR
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Constrain nodes 40 and 42 to move identically in the z-direction.
$
$  When the linear constraint equation is applied, it goes like this:
$
$     0 = C40uz40 + C42uz42
$
$       = uz40 - uz42
$
$     uz40 = uz42
$
$   where,
$     C40  =  1.00  coefficient for node 40
$     C42  = -1.00  coefficient for node 42
$     uz40 = displacement of node 40 in z-direction
$     uz42 = displacement of node 42 in z-direction
$
$
*CONSTRAINED_LINEAR
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$        i
$      num
         2
$
$      nid      dofx      dofy      dofz     dofrx     dofry     dofrz      coef
        40                             1                                    1.00
        42                             1                                   -1.00
$
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

*CONSTRAINED

*CONSTRAINED_NODAL_RIGID_BODY_{*OPTION*}

If the inertial properties are defined rather than computed, then the following option is available:

INERTIA

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.

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 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.

Card 1 - Required.

Card 1	1	2	3	4	5	6	7	8
Variable	PID	CID	NSID	PNODE	IPRT			
Type	I	I	I	I	I			
Default	none	none	none	0	0			

VARIABLE	DESCRIPTION
PID	Part ID of the nodal rigid body.
CID	Coordinate system ID for output of data in local system, see *DEFINE_COORDINATE_OPTION. Only necessary if no local system is defined below.

VARIABLE	DESCRIPTION
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, velocites, and displacements of this node will coorespond 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.
IPRT	Print flag. For nodal rigid bodies with more than two nodes the following values apply: EQ.0: write 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 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.

Card 2 of 4 - Required for the INERTIA option.

Card 2	1	2	3	4	5	6	7	8
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

VARIABLE	DESCRIPTION
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.

Card 3 of 4 - 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

Card 4 of 4 - 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.

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	CID	
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
CID	Local coordinate system ID, see *DEFINE_COORDINATE_.... With this option leave fields 1-6 blank.

Remark:

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.

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$  *CONSTRAINED_NODAL_RIGID_BODY
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Define a rigid body consisting of the nodes in nodal set 61.
$
$  This particular example was used to connect three separate deformable
$  parts.  Physically, these parts were welded together.  Modeling wise,
$  however, this joint is quit messy and is most conveniently modeled
$  by making a rigid body using several of the nodes in the area.  Physically,
$  this joint was so strong that weld failure was never of concern.
$
*CONSTRAINED_NODAL_RIGID_BODY
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$     nsid       cid
        61
$
$     nsid = 61   nodal set ID number, requires a *SET_NODE_option
$      cid        not used in this example, output will be in global coordinates
$
$
*SET_NODE_LIST
$      sid
        61
$     nid1      nid2      nid3      nid4      nid5      nid6      nid7      nid8
       823      1057      1174      1931      2124      1961      2101
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

*CONSTRAINED_NODE_SET

Purpose: Define nodal constraint sets for translational motion in global coordinates. No rotational coupling. See Figure 5.18. 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.

Card Format

	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
NSID	Nodal set ID, see *SET_NODE_*OPTION*.
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 definiton 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.

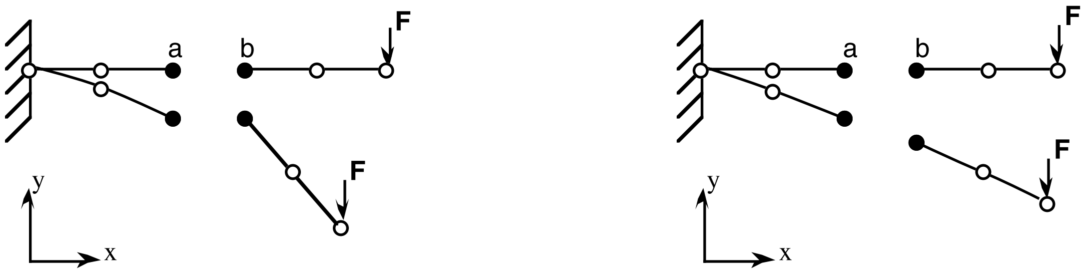
2. When the failure time, *TF*, is reached the nodal constraint becomes inactive and the constrained nodes may move freely.

<u>*CONSTRAINED_NODE_SET</u>

Since no rotation is permitted, this option should <u>not</u> be used to model rigid body behavior that involves rotations.

<u>*CONSTRAINED_NODAL_RIGID_BODY</u>
<u>*CONSTRAINED_SPOTWELD</u>

Behavior is like a rigid beam. These options may be used to model spotwelds.



<u>Offset nodes a and b are constrained to move together.</u>

Figure 5.18. *CONSTRAINED_NODE_SET can lead to nonphysical responses.

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$   *CONSTRAINED_NODE_SET
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Constrain all the nodes in a nodal set to move equivalently
$  in the z-direction.
$
*CONSTRAINED_NODE_SET
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
      nsid       dof        tf
         7         3      10.0
$
$     nsid = 7    nodal set ID number, requires a *SET_NODE_option
$      dof = 3    nodal motions are equivalent in z-translation
$      tf  = 3    at time=10. the nodal contraint is removed
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

*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
Variable	CID							
Type	I							
Default	none							

Card Format (I8,3E16.0)

Card 2	1	2	3	4	5	6	7	8	9	10
Variable	EID1	X1		Y1		Z1				
Type	I	F		F		F				
Default	none	0.		0.		0.				

Card 3	1	2	3	4	5	6	7	8	9	10
Variable	EID2	X2		Y2		Z2				
Type	I	F		F		F				
Default	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 Format

	1	2	3	4	5	6	7	8
Variable	PIDM	PIDS						
Type	I	I						
Default	none	none						

VARIABLE	DESCRIPTION
PIDM	Master rigid body part ID, see *PART.
PIDS	Slave rigid body part ID, see *PART.

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.

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 exists 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 velocity and displacement boundary conditions for both the master and slaved rigid bodies. See Figure 5.19.

Card Format

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	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. In the part set, see *SET_PART_*OPTION*, definition the "COLUMN" option may be used to defined as a part attribute the closure distance (D_1 and D_2 in Figure 5.19) 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.
PSIDMN	Optional part set ID of rigid bodies that are slaved in the minimum coordinate direction to the master rigid body. In the part set, see *SET_PART_DEFINITION, definition the "COLUMN" option may be used to defined as a part attribute the closure distance (D_1 and D_2 in Figure 5.11) 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.
LCVMNX	Load curve ID which defines the maximum absolute value of the velocity as a function of time that is allowed within the stopper. 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).
VID	Vector for arbitrary orientation of stopper, see *DEFINE_VECTOR.
TB	Time at which stopper is activated.
TD	Time at which stopper is deactivated.

Remark:

The optional definition of part sets in minimum or maximum coordinate direction allows the motion to be controlled in arbitrary direction.

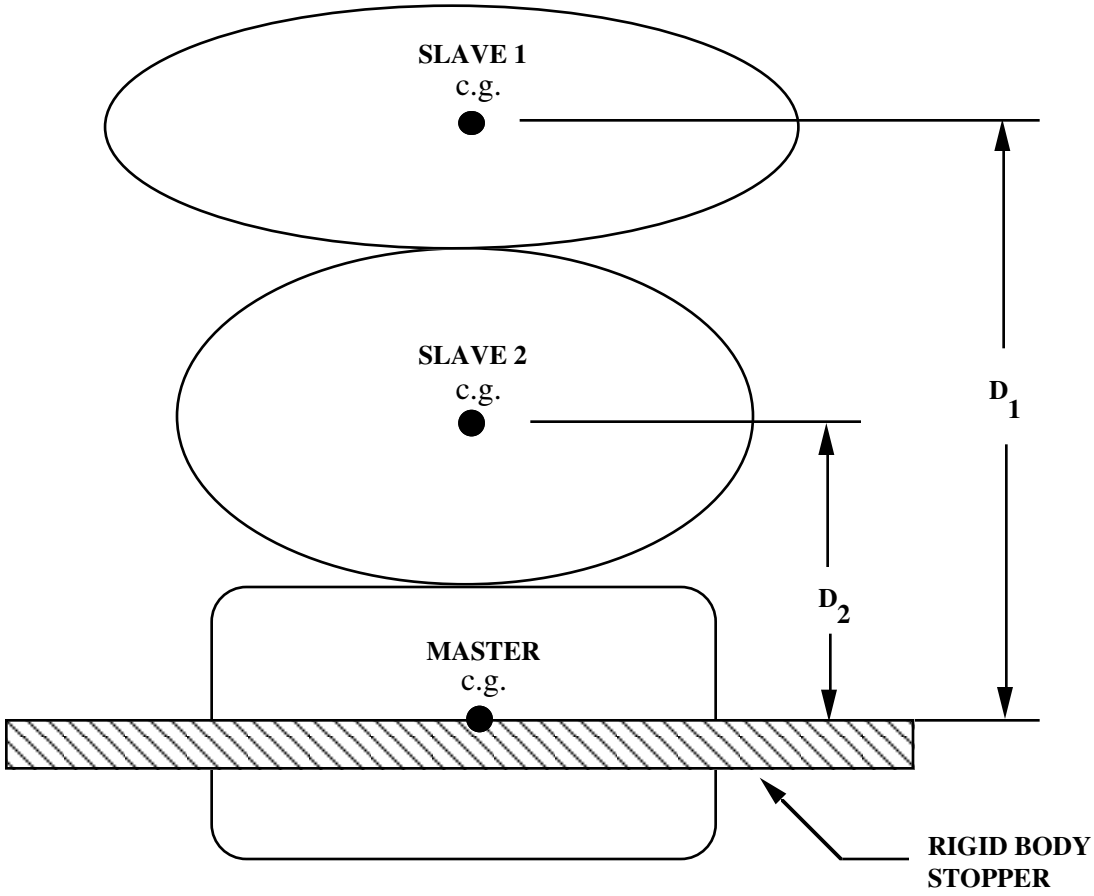


Figure 5.19 When the master rigid body reaches the rigid body stopper, the velocity component into the stopper is set to zero. Slave rigid bodies 1 and 2 also stop if the distance between their mass centers and the master rigid body is less than or equal to the input values D_1 and D_2, respectively. (c.g. ≡ center of gravity).

*CONSTRAINED

*CONSTRAINED_RIVET

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.

Card Format

	1	2	3	4	5	6	7	8
Variable	N1	N2	TF					
Type	I	I	F					
Default	none	none	1.E+20					
Remarks	1		2					

VARIABLE	DESCRIPTION
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 Format

	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 5.20. During the calculation, nodes thus constrained must lie along the fiber but can move relative to each other in the fiber direction. 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.

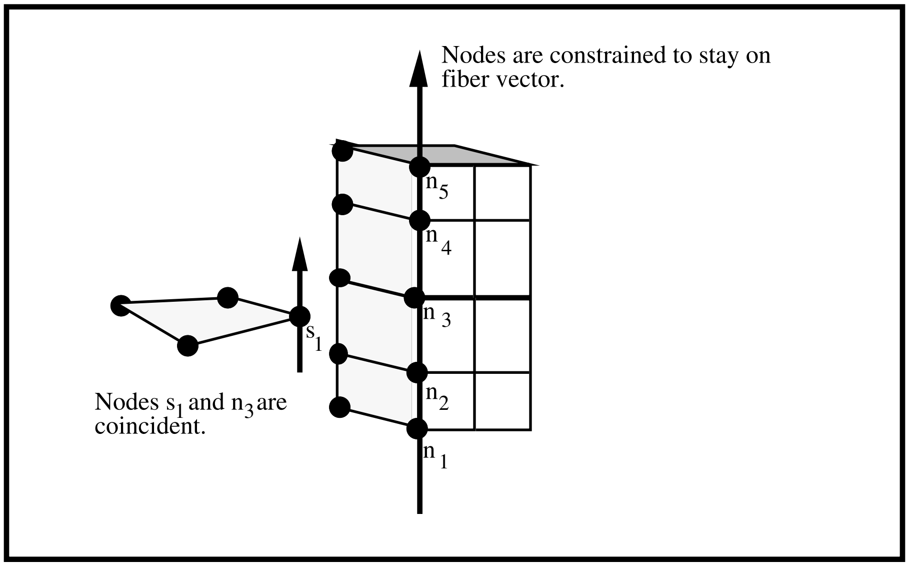


Figure 5.20. The interface between shell elements and solids ties shell node s1 to a line of nodes on the solid elements n1-n5. It is very important for the nodes to be aligned.

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$  *CONSTRAINED_SHELL_TO_SOLID
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Tie shell element, at node 329, to a solid element at node 203.
$    - nodes 329 and 203 are coincident
$
$  Additionally, define a line of nodes on the solids elements, containing
$  node 203, that must remain in the same direction as the fiber of the shell
$  containing node 329.  In other words:
$
$    - Nodes 119, 161, 203, 245 and 287 are nodes on a solid part that
$       define a line on that solid part.
$    - This line of nodes will be constrained to remain linear throughout
$       the simulation.
$    - The direction of this line will be kept the same as the fiber of the
$       of the shell containing node 329.
$
*CONSTRAINED_SHELL_TO_SOLID
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$      nid     nsid
       329        4
$
*SET_NODE_LIST
$      sid
         4
$     nid1      nid2      nid3      nid4      nid5      nid6      nid7      nid8
       119       161       203       245       287
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
```

*CONSTRAINED_SPOTWELD_{*OPTION*}

If it is desired to use a time filtered force calculation for the forced basae failure criterion then the following option is available:

FILTERED_FORCE

and one additional card must be defined below.

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. 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 spot weld can be handeled 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.

Card 1 Format

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	optional	optional	1.E+20	1.E+20
Remarks	1.		2.				3	4

Card 2 Format 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
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).
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^p_{fail}. 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 Format

	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 Format

	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 only to deformable plastic three and four noded shell elements and to brick elements using the honeycomb material *MAT_HONEYCOMB. 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 be between disjoint parts. Different types of contact may be defined:

***CONTACT**_{*OPTION1*}_{*OPTION2*}_{*OPTION3*}_{*OPTION4*}

***CONTACT_ENTITY**

***CONTACT_GEBOD**_*OPTION*

***CONTACT_INTERIOR**

***CONTACT_RIGID_SURFACE**

***CONTACT_1D**

***CONTACT_2D**_*OPTION1*_{*OPTION2*}_{*OPTION3*}

The first, *CONTACT_..., is the general 3D contact algorithms. The second, *CONTACT_ ENTITY, treats contact using mathematical functions to describe the surface geometry for the master surface. The third, *CONTACT_GEBOD is a specialized form of the contact entity for use with the rigid body dummies (see *COMPONENT_GEBOD). The fourth, *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 fifth, *CONTACT_RIGID_SURFACE is for modeling road surfaces for durability and NVH calculations. The sixth, *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

***CONTACT_{*OPTION1*}_{*OPTION2*}_{*OPTION3*}_{*OPTION4*}**

Purpose: Define a contact interface.

OPTION1 specifies the contact type. Not all options are implemented for implicit solutions. A list of avaiable contact options is given in remark 4:

AIRBAG_SINGLE_SURFACE
AUTOMATIC_GENERAL
AUTOMATIC_GENERAL_INTERIOR
AUTOMATIC_NODES_TO_SURFACE
AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE
AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE_TIEBREAK
AUTOMATIC_SINGLE_SURFACE
AUTOMATIC_SURFACE_TO_SURFACE
AUTOMATIC_SURFACE_TO_SURFACE_TIEBREAK
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_ONE_WAY_SURFACE_TO_SURFACE
FORMING_SURFACE_TO_SURFACE
NODES_TO_SURFACE
NODES_TO_SURFACE_INTERFERENCE
ONE_WAY_SURFACE_TO_SURFACE
ONE_WAY_SURFACE_TO_SURFACE_INTERFERENCE
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
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 takes the single option:

THERMAL

Only the SURFACE_TO_SURFACE contact type 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

OPTION4 specifies that offsets may be used with the tied contacts types and takes the single option.

OFFSET

Only 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, and, if not, the nodes are projected back to the contact surface during the initialization phase. The OFFSET option switches the formulation from a constraint type formulation to one that is penalty based.

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
   NODES_TO_SURFACE

ONE_WAY_SURFACE_TO_SURFACE
FORMING_SURFACE_TO_SURFACE
FORMING_NODES_TO_SURFACE
FORMING_ONE_WAY_SURFACE_TO_SURFACE
AUTOMATIC_SURFACE_TO_SURFACE
AUTOMATIC_NODES_TO_SURFACE
AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE
AUTOMATIC_SINGLE_SURFACE
TIED_SURFACE_TO_SURFACE_OFFSET
TIED_NODES_TO_SURFACE_OFFSET
2D_AUTOMATIC_SURFACE_TO_SURFACE

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 TITLE option is inserted here; otherwise, do not define this card.**

  Define the title card first.

- **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**

  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 TITLE option is specified.

Optional 1 2

Variable	CID	NAME
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 title card 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.
NAME	Interface descriptor. It is suggested that unique descriptions be used.

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, partset 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 partset ID.
EQ.0: all part IDs are included for single surface contact, automatic single surface, and eroding single surface.

MSID — Master segment set ID, partset 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 defintion.
EQ.6: part set ID for exempted parts. All non exempted parts are included in the contact.

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.

Card 1 (*continued*)

VARIABLE	DESCRIPTION
SBOXID	BOXID, Include only slave nodes/segments within specified box, see *DEFINE_BOX, in contact definition. Only applies when SSID is defined by PART or PART SET.
MBOXID	BOXID, Include only master segments within specified box, see *DEFINE_BOX, in contact. Only applies when MSID is defined by PART or PART SET.
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) or FORCE_ TRANSDUCERS.

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.-1: If the frictional coefficients defined in the *PART section are to be used, set FS to a negative number (-1.0).

WARNING: Please note that the FS=-1.0 option applies only to contact types: SINGLE_SURFACE, AUTOMATIC_GENERAL, AUTOMATIC_SINGLE_SURFACE, AUTOMATIC_NODES_TO_SURFACE, AUTOMATIC_SURFACE_TO_SURFACE, AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE, and ERODING_SINGLE_SURFACE.

EQ. 2: For contact types SURFACE_TO_SURFACE and 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 6.1.

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.

Note: For the special contact option "TIED_SURFACE_TO_SURFACE_FAILURE" only, the variables FS and FD act as failure stresses, i.e.,

VARIABLE	DESCRIPTION
	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 6.1 for contact types and more details. 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).
DT	Death time (contact surface is deactivated at this time).

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, see also *CONTROL_ CONTACT.
SFM	Scale factor on default master penalty stiffness, 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**
***CONTACT_AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE_TIEBREAK**

Card 4	1	2	3	4	5	6	7	8
Variable	OPTION	NFLS	SFLS	CCRIT				
Type	I	F	F	F				
Default	required	required	required	option=6				

VARIABLE	DESCRIPTION
OPTION	Response: 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. 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. Damage is a nonlinear function of the crack width opening and 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 comtact. Damage is a linear function of the (maximum over time) distanc C between points initially in contact. When the distance is equal to CCRIT damage is fully developed and interface failure occurs. After failure, this contact olption behaves as a surface to surface contact. Assuming no load reversals, the energy released due to the failure of the interface is

VARIABLE	DESCRIPTION
	approximately 0.5*S*CCRIT, wher S is equal to $\sqrt{\max(\sigma_n,0)^2+\sigma_n^2}$ 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 $\frac{MIN(NFLF,SFLS)}{CCRIT}$.
NFLS	Normal failure stress for OPTION=2 or 3 below. For OPTION=5 NFLS becomes the plastic yield stress. See remark below.
SFLS	Shear failure stress for OPTION=2 or 3 below. Failure criterion: $\left(\frac{\lvert\sigma_n\rvert}{NFLS}\right)^2+\left(\frac{\lvert\sigma_s\rvert}{SFLS}\right)^2\geq 1$. For OPTION=5 SFLS becomes the load curve ID of the damage model.
CCRIT	Critical distance. Define for option 6 above.

Remarks:

After failure, this contact option behaves as a surface-to-surface contact with no thickness offsets. After failure, no interface tension is possible. The soft constraint option with SOFT=2 is not implemented for the tiebreak option.

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.

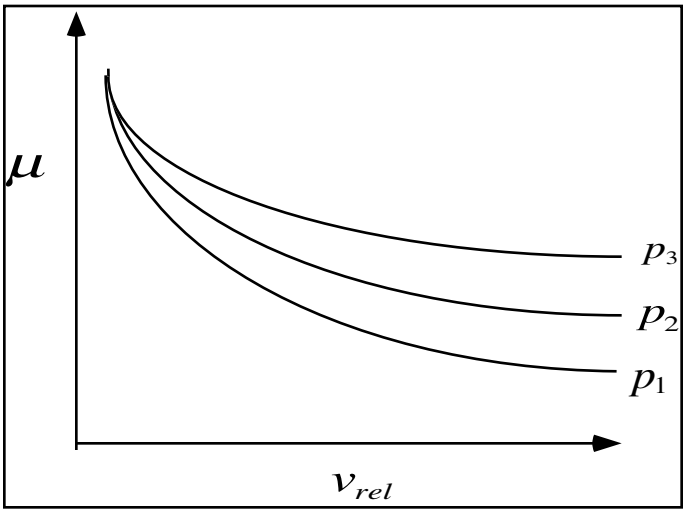


Figure 6.1. Friction coefficient, μ, can be a function of realtive velocity and pressure. Specify a flag for the static coefficient of friction, FS, and a table ID for the dynamic coefficient. This option only works with SURFACE_TO_SURFACE and ONE_WAY_SURFACE_TO_SURFACE with thickness offsets.

This Card 4 is mandatory for:

***CONTACT_ DRAWBEAD**

Card 4	1	2	3	4	5	6	7	8
Variable	LCIDRF	LCIDNF	DBDTH	DFSCL	NUMINT			
Type	I	I	F	F	I			
Default	required	none	0.0	1.0	0			

VARIABLE	DESCRIPTION
LCIDRF	If LCIDRF is positive then it defibes 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 6.2. This force is due to the bending and unbending of the blank as it moves through the drawbead. 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 drawbead length. The abscissa values is between zero and 1 and is the normalized drawbead length. The ordinate gives the maximum allowed drawbead retaining force when the bead is in the fully closed position. If the drawbead is not fully closed linear interpolation is used to compute the drawbead force.
LCIDNF	Load curve ID giving the normal force per unit draw bead length as a function of displacement, δ, see Figure 6.2. 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 explanation below.
DBDTH	Draw bead depth, see Figure 6.2. 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.

VARIABLE	DESCRIPTION
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.

Remarks:

The draw bead is defined by a *consecutive* list of *slave* nodes that lie along the draw bead. For straight draw beads only two nodes need to be defined, i.e., one at each end, but for curved beads sufficient nodes are required to define the curvature of the bead geometry. 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_*OPTION*) 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.

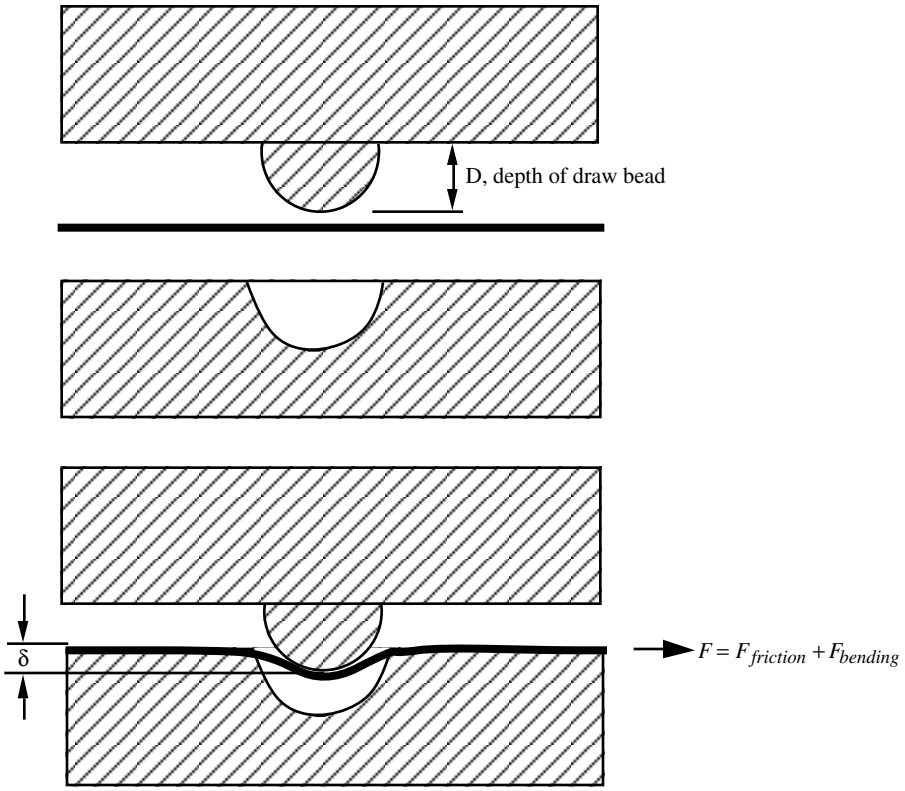


Figure 6.2. Draw bead contact model defines a resisting force as a function of draw bead displacement. The friction force is computed from the normal force in the draw bead and the given friction coefficient.

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.

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 of 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 is 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					
Type	I	I	F					
Default	required	required	from LCID					

VARIABLE	DESCRIPTION
LCID	Load curve ID giving force versus penetation 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 not be larger than the maximum value used in the loading curve.

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**

Card 4	1	2	3	4	5	6	7	8
Variable	NFLS	SFLS	TBLCID					
Type	F	F	I					
Default	required	required	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.

Remarks:

The failure attributes can be overridden segment by segment on the *SET_SEGMENT or *SET_SHELL_option cards for the **slave surface only** 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, this contact option behaves as a surface-to-surface contact with no thickness offsets. After failure, no interface tension is possible.

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	CF	FRAD	HTC	GCRIT	GMAX	CD_FACT		
Type	F	F	F	F	F	F		
Default	none	none	none	none	none	1.0		

VARIABLE — **DESCRIPTION**

CF — 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.

FRAD — Radiation factor, f, between the slide surfaces. A radient-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

$$h = h_{cond} + h_{rad}$$

HTC — 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.

GCRIT — Critical gap (l_{min}), use the heat transfer conductance defined (HTC) for gap thicknesses less than this value.

VARIABLE	DESCRIPTION
GMAX	No thermal contact if gap is greater than this value (l_{max}).
CD_FACT	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

Remarks:

In summary:

$h = h_{cont}$, 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 $l_{gap} > l_{max}$

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	EDGE	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			type 13		

VARIABLE — **DESCRIPTION**

SOFT — Soft constraint option:
EQ.0: penalty formulation,
EQ.1: soft constraint formulation,
EQ.2: pinball 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 modulii. 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 alternate 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).

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. This factor allows an increase in the size of the segments . May be useful at sharp corners.

EDGE — Edge-to-edge penetration check for alternate penalty formulation (SOFT=2).
EQ.0: Check only surface penetrations (default).
GT.0: Check both surface and edge-edge penetrations.

Optional Card A (*continued*)

VARIABLE	DESCRIPTION
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.
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.

Remark:

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 timestep 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. 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 alternate 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. The EDGE parameter on optional card A is used to enable a segment edge to segment edge penetration check. Setting EDGE=0 disables this check and is recommended when edge penetrations are not likely to occur. Setting EDGE>0 enables the edge-edge penetration judgment and EDGE=1 is recommended. Smaller values may be tried if problems occur when the EDGE option is active. In this version , all parameters on the first three cards are active except for VC, and VSF. Only the SOFT and EDGE parameters on optional card A are active. Only the ISYM parameter on optional card B is active.

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 6.1):

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:

EQ.0: thickness is not considered,

EQ.1: thickness is considered but rigid bodies are excluded,

EQ.2: thickness is considered including rigid bodies.

VARIABLE	DESCRIPTION
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 with 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						
Type	I	I						
Default	2	0						
Remarks								

VARIABLE	DESCRIPTION
IGAP	Flag to improve implicit convergence behavior at the expense of creating some sticking if parts attempt to separate. (IMPLICIT ONLY) EQ. 1: apply method to improve convergence EQ. 2: do not apply method (DEFAULT)
IGNORE	Ignore initial penetrations in the *CONTACT_AUTOMATIC options. This option can also be specified for each interface. The value defined here will be the default. 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: Move nodes to eliminate initial penetrations in the model definition.

General Remarks on *CONTACT:

1. TIED_NODES_TO_SURFACE

   TIED_SHELL_EDGE_TO_SURFACE

   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_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 sufaces 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 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; 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

$$\delta = abs(\delta_1)$$

2. The contact algorithm for tying spotwelds 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.

3. 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 sheetmetalforming, 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.

4. 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.

5. 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.

6. 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. 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.

7. FORMING_... These contacts are mainly used for metal forming applications. A connected mesh is not required for the master (tooling) side but the orienation 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.

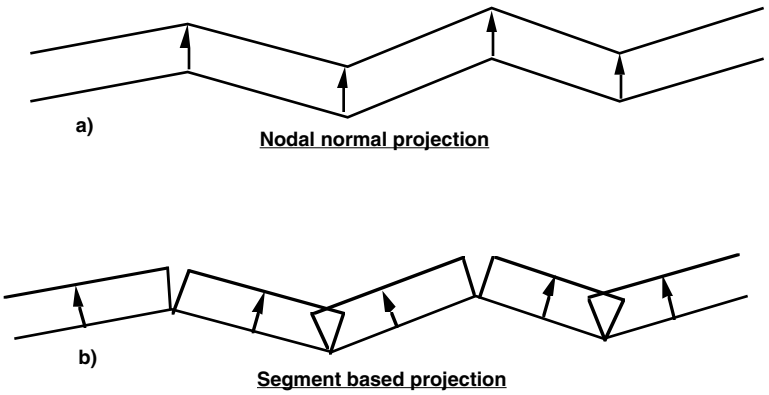


Figure 6.3. Nodal normal and segment based projection is used in the contact options.

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 6.1. 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.

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
$
$  Additonal 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

$
$$$$ Card 4 required because it's a drawbead contact
$
$  lcdidrf    lcidnf     dbdth     dfscl    numint
         3             0.17436       2.0
$
$  lcdidrf =       3  load curve 3 specifies the bending component of the
$                       restraining force per unit draw bead length
$    dbdth = 0.17436  drwa bead depth
$    dfscl =     2.0  scale load curve 3 (lcdidrf) by 2
$
$$$$  Optional Cards A and B not specified (default values will be used).
$
$
*DEFINE_BOX
$    boxid       xmm       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_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 multilinear 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 Format

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 F, EQ.7: CAL3D/MADYMO Ellipsoid, see Appendix F, EQ.8: VDA surface, see Appendix I,

VARIABLE	DESCRIPTION
	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.
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.

Remark:

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 Format

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.

Cards 3 and 4 Format

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	1	2	3	4	5	6	7	8
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 Format

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 F).
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 6.4a and 6.4b show the definitions of the geometric contact entities. The relationships between the entity coefficients and the Figure 6.4a and 6.4b 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)

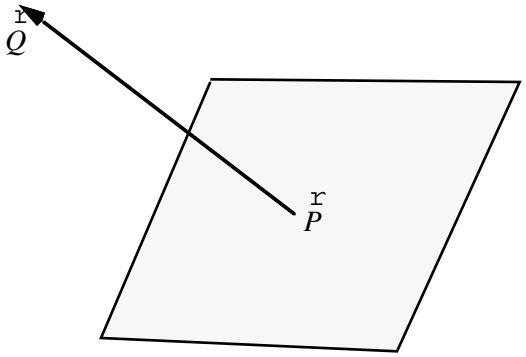
GEOTYP =10: g1 = Length of edge along X′ axis
g2 = Length of edge along Y′ axis

GEOTYP=11: g1 =Load curve ID defining axisymmetric surface profile about Z′-axis

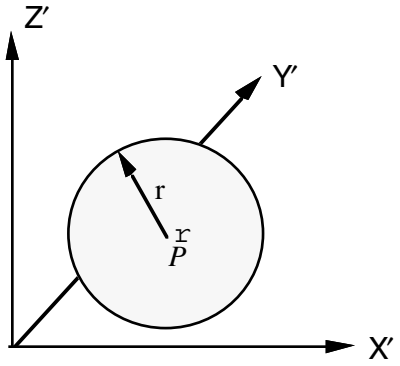
g2 = Number of elements along circumference
EQ.0: default set to 10

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

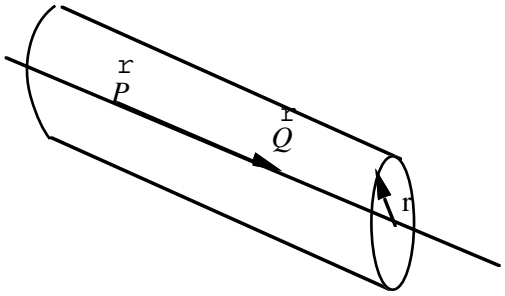
g4 = Number of sub divisions on load curve used to calculate contact
EQ:0 default set to 1000



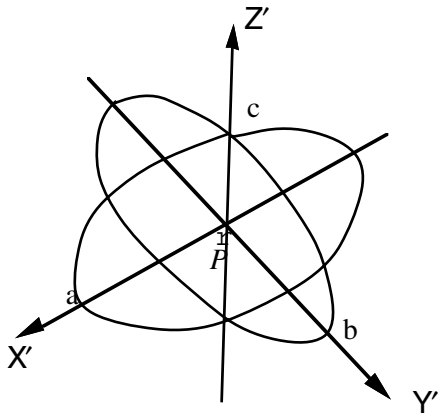
IGTYPE= 1: Infinite Plane



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 6.4a. Contact Entities.

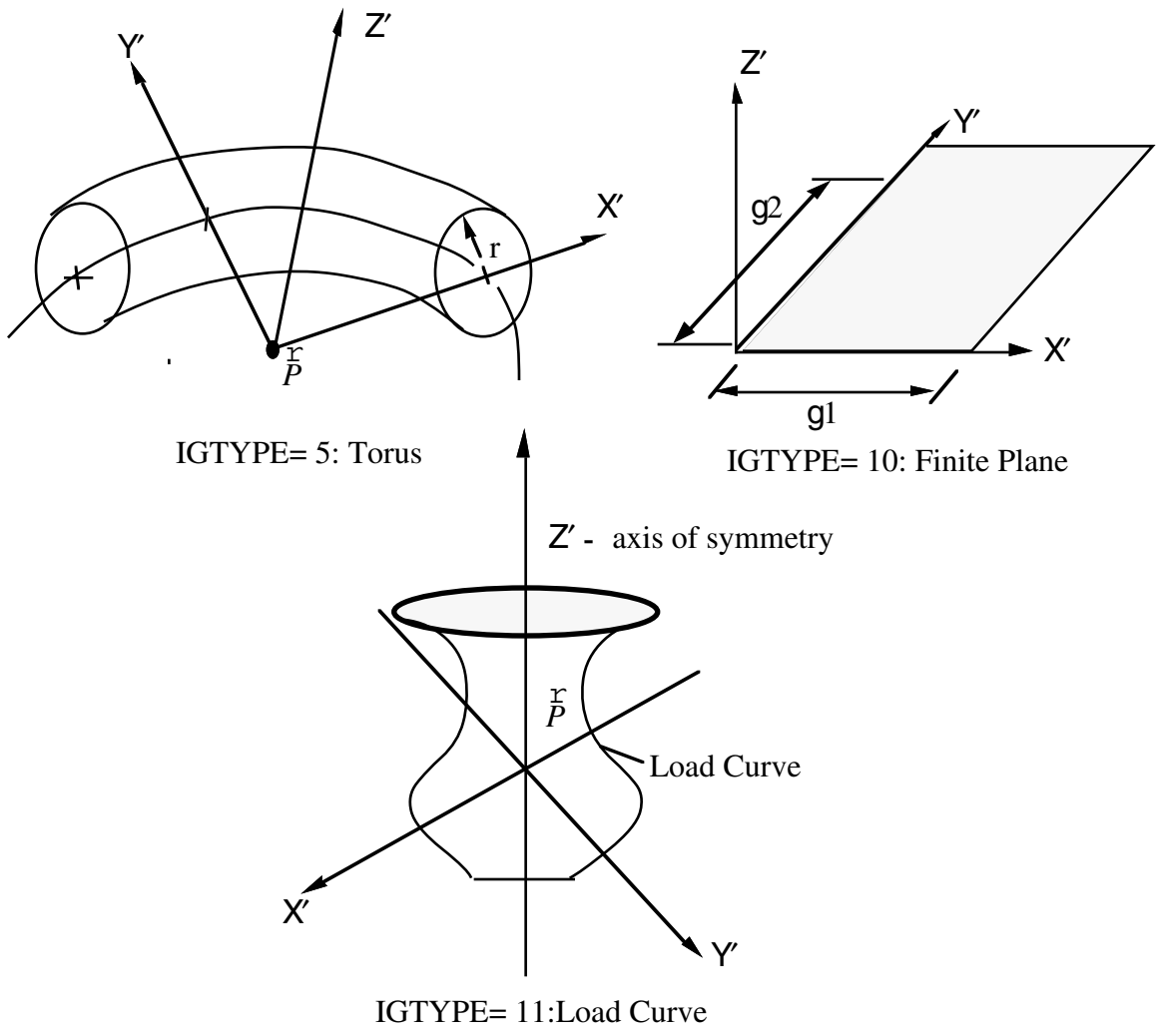


Figure 6.4b. Contact Entities.

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 K 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 LSDYNA'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 Format

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 Format

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 hose typically used with faceted contact surfaces.

*CONTACT_INTERIOR

Purpose: Define interior contact for foam brick 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 Format

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.

Three 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.

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 consitutive 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 Format

Card 1

Variable	CID	PSID	BOXID	SEGID	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			
Type	F	F	F	F	F			
Default	1.0	0.0	1.0	4.0	10			

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.
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.

VARIABLE	DESCRIPTION
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.

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-POST currently displays rigid surfaces and animates their motion.

*CONTACT_1D

Purpose: Define one-dimensional slide lines for rebar in concrete.

Card Format

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

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 Format

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

Card 2 Format

Card 2	1	2	3	4	5	6	7	8
Variable	EXT_PAS	THETA1	THETA2	TOL_IG	PEN			
Type	I	F	F	F	F			
Default	none	none	none	0.001	0.1			

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				
Default	none	none	none	none				

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.
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 first master segment.
TOL_IG	Tolerance for determing initial gaps. EQ:0.0 default set to 0.001
PEN	Scale factor or penalty. EQ:0.0 default set to 0.10
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	1	2	3	4	5	6	7	8
Variable	PSIDS	PSIDM	SFACT	FREQ	FS	FD	DC	MEMBS
Type	I	I	F	I	F	F	F	I
Default	none	none	1.0	50	0.	0.	0.	6
Remarks	1,2	1,2						

Card 2	1	2	3	4	5	6	7	8
Variable	TBIRTH	TDEATH	SOS	SOM	NDS	NDM	IPF/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		

This Card is mandatory for the THERMAL option, i.e.,:

***CONTACT_ AUTOMATIC_..._THERMAL_.....**

Optional	1	2	3	4	5	6	7	8
Variable	CF	FRAD	HTC	GCRIT	GMAX	CD_FACT		
Type	F	F	F	F	F	F		
Default	none	none	none	none	none	1.0		

VARIABLE	DESCRIPTION
PSIDS	Part set ID to define the slave surface, see *SET_PART.
PSIDM	Part set ID to define the master surface, see *SET_PART. Do not define if single surface contact is desired.
SFACT	Scale factor for the penalty force stiffness.
FREQ	Search frequency. The number of timesteps between bucket sorts. 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 $\mu_c = FD + (FS - FD)e^{-DC \cdot \lvert v_{rel} \rvert}$.
MEMBS	Parameter to allocate memory for bucket sort pair information.
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

VARIABLE	DESCRIPTION
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
IPF	Initial penetration flag for explicit analysis EQ.0: Allow initial penetrations to remain EQ.1: Push apart initially penetrated surfaces
COF	Closing/Opening flag for implicit analysis EQ.0: Recommended for most problem where gaps are only closing. EQ.1: Recommended when gaps are opening to aviod sticking.
INIT	Special processing during initializaion EQ.0: No special processing. EQ.1: Forming option.
CF	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.
FRAD	Radiation factor, f, between the slide surfaces. A radient-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 $$h = h_{cond} + h_{rad}$$
HTC	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.
GCRIT	Critical gap (l_{min}), use the heat transfer conductance defined (HTC) for gap thicknesses less than this value.
GMAX	No thermal contact if gap is greater than this value (l_{max}).

CD_FACT — 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

Remarks:

Remarks 1 through 9 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 2D solid and 2D shell parts are permitted in the master part set. Neither the slave part set ID nor 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 though 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 continnum 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.

9. For the thermal option:

$h = h_{cont}$, if the gap thickness is $0 \le l_{gap} \le l_{\min}$

$h = h_{cond} + h_{rad}$, if the gap thickness is $l_{min} \le l_{gap} \le l_{max}$

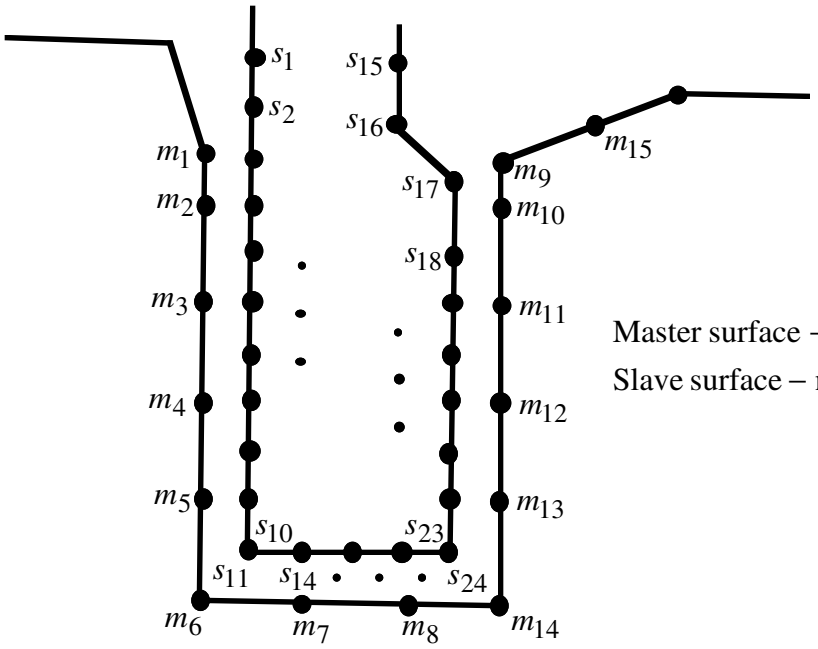
$h = 0$, if the gap thickness is $l_{gap} > l_{max}$

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 6.5. 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 6.6 and 6.7, 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 6.5. 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).

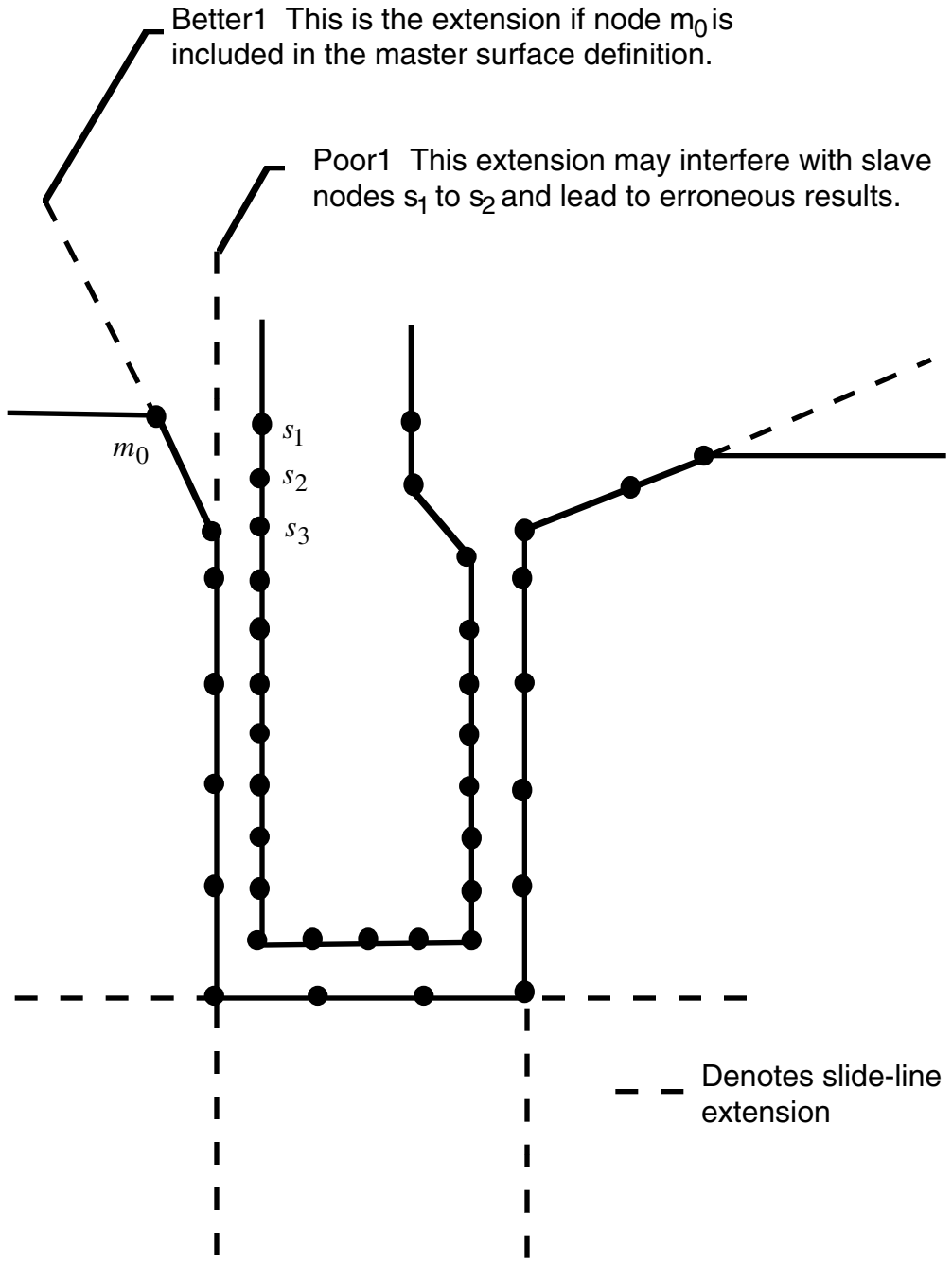


Figure 6.6. Master surface extensions defined automatically by DYNA (extensions are updated every time step to remain tangent to ends of master sides of slidelines unless angle of extension is defined in input).

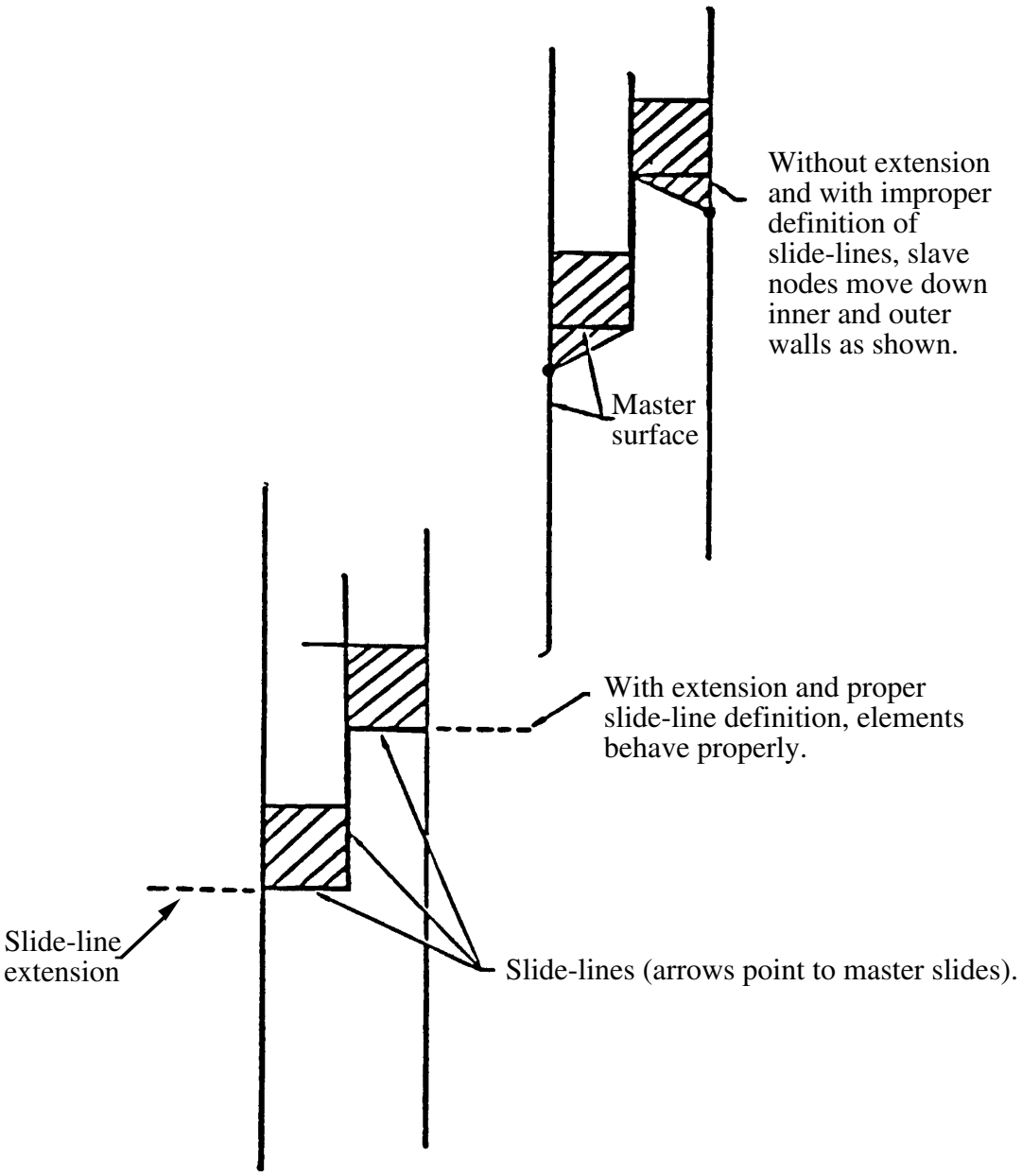


Figure 6.7 Example of slideline extensions helping to provide realistic response.

*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 an alphabetical order:

***CONTROL_ACCURACY**
***CONTROL_ADAPSTEP**
***CONTROL_ADAPTIVE**
***CONTROL_ALE**
***CONTROL_BULK_VISCOSITY**
***CONTROL_CFD_AUTO**
***CONTROL_CFD_GENERAL**
***CONTROL_CFD_MOMENTUM**
***CONTROL_CFD_PRESSURE**
***CONTROL_CFD_TRANSPORT**
***CONTROL_CFD_TURBULENCE**
***CONTROL_COARSEN**
***CONTROL_CONTACT**
***CONTROL_COUPLING**
***CONTROL_CPU**
***CONTROL_DYNAMIC_RELAXATION**
***CONTROL_ENERGY**
***CONTROL_EXPLOSIVE_SHADOW**
***CONTROL_HOURGLASS_**{*OPTION*}
***CONTROL_IMPLICIT_AUTO**
***CONTROL_IMPLICIT_DYNAMICS**
***CONTROL_IMPLICIT_EIGENVALUE**
***CONTROL_IMPLICIT_GENERAL**
***CONTROL_IMPLICIT_SOLUTION**
***CONTROL_IMPLICIT_SOLVER**
***CONTROL_IMPLICIT_STABILIZATION**
***CONTROL_OUTPUT**
***CONTROL_PARALLEL**
***CONTROL_RIGID**

***CONTROL_SHELL**

***CONTROL_SOLID**

***CONTROL_SOLUTION**

***CONTROL_SPH**

***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 a specific 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. Seven 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_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_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 Format

Card 1	1	2	3	4	5	6	7	8
Variable	OSU	INN	PIDOSU					
Type	I	I	I					
Default	0 (off)	1 (off)	optional					

VARIABLE	DESCRIPTION
OSU	Global flag for objective stress updates (See Remark 1 below). Generally, for explicit calculations only those parts undergoing large rotations, such as rolling tires, need this options. 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	Invarient node numbering for shell element (See Remark 2 below). EQ.1: Off (default) EQ.2: On
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. 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 significant 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, Belyschko-Tsay with warping stiffness, Belyschko-Chiang-Wong, S/R Hughes-Liu, and the type 16 fully integrated shell element.

2. Invarient 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 invarient node numbering, permuting the nodes shifts the local system by an exact multiple of 90 degrees. In spite of its higher costs [<5%], the invarient 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.

*CONTROL_ADAPSTEP

Purpose: Define control parameters for contact interface force update during each adaptive cycle.

Card Format

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_SUR-FACE, and *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.

Card Format

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

Card Format (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	10^{20}	0	0	0.0	inactive	inactive	0	inactive

VARIABLE	DESCRIPTION
ADPFREQ	Time interval between adaptive refinements, see Figure 7.1.
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.

VARIABLE	DESCRIPTION
	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.7: 3D r-adaptive remeshing for solid elements. Solid element type 13, a tetrahedron, is 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. EQ.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].
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.
TBIRTH	Birth time at which the adaptive remeshing begins, see Figure 7.1.
TDEATH	Death time at which the adaptive remeshing ends, see Figure 7.1.
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.
ADPASS	One or two pass adaptivity flag: EQ.0: two pass adaptivity as shown in Figure 7.1a, EQ.1: one pass adaptivity as shown in Figure 7.1b.
IREFLG	Uniform refinement level. A values of 1, 2, 3, ... allow 4, 16, 64, ... elements, respectively, to be created uniformly for each original element.

VARIABLE	DESCRIPTION
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	Absolute shell thickness level below which adaptive remeshing should began. If zero, this parameter is ignored. **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 expension.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. 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.

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.

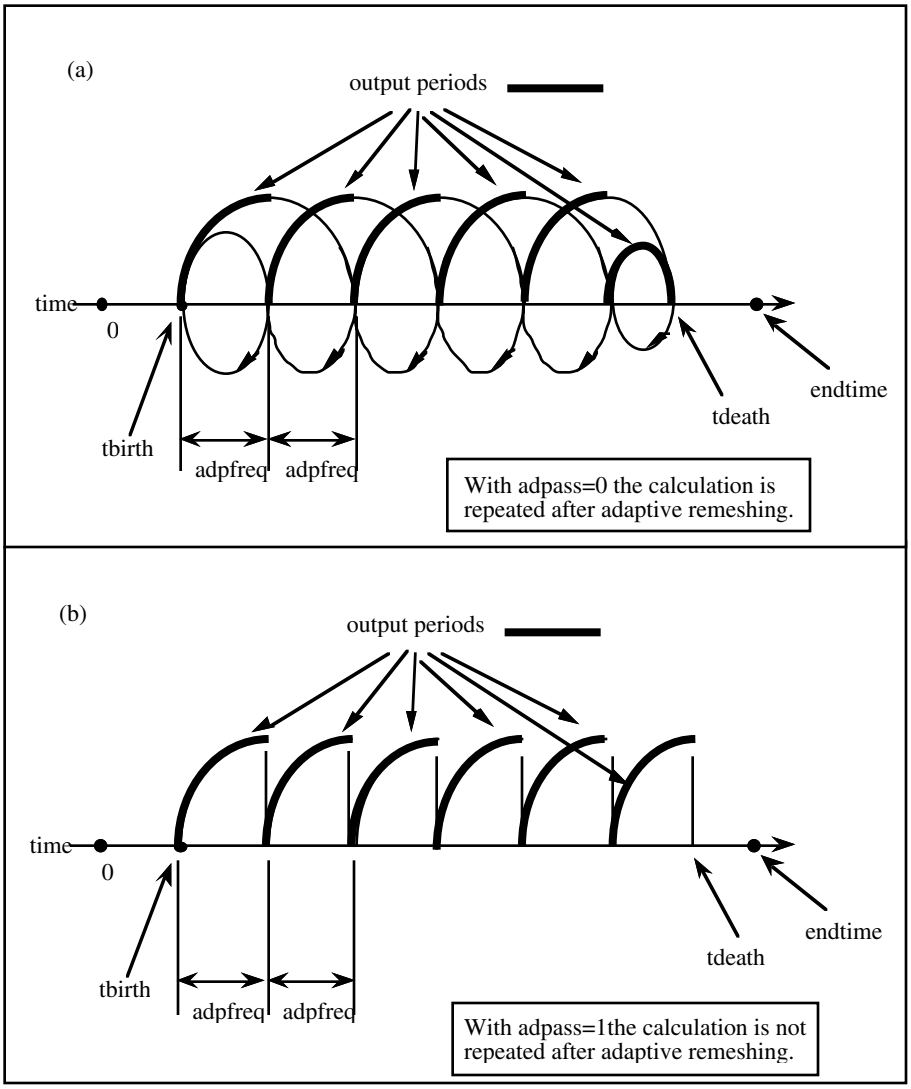


Figure 7.1. At time=*tbirth* the adaptive calculation begins. After computing for a time interval *adpfreq* error norms are computed. If ADPASS=0, then the mesh that existed at time=*tbirth* is refined based on the computed error norms. With the new mesh, the calculation continues to time=*tbirth+2 x 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_ALE

Purpose: Set default control parameters for the Arbitrary Lagrange-Eulerian and Eulerian calculations. See also *ALE_MULTI-MATERIAL_GROUP, *ALE_SMOOTHING, *INITIAL_VOID_*OPTION*., and *SECTION_SOLID_ALE.

Card Format

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	VLIMIT	EBC		
Type	F	F	F	F	F	I		
Default	0	1. 0E+20	1	1.0E-06	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.
METH	Advection method: EQ.1: donor cell + HIS (first order accuracte), EQ.2: Van Leer + HIS (second order).

VARIABLE	DESCRIPTION
AFAC	ALE smoothing weight factor - Simple average: EQ.-1: turn smoothing off.
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.
VLIMIT	Velocity limit. The time step is scaled down if the velocitiy exceed this limit.
EBC	Automatic Euler boundary condition EQ.0. off EQ.1. On with stick condition EQ.2. On with slip condition This option, used for ALE and EULER formulations, defines velocity boundary conditions for the user. Velocity boundary conditions are applied to all nodes on free surfaces of an ALE or Eulerian material. 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 ALE and Euler material is set to zero.

*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 Format

	1	2	3	4	5	6	7	8
Variable	Q1	Q2	TYPE					
Type	F	F	I					
Default	1.5	.06	1					

VARIABLE	DESCRIPTION
Q1	Default quadratic viscosity coefficient.
Q2	Default linear viscosity coefficient.
TYPE	Default bulk viscosity type, IBQ (Default=1) EQ. -1: standard (also type 2, 10, and 16 shell elements) EQ.+1: standard

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 18 in Theoretical Manual for more details

*CONTROL_CFD_AUTO

Purpose: Set the time-step control options for the Navier-Stokes flow solver. *CONTROL_CFD_GENERAL is used in conjunction with this keyword to control the flow solver time-integration options.

Card Format

	1	2	3	4	5	6	7	8
Variable	IAUTO	EPSDT	DTSF	DTMAX				
Type	I	F	F	F				
Default	0	1.0e-3	1.25	-				
Remarks	1			2				

VARIABLE	DESCRIPTION
IAUTO	Set the time step control type: EQ.0: IAUTO=1 for fixed time step size, EQ.1: Fixed time step based on DTINIT - see *CONTROL_CFD_GENERAL (default), EQ.2: Time step based on CFL/stability for INSOL=3/1, EQ.3: Automatic time step selection based on a second-order Adams-Bashforth predictor with a trapezoidal rule corrector.
EPSDT	Set the tolerance for local truncation error in time. EQ.0: EPSDT=1.0e-3 (default).
DTSF	Set the maximum time step scale factor that may be applied at any given time step. This puts the upper limit on the amount that the time step can be increased during any given time step. EQ.0: DTSF=1.25 (default).
DTMAX	Set the upper limit on the time step size. This value puts a ceiling on how far the time step may be increased for IAUTO=3. EQ.0: 10*DTINIT (default).

Remarks:

1. There are multiple solver options for a variety of flow-related physics in LS-DYNA. The selection of the time step control mechanism is dependent upon the flow solver that is selected. IAUTO=1 may be used with any of the solution methods. IAUTO=2 forces the time step to be based on either stability or the CFL number (see *CONTROL_CFD_GENERAL)

for either the explicit (INSOL=1) or the semi-implicit (INSOL=3) methods. For IAUTO=2, the ICKDT parameter may be used to control the interval at which the time step is checked and adjusted. The use of the second-order predictor-corrector time step control is restricted to the full-implicit solver, i.e., INSOL=3 and IADVEC=40.

2. For IAUTO=3, the default maximum time step, DTMAX, is set 10 times larger than the starting time step DTINIT.

*CONTROL_CFD_GENERAL

Purpose: Set solver parameters for the Navier-Stokes flow solver. ***CONTROL_CFD**_*OPTION* where *OPTION* = **MOMENTUM**, **TRANSPORT**, and **PRESSURE** are used in conjunction with this keyword to control the flow solver options. Material models may be specified with the ***MAT_CFD__***OPTION* keyword input and turbulence models are activated with the ***CONTROL_CFD_TURBULENCE** keyword input.

Card Format

	1	2	3	4	5	6	7	8
Variable	INSOL	DTINIT	CFL	ICKDT	IACURC			
Type	I	F	F	I	I			
Default	3	-	0.9 (2.0)	10	0			
Remarks	1				2			

VARIABLE	DESCRIPTION
INSOL	Set the solver type: EQ.0: INSOL=3 (default). EQ.1: Explicit, transient, incompressible Navier-Stokes, EQ.3: Semi-implicit/fully implicit, transient, incompressible Navier-Stokes using staggered velocity-pressure.
DTINIT	Set the initial time step for the Navier-Stokes and all auxiliary transport equations. The time step is computed based on either the prescribed CFL number (INSOL=3) or stability (INSOL=1) unless ICKDT<0 or IAUTO=3 on the *CONTROL_CFD_AUTO keyword.
CFL	Set the maximum advective grid-CFL number to be maintained during the computation. EQ.0: CFL=0.9 (default for INSOL=1), CFL=2.0 (default for INSOL=3).
ICKDT	Set the interval to check and report the grid Reynolds and advective CFL numbers. ICKDT<0 checks and reports the grid Reynolds and advective CFL numbers but does not modify the time step. ICKDT>0 modifies the time step according to the prescribed CFL limit and any required stability limits. The report of the grid Reynolds and CFL numbers to the screen may be toggled with the “grid” sense switch. EQ.0: ICKDT=10 (default).

IACURC	Activate the use of full-quadrature for certain terms in the momentum and transport equations. The accuracy flag improves the accuracy of body force calculations and certain advective/convective terms with a modest increase in computational time. EQ.0: don't use the increased quadruature rules (default), EQ.1: use the increased quadrature on advective/convective and body force terms.

Remarks:

1. There are multiple solver options for a variety of flow-related physics in LS-DYNA. The selection of the incompressible/low-Mach flow physics and related flow solver is determined by the INSOL input on the *CONTROL_CFD_GENERAL keyword. Currently, there are two valid values for INSOL. INSOL=1 selects the explicit time integrator that requires the use of a lumped mass matrix. In this case, the IMASS, THETAK, THETAB, THETAA and THETAF variables associated with the *CONTROL_CFD_MOMENTUM keyword are ignored. INSOL=3 selects the semi-implicit projection algorithm which makes use of these variables.

2. This option is only available for the semi-implicit/implicit solution algorithm INSOL=3.

*CONTROL_CFD_MOMENTUM

Purpose: Set the solver parameters to be used for the momentum equations in the Navier-Stokes solver. Card 1 is used to control the time integrator and advective transport options. Card 2 is used to set the linear solver options such as the maximum iteration count and interval to check the convergence criteria.

Card Format

Card 1

1	2	3	4	5	6	7	8
Variable	IMASS	IADVEC	IFCT	DIVU	THETAK	THETAA	THETAF	
Type	I	I	I	F	F	F	F	
Default	1	10	1	1.0e-5	0.5	0.5	0.5	
Remarks	1	2	3	4	5	6	6	

Card 2

1	2	3	4	5	6	7	8
Variable	MSOL	MAXIT	ICHKIT	IWRT	IHIST	EPS	IHG	EHG
Type	I	I	I	I	I	F	I	F
Default	20	100	2	0	0	1.0e-5	1	1.0
Remarks	6							

VARIABLE	DESCRIPTION
IMASS	Select the mass matrix formulation to use: EQ.0: IMASS=1 (default), EQ.1: Lumped mass matrix, EQ.2: Consistent mass matrix, EQ.3: Higher-order mass matrix.

VARIABLE	DESCRIPTION
IADVEC	Toggle the treatment of advection between explicit with balancing tensor diffusivity (BTD) or fully-implicit.. EQ.0: IADVEC=10 for forward-Euler with BTD (default), EQ.-1: IADVEC=0 for foward-Euler without BTD, EQ.10: forward-Euler with BTD, EQ.40: fully-implicit with simplified trapezoid rule.
IFCT	Toggle the use of the advective flux limiting advection scheme. EQ.0: IFCT=1 (default), EQ.1: Advective flux limiting is on, EQ.-1: Advective flux limiting is off.
DIVU	Set the RMS divergence tolerance, i.e., $\|\nabla \cdot u\|_{RMS} \le \varepsilon$. This tolerance is used for the initial startup procedure to insure that proper initial conditions are prescribed for the momentum equations. EQ.0: DIVU=1.0e-5 (default).
THETAK	Time weighting for viscous/diffusion terms. Valid values are $0 \le \theta_K \le 1$ with $\theta_K = ½$ for second-order accuracy in time. EQ.0: THETAK=0.5 (default).
THETAA	Time weighting for advection terms.
THETAF	Time weighting for body forces and boundary conditions. Valid values are $0 \le \theta_F \le 1$ with $\theta_F = ½$ for second-order accuracy in time. EQ.0: THETAF=0.5 (default).
MSOL	Set the equation solver type for the momentum equations. EQ.0: MSOL=20 (default), EQ.20: Jacobi preconditioned conjugate gradient method, EQ.30: Jacobi preconditioned conjugate gradient squared method (default when IADVEC=40).
MAXIT	Set the maximum number of iterations for the iterative equation solver. EQ.0: MAXIT=100 (default).
ICHKIT	Set the interval to check the convergence criteria for the iterative equation solver. EQ.0: ICHKIT=2 (default).
IWRT	Activate the output of diagnostic information from the equation solver. EQ.0: Diagnostic information is off (default), EQ.1: Diagnostic information is on.
IHIST	Activate the generation of a convergence history file from the equation solver. The ASCII history files are “velx.his”, “vely.his” and “velz.his” EQ.0: Convergence history is off (default), EQ.1: Convergence history is on.

VARIABLE	DESCRIPTION
EPS	Set the convergence criteria for the iterative equation solver. EQ.0: EPS=1.0e-5 (default).
IHG	Set the type of hourglass stabilization to be used with the momentum equations. This only applies to the explicit treatment of the momentum equations (INSOL=1). EQ.0: IHG=1 (default), EQ.1: LS-DYNA CFD viscous hourglass stabilization, EQ.2: γ-hourglass stabilization viscous form.
EHG	Set the hourglass stabilization multiplier. (see IHG above). EQ.0: EHG=1.0 (default).

Remarks:

1. The IMASS variable is only active when INSOL ≥ 2 on the *CONTROL_CFD_GENERAL keyword.

2. The balancing tensor diffusivity should always be used with the explicit forward-Euler treatment of the advection terms. This is the default.

3. The use of the flux limiting procedures is currently restricted to the explicit advection procedures.

4. DIVU sets the ceiling on the discrete divergence that is permitted during a simulation when INSOL=1. If the divergence at a given time step exceeds the value set by DIVU, then an intermediate projection is performed to return the velocity to a div-free state.

5. The time weighting variables only apply to the case when INSOL≥2 on the *CONTROL_CFD_GENERAL keyword.

6. The MSOL keyword for the *CONTROL_CFD_MOMENTUM keyword only applies for INSOL ≥ 2 on the *CONTROL_CFD_GENERAL keyword.

*CONTROL_CFD_PRESSURE

Purpose: Set the pressure solver parameters to be used for the incompressible Navier-Stokes equations.

Card Format

Card 1

Variable	IPSOL	MAXIT	ICHKIT	IWRT	IHIST	EPS		
Type	I	I	I	I	I	F		
Default	22	200	5	0	0	1.0e-5		
Remarks								

Card 2

Variable	NVEC	ISTAB	BETA	SID	PLEV	LCID		
Type	I	I	F	I	F	I		
Default	5	1	0.05	none	0.0	0		
Remarks								

VARIABLE	DESCRIPTION
IPSOL	Set the pressure solver type: EQ.0: IPSOL=22 for serial, IPSOL=21 for MPP (default). EQ.10: Sparse direct solver, EQ.11: PVS direct solver, EQ.20: Jacobi preconditioned conjugate gradient method, EQ.21: SSOR preconditioned conjugate gradient method, EQ.22: SSOR preconditioned conjugate gradient using the Eisenstat transformation.

VARIABLE	DESCRIPTION
MAXIT	Set the maximum number of iterations for the pressure solver. EQ.0: MAXIT=200 (default).
ICHKIT	Set the interval to check the convergence criteria for the pressure solver. EQ.0: ICHKIT=5 (default).
IWRT	Activate the output of diagnostic information from the pressure solver. EQ.0: Diagnostic information is off (default), EQ.1: Diagnostic information is on. (NOTE: during execution, sense switch "lprint" can be used to toggle this flag on or off.)
IHIST	Activate the generation of a convergence history file for the pressure solver. The ASCII history file is "ppe.his". EQ.0: Convergence history is off (default), EQ.1: Convergence history is on.
EPS	Set the convergence criteria for the pressure solver. EQ.0: EPS=1.0e-5 (default).
NVEC	Set the number of A-conjugate vectors to use during the iterative pressure solve. EQ.0: NVEC=5 (default), LT.0: A-conjugate projection is disabled.
ISTAB	Set the stabilization type. EQ.0: ISTAB=1 (default), EQ.1: Local jump stabilization, EQ.2: Global jump stabilization, EQ.-1: No stabilization is active.
BETA	Stabilization parameter for ISTAB=1,2. Valid values for the stabilization parameter are $0 \le \beta \le 1$. EQ.0: BETA=0.05 (default).
SID	Solid element set ID or shell element set ID (see SET_SOLID/ SET_SHELL_*OPTION*) to be used for the prescription of hydrostatic pressure.
PLEV	Set the hydrostatic pressure level. This value multiplies the values of the load curve specified with the LCID option. EQ.0: PLEV=0.0 (default).
LCID	Load curve to be used for setting the hydrostatic pressure. By default, LCID=0 which forces a constant pressure level to be set at the level prescribed by PLEV. EQ.0: LCID=0 (default).

*CONTROL_CFD_TRANSPORT

Purpose: Activate the calculation of transport variables and associated solver parameters to be used for the auxiliary scalar transport equations. Card 1 is used to activate the auxiliary transport equations and Card 2 is used to set the mass matrix, advection, and time-weighting options. Card 3 is used to set the linear solver options such as the maximum iteration count and interval to check the convergence criteria.

Card Format

Card 1

Variable	ITEMP		NSPEC					
Type	I		I					
Default	0		0					
Remarks								

Card 2

Variable	IMASS	IADVEC	IFCT	THETAK	THETAA	THETAF		
Type	I	I	I	F	F	F		
Default	1	10	1	0.5	0.5	0.5		
Remarks	1	2	3	4	4	4		

Card 3	1	2	3	4	5	6	7	8
Variable	ITSOL	MAXIT	ICHKIT	IWRT	IHIST	EPS	IHG	EHG
Type	I	I	I	I	I	F	I	F
Default	20	100	2	0	0	1.0e-5	1	1.0
Remarks	5							

VARIABLE	DESCRIPTION
ITEMP	Solve the energy equation in terms of temperature. EQ.0: No energy equation (default), EQ.1: Energy equation is solved in terms of temperature.
NSPEC	Activate the solution of NSPEC species transport equations. EQ.0: No species equations are solved (default), EQ.NSPEC: Solve for NSPEC species. Up to 10 species transport equations may be active ($0 \le NSPEC \le 10$).
IMASS	Select the mass matrix formulation to use: EQ.0: IMASS=1 (default), EQ.1: Lumped mass matrix, EQ.2: Consistent mass matrix, EQ.3: Higher-order mass matrix.
IADVEC	Toggle the treatment of advection between explicit with balancing tensor diffusivity (BTD) or fully-implicit.. EQ.0: IADVEC=10 for forward-Euler with BTD (default), EQ.-1: IADVEC=0 for foward-Euler without BTD, EQ.10: forward-Euler with BTD, EQ.40: fully-implicit with simplified trapezoid rule.
IFCT	Toggle the use of the advective flux limiting advection scheme. EQ.0:IFCT=1 (default), EQ.1: Advective flux limiting is on, EQ.-1: Advective flux limiting is off.
THETAK	Time weighting for viscous/diffusion terms. Valid values are $0 \le \theta_K \le 1$ with $\theta_K = ½$ for second-order accuracy in time. EQ.0: THETAK=0.5 (default).

VARIABLE	DESCRIPTION
THETAA	Time weighting for advection terms.
THETAF	Time weighting for body forces and boundary conditions. Valid values are $0 \le \theta_F \le 1$ with $\theta_F = ½$ for second-order accuracy in time. EQ.0: THETAF=0.5 (default).
ITSOL	Set the equation solver type for the transport equations. EQ.0: ITSOL=20 (default), EQ.20: Jacobi preconditioned conjugate gradient method, EQ.30: Jacobi preconditioned conjugate gradient squared method (default when IADVEC=40).
MAXIT	Set the maximum number of iterations for the iterative equation solver. EQ.0: MAXIT=100 (default).
ICHKIT	Set the interval to check the convergence criteria for the iterative equation solver. EQ.0: ICHKIT=2 (default).
IWRT	Activate the output of diagnostic information from the equation solver. EQ.0: Diagnostic information is off (default), EQ.1: Diagnostic information is on.
IHIST	Activate the generation of a convergence history file from the equation solver. EQ.0: Convergence history is off (default), EQ.1: Convergence history is on.
EPS	Set the convergence criteria for the iterative equation solver. EQ.0: EPS=1.0e-5 (default).
IHG	Set the type of hourglass stabilization to be used with the momentum equations. This only applies to the explicit treatment of the momentum equations (INSOL=1). EQ.0: IHG=1 (default), EQ.1: LS-DYNA CFD viscous hourglass stabilization, EQ.2: γ-hourglass stabilization viscous form.
EHG	Set the hourglass stabilization multiplier. (see IHG above). EQ.0: EHG=1.0 (default).

Remarks:

1. The IMASS variable is only active when INSOL≥2 on the *CONTROL_CFD_GENERAL keyword.

2. The balancing tensor diffusivity should always be used with explicit treatment of the advection terms. (This is the default.)

3. The use of flux limiting procedures is currently restricted to the explicit advective procedures.

4. The time weighting variables only apply to the case when INSOL≥2 on the *CONTROL_CFD_GENERAL keyword.

5. The ITSOL keyword for the *CONTROL_CFD_TRANSPORT keyword only applies for INSOL≥2 on the *CONTROL_CFD_GENERAL keyword.

*CONTROL_CFD_TURBULENCE

Purpose: Activate a turbulence model and set the associated model parameters.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	ITRB	SMAGC						
Type	I	F						
Default	0	0.1						
Remarks		1						

VARIABLE	DESCRIPTION
ITRB	Select the turbulence model: EQ.0: Turbulence models are disabled (default), EQ.1: Smagorinsky LES sub-grid scale model, EQ.101: Spalart-Almaras model.

Remarks:

1. The default value of the Smagorinsky constant is $C_s = 0.1$.

*CONTROL_COARSEN

Purpose: Adaptively de-refine (coarsen) a shell mesh by selectively merging four adjcent elements into one. Adaptive constraints are added and removed as necessary.

Card Format

Card 1

Variable	ICOARSE	ANGLE	NSEED					
Type	I	F	I					
Default	0	none	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
ANGLE	Allowable angle change between neighboring elements. Adjcent 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)
N1...N8	Optional list of seed node IDs for extra searching.

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_DYNA3D, 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 Format

Card 1	1	2	3	4	5	6	7	8
Variable	SLSFAC	RWPNAL	ISLCHK	SHLTHK	PENOPT	THKCHG	ORIEN	ENMASS
Type	F	F	I	I	I	I	I	I
Default	.1	none	2	0	1	0	1	0

Card 2	1	2	3	4	5	6	7	8
Variable	USRSTR	USRFRC	NSBCS	INTERM	XPENE	SSTHK	ECDT	TIEDPRJ
Type	I	I	I	I	F	I	I	I
Default	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.

Card 3	1	2	3	4	5	6	7	8
Variable	SFRIC	DFRIC	EDC	VFC	TH	TH_SF	PEN_SF	
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 is optional. If this card is defined, then Card 3 above must be included. A blank card may be inserted for Card 3.

Card 3	1	2	3	4	5	6	7	8
Variable	IGNORE	FRCENG						
Type	I	I						
Default	0	0						

VARIABLE	DESCRIPTION
SLSFAC	Scale factor for sliding interface penalties, SLSFAC: EQ.0: default = .1.
RWPNAL	Scale factor for rigid wall penalties for treating rigid bodies interacting with fixed rigid walls, RWPNAL. The penalties are set so that a scale factor of unity should be optimal; however, this 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. EQ.0.0: rigid bodies interacting with rigid walls are not considered. GT.0.0: rigid bodies interact with fixed rigid walls. A value of 1.0 is recommended. Seven (7) variables are stored for each slave node. This can increase memory requirements significantly if all nodes are slaved to the rigid walls.
ISLCHK	Initial penetration check in contact surfaces with indication of initial penetration in output file, ISLCHK (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 Theoretical Manual. EQ.0: the default is set to 1, EQ.1: minimum of master segment and slave node (default for most contact types),

VARIABLE	DESCRIPTION
	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 are recommended for metalforming calculations.
THKCHG	Shell thickness changes considered in single surface contact: EQ.0: no consideration (default), EQ.1: shell thickness changes are included.
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.
ENMASS	Treatment of the mass of eroded nodes in contact. This option effects 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 D. 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 E. 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.

VARIABLE	DESCRIPTION
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 *CONTROL_*OPTION*_...: EQ.0: default is set to 4.0.
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. 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.
FRCENG	Flag to activate the calculation of frictional sliding energy: EQ.0: do not calculate, EQ.1: calculation frictional energy in contact.

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, and ERODING_SINGLE_SURFACE, the default contact thickness is taken as the smaller value of the shell thickness or the shell edge lengths between shell nodes 1-2, 2-3, and 4-1. 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.

*CONTROL_COUPLING

Purpose: Change defaults for MADYMO3D/CAL3D coupling, see Appendix F.

Card Format

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-DYNA3D remains inactive. Important for saving computer time.
FLIPX	Flag for flipping X-coordinate of CAL3D/MADYMO3D relative to the LS-DYNA3D model: EQ.0: off, EQ.1: on.
FLIPY	Flag for flipping Y-coordinate of CAL3D/MADYMO3D relative to the LS-DYNA3D model: EQ.0: off, EQ.1: on.
FLIPZ	Flag for flipping Z-coordinate of CAL3D/MADYMO3D relative to the LS-DYNA3D 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 Format

	1	2	3	4	5	6	7	8
Variable	CPUTIM							
Type	F							

VARIABLE	DESCRIPTION
CPUTIM	Seconds of cpu time: EQ:0.0 no cpu time limit set

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: Define controls for dynamic relaxation. Important for stress initialization.

Card Format

	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								1, 2

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.
IRELAL	Automatic control for dynamic relaxation option based on algorithm of Papadrakakis [Papadrakakis 1981]: EQ.0: not active, EQ.1: active.
EDTTL	Convergence tolerance on automatic control of dynamic relaxation.

VARIABLE	DESCRIPTION
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 below**, EQ.0: not active, EQ.1: dynamic relaxation is activated, EQ.2: initialization to a prescribed geometry, **see Remark 1 below**

Remarks:

1. Stress initialization in LS-DYNA for small strains may be accomplished by linking to an implicit code (option 2). A displacement state is required that gives for each nodal point its label, xyz displacements, xyz rotations and temperature. This data is read from unit 7 (m=) with the format (i8,7e15.0). See also INTRODUCTION, Execution Syntax.

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 writen 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.

*CONTROL_ENERGY

Purpose: Provide controls for energy dissipation options.

Card Format

	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 possiblities 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_HOURGLASS_{*OPTION*}**

One option is available:

936

which switches the hourglass formulation so that it is identical to that used in version 936 of LS-DYNA. 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 Format

	1	2	3	4	5	6	7	8
Variable	IHQ	QH						
Type	I	F						
Default	1	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 activates warping stiffness for accurate solutions. A speed penalty of 25% is common for this option.

VARIABLE	DESCRIPTION
	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 recommeded and for low velocities the stiffness forms are recommended. For large deformations and nonregular 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.

Remark:

1. Hourglass coefficients and type can be set by part ID in the ***HOURGLASS** Section.

*CONTROL_IMPLICIT_AUTO

Purpose: Define parameters for automatic time step control during implicit analysis.

Card Format

	1	2	3	4	5	6	7	8
Variable	IAUTO	ITEOPT	ITEWIN	DTMIN	DTMAX			
Type	I	I	I	F	F			
Default	(see below)	11	15	DT/1000.	DT*10.			

VARIABLE	DESCRIPTION
IAUTO	Automatic time step control flag EQ.0: constant time step size (Default for "standard" implicit analysis) EQ.1: automatically adjusted time step size (Default for "springback" implicit analysis)
ITEOPT	Optimum equilibrium iteration count per time step
ITEWIN	Allowable iteration window. If iteration count is within ITEWIN iterations of ITEOPT, step size will not be adjusted.
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: load curve gives DTMAX(t), each time point reached exactly (see Figure 7.2)

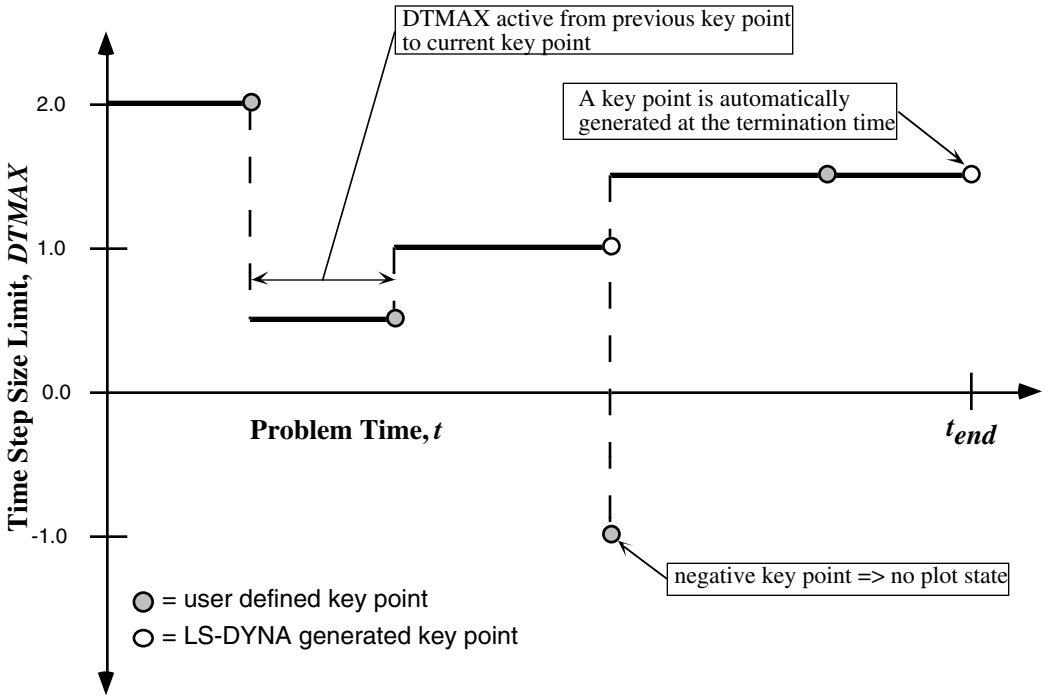


Figure 7.2. A *key point load curve* can be identified with a negative value for DTMAX. Function values of each load curve point give DTMAX. Time values are reached exactly by the automatic step controller, and a plot state is output unless DTMAX is negative.

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 The time step size is adjusted so that equilibrium is reached in ITEOPT iterations, increasing after "easy" steps, and decreasing after "difficult" but successful steps. A value of ITEOPT=21 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, use a key point load curve (Figure 7.2) and enter DTMAX = -(curve ID).

*CONTROL_IMPLICIT_DYNAMICS

Purpose: Activate implicit dynamic analysis and define time integration constants.

Card Format

	1	2	3	4	5	6	7	8
Variable	IMASS	GAMMA	BETA					
Type	I	F	F					
Default	0	.50	.25					

VARIABLE	DESCRIPTION
IMASS	Implicit analysis type EQ.0: static analysis EQ.1: dynamic analysis using Newmark time integration. EQ.2: dynamic analysis by modal superpostion
GAMMA	Newmark time integration constant (see remarks below.)
BETA	Newmark time integration constant.

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.

The time integration is by the unconditionally stable, one-step, Newmark-β time integration scheme

$$\ddot{u}^{n+1} = \frac{\Delta u}{\beta \Delta t^2} - \frac{\dot{u}^n}{\beta \Delta t} - \frac{1}{\beta}\left(\frac{1}{2} - \beta\right)\ddot{u}^n$$

$$\dot{u}^{n+1} = \dot{u}^n + \Delta t(1-\gamma)\ddot{u}^n + \gamma \Delta t \ddot{u}^{n+1}$$

$$x^{n+1} = x^n + \Delta u$$

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.

*CONTROL_IMPLICIT_EIGENVALUE

Purpose: Activate implicit eigenvalue analysis and define associated input parameters.

Card Format

	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

VARIABLE	DESCRIPTION
NEIG	Number of eigenvalues to extract. This must be specified. The other parameters below are optional.
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.
RHTEND	Right end point of interval. Only used when RFLAG = 1.
EIGMTH	Eigenvalue extraction method: EQ.1: Subspace iteration (not recommended). EQ.2: Block Shift and Invert Lanczos (default).
SHFSCL	Shift scale. Generally not used, but see explanation below.

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.

It is strongly recommended that the default eigenvalue extraction method, Block Shift and Invert Lanczos, is used. The Block Shift and Invert Lanczos code is from BCSLIB-EXT, Boeing's Extreme Mathematical Library. This method is much more robust and efficient than subspace iteration. Subscape iteration is only included for comparison and testing purposes.

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.

The print control parameter, LPRINT, and ordering method paramenter, ORDER, from the *CONTROL_IMPLICIT_LINEAR keyword card also affects the Block Shift and Invert Eigensolver. LPRINT and LSOLVR affects Subspace Iteration.

*CONTROL_IMPLICIT_GENERAL

Purpose: Define control parameters for implicit analysis.

Card Format

	1	2	3	4	5	6	7	8
Variable	IMFLAG	DT0	IMFORM	NSBS	IGS	CNSTN	FORM	
Type	I	F	I	I	I	I	I	
Default	0	none	2	1	2	0	0	

(see remarks below)

VARIABLE	DESCRIPTION
IMFLAG	Implicit/Explicit switching flag EQ.0: explicit analysis EQ.1: implicit analysis EQ.2: explicit followed by one implicit step ("*springback*" analysis)
DT0	Initial time step size for implicit analysis
IMFORM	Element formulation switching flag EQ.1: switch to fully integrated formulation for implicit springback. Recommended for stability. EQ.2: retain original element formulation (default)
NSBS	Number of steps in nonlinear springback
IGS	Geometric (initial stress) stiffness flag EQ.1: include EQ.2: ignore
CNSTN	Indicator for consistent tangent stiffness: EQ.0: do not use (default) EQ.1: use.
FORM	Element formulation when using IMFORM flag. EQ.0: type 16 EQ.1: type 6.

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 when the termination time is reached. 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.

DT0 — This parameter selects the initial time step size for the implicit phase of a simulation. In a springback simulation, the default initial time step size is the termination time from the explicit forming phase of the simulation. The step size may be adjusted during a multiple step simulation if the automatic time step size control feature is active.

IMFORM — The default Belytschko-Tsay shell element works well for forming analysis, but can perform poorly for springback analysis. This element formulation switching flag causes the more stable fully integrated shell elements to be used for the springback phase. Adaptive mesh must be activated when using element formulation switching.

NSBS — The default springback analysis is nonlinear, single step. When the automatic time step control and artificial stabilization are activated, a multiple step solution will be performed automatically if the single step fails. The NSBS option allows a user to skip the single step attempt, and proceed directly to a multiple step solution. *Artificial stabilization must be used for all multiple step solutions.*

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, so its inclusion is optional.

*CONTROL_IMPLICIT_SOLUTION

Purpose: Define these control cards for an implicit calculation. These cards specify whether a linear or nonlinear solution is desired. If nonlinear, set the parameters to control the implicit nonlinear solution.

Card 1 Format

	1	2	3	4	5	6	7	8
Variable	NSOLVR	ILIMIT	MAXREF	DCTOL	ECTOL	(blank)	LSTOL	
Type	I	I	I	F	F	F	F	
Default	2	11	15	0.001	0.01		0.90	

Optional Card 2

	1	2	3	4	5	6	7	8
Variable	DNORM	DIVERG	ISTIF	NLPRINT				
Type	I	I	I	I				
Default	2	1	1	2				

Optional Card 3 (if card 3 is used, then card 2 above must also be used)

	1	2	3	4	5	6	7	8
Variable	ARCCTL	ARCDIR	ARCLEN	ARCMTH	ARCDMP			
Type	I	I	F	I	I			
Default	0	none	0	1	2			

(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
ILIMIT	Iteration limit between automatic stiffness reformations
MAXREF	Stiffness reformation limit per time step
DCTOL	Displacement convergence tolerance
ECTOL	Energy convergence tolerance
LSTOL	Line search 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) 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.1: print iteration information to screen, messag, d3hsp files EQ.2: print information only to messag, d3hsp files (default) (NOTE: during execution, sense switch "nlprint" can also be used to toggle this print flag on and off.)

VARIABLE	DESCRIPTION

The following parameters are for use with arc length methods only (6 ≤ NSOLVR ≤ 9):

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.
LE.0.0: use automatic size size,
GT.0.0: use ARCLEN*automatic step size.

ARCMTH — Arc length method
EQ.1: Crisfield (default)
EQ.2: Ramm

ARCDMP — Arc length damping option
EQ.2: off (default)
EQ.1: on, oscillations in static solution are supressed

Remarks:

NSOLVR — This flag may be used to select a linear springback analysis. This disables equilibrium checking and iterations. The default nonlinear BFGS method can be used as a Full Newton method by resetting the ILIMIT parameter below.

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 equaiton to limit the load step to a constatnt "arc Length" in load-displacement space. This latter method is frequently used to solve snap through buckling problems. When applying the arc-length method, the load curves that define the loading should contain two points and start at the origin (0,0). If the arc-length method is flagged and if two points characterize the load curve, LS-DYNA will extrapolate, if necessary, to determine the load. Time and load magnitude are related by a constant when the arc length method is used and it is possible that time can be negative. The arc length apply cannot be used with 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.

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 ratio (shown for each iteration) is reduced below DCTOL, 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.

ECTOL When the energy ratio shown for each iteration 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.

LSTOL A line search is sometimes performed to this tolerance to guard against divergence. The search is done in the event that the system is stiffening which can often lead to a failure to converge during the equilibrium iterations.

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 supress this stiffness reformation.

ISTIF By default, a new stiffness matrix is formed at the start of every time step. Supressing this stiffness reformation can decrease the cost of simulations which have many tiny steps that are mostly linear.

NLPRT 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. See also the section on interactive sense switches "<ctrl-c> nlprt" and "<ctrl-c> iter".

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 defalut), 0.5, 0.25, and 0.10.

ARCDMP Some static problems exhibit oscillatory response near instability points. This option numerically supresses these oscillations, and may improve the convergence behavior of the post-buckling solution.

*CONTROL_IMPLICIT_SOLVER

Purpose: Define control parameters for the implicit analysis linear equation solver. The linear equation solver performs the CPU-intensive stiffness matrix inversion.

Card Format

	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

(see remarks below)

VARIABLE — **DESCRIPTION**

LSOLVR — Linear equation solver method
- EQ.1: direct, sparse, incore with automatic out-of-core
- EQ.3: direct, sparse, double precision
- EQ.4: SMP parallel multi-frontal sparse solver #2 (default).
- EQ.5: SMP parallel multi-frontal sparse solver # 2, 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

LPRINT — Linear solver print flag
- EQ.0: no printing
- EQ.1: summary statistics on memory, cpu time, and iteration count.
- EQ.2: more statistics
- EQ.3: even more statistics and debug checking, e.g., residual of eigenvalues and eigenvectors.

(NOTE: during execution, sense switch "lprint" can also be used to toggle this print flag on and off.)

NEGEV — Negative eigenvalue flag. Selects procedure when negative eigenvalues are detected during stiffness matrix inversion.
- EQ.1: stop or retry step
- EQ.2: print warning message, try to continue (default)

VARIABLE	DESCRIPTION
ORDER	Ordering option EQ.0: method set automatically by LS-DYNA EQ.1: MMD, Multiple Minimum Degree. EQ.2: Metis
DRCM	Drilling rotation constraint methoc: EQ.1: add stiffness (default) EQ.2: generate geometry based drilling constraint, EQ.3: do neither.
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. If generate geometry based drilling constraints is active, DRCM=2, then DRCPRM controls the "flatness" test. The default value of DRCPRM=10.0 for this case. If the maximum deviation of a neighbor node to the best fit plane at a candidate node is less than this parameter the local geometry is declared flat and a constraint is generated on the rotation around the outward pointing normal at the candidate node.
AUTOSPC	AUTOSPC switch: EQ.1: automatically scan the assembled stiffness matrix after all constraints have been applied looking for triples of columns associated with translations or rotations at a node or master of a rigid body. If the set of 3 columns is rank deficient, a constraint is generated to remove the column most associated with the null space of the singularity, EQ.2: do not do the scan.
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.

Remarks:

LSOLVR The user can select one of 5 direct factorization methods and 6 iterative methods. Solver options 4 (default) and 5 are updated versions of options 1 and 3. The updates include faster single cpu performance, parallel implementation, and the ability to select either MMD or Metis ordering (see ORDER). Options 1 and 3 are still included for backward compatibility with previous versions.

The direct linear equation solver from BCSLIB-EXT, Boeing's Extreme Mathematical Library is option 6. 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. It is also faster than the older options 1 and 3.

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".

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.

ORDER The system of linear equations must be reordered to preserve the sparsity of the factorization when using direct methods. The older sparse solvers (LSOLVR = 1 and LSOLVR = 3) can only use the Multiple Minimum Degree ordering method. The newer sparse direct solvers (LSOLVR = 4, 5, and 6) can use either MMD or Metis. Metis is a ordering method from University of Minnesota and is very effective for larger problems and for 3D solid problems. MMD is best for smaller problems, that is less than 100,000 rows in the assembled stiffness matrix.

Note that the values of LPRINT and ORDER also affect the eigensolution software. That is LPRINT and ORDER from tthis keyword card is applicable to eigensolution.

*CONTROL_IMPLICIT_STABILIZATION

Purpose: Define parameters for artificial stabilization during multi-step implicit springback analysis.

Card Format

	1	2	3	4	5	6	7	8
Variable	IAS	SCALE	TSTART	TEND				
Type	I	F	F	F				
Default	(see below)	1.0	(see below)	(see below)				

VARIABLE	DESCRIPTION
IAS	Artificial Stabilization flag EQ.1: active (Default for "springback" implicit analysis) EQ.2: inactive (Default for "standard" implicit analysis)
SCALE	Scale factor for artificial stabilization. Values greater than 1.0 cause less springback in the first few steps, while values less than 1.0 allow the part to springback more freely over the first few steps. For flexible parts with large springback a value of 0.001 may be required. EQ.-n: curve "n" defines 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. At this time, all stabilization has been removed from the simulation.

IAS The default for IAS depends on the analysis type in *CONTROL_IMPLICIT_GENERAL. For "springback" analysis, automatic time step control and artificial stabilization are activated 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 supress springback until very near the termination time. Small values may not stabilize the solution enough to allow equilibrium iterations to converge.

*CONTROL_NONLOCAL

Purpose: Allocate additional memory for *MAT_NONLOCAL option.

Card Format

	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 vaiue 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 Format

	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 Format

	1	2	3	4	5	6	7	8
Variable	IPRTF							
Type	I							
Default	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	Averaged accelerations from velocities in file "nodout" and the time history database file "d3thdt": EQ.0: no average (default), EQ.1: averaged between output intervals.
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

*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 Format

	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.
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_REMESHING

Purpose: Control the element size for three dimensional adaptivity for solids element. This commands control the size of the elements on the surface of the solid part.

Card Format

	1	2	3	4	5	6	7	8
Variable	RMIN	RMAX						
Type	F	F						
Default	none	none						

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.

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 Format

	1	2	3	4	5	6	7	8
Variable	LMF	JNTF	ORTHMD	PARTM	SPARCE			
Type	I	I	I	I	I			
Default	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 underdevelopment in version 960. 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.
LMF	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.

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 prevents this occurrence and LMF=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 multiplies save a substantial number of operations if the matrix is truly sparse. However, the overhead will slow the multiplies for densely populated matrices.

*CONTROL_SHELL

Purpose: Provide controls for computing shell response.

Card Format

	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

Optional Card

	1	2	3	4	5	6	7	8
Variable	ROTASCL	INTGRD	LAMSHT					
Type	F	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.
ESORT	Automatic sorting of triangular shell elements to treat degenerate quadrilateral shell elements as C0 triangular shells, see option THEORY below: EQ.0: no sorting required (default). EQ.1: full sorting,
IRNXX	Shell normal update option. This option affects the Hughes-Liu, Belytschko-Wong-Chiang, and the Belytschko-Tsay shell formultions. The latter is affected if and only if the warping stiffness option is active, i.e., BWC=1. IRNXX must be set to 2 to invoke the top or bottom surface as the reference surface for the Hughes-Liu shell elements. EQ.-2: unique nodal fibers which are incrementally updated based on the nodal rotation at the location of the fiber, EQ.-1: recompute fiber directions each cycle,

VARIABLE	DESCRIPTION
	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. This option affects all shell element formulations: EQ.0: no change. EQ.1: membrane straining causes thickness change. This option is very important in sheet metal forming or whenever membrane stretching is important.
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, 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 EQ.20: C^0 linear shell element with drilling stiffness. 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 weigthed 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.

VARIABLE	DESCRIPTION
PROJ	Projection method for warping stiffness in the Belytschko-Tsay shell and Belytschko-Wong-Chiang elements (see remarks below). EQ.0: drill projection, EQ.1: full projection.
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	For composite shells with material types *MAT_COMPOSITE_DAMAGE and *MAT_ENHANCED_COMPOSITE_DAMAGE. If this flag is set laminated shell theory is used. 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. EQ.0: do not update shear corrections, EQ.1: activate laminated shell theory.

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. 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 verion 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.

*CONTROL_SOLID

Purpose: Provide controls for solid element response.

Card Format

	1	2	3	4	5	6	7	8
Variable	ESORT							
Type	I							
Default	2							

VARIABLE	DESCRIPTION
ESORT	Automatic sorting of tetrahedron and pentahedron elements to treat degenerate hexahedron elements as tetrahedron and pentahedron solids, respective. See option THEORY below: EQ.0: no sorting required (default). EQ.1: full sorting,

*CONTROL_SOLUTION

Purpose: To specify the analysis solution procedure if thermal only or coupled thermal analysis is performed.

Card Format

	1	2	3	4	5	6	7	8
Variable	SOLN							
Type	I							
Default	0							

VARIABLE	DESCRIPTION
SOLN	Analysis solution procedure: 0: Structural analysis only, 1: Thermal analysis only, 2: Coupled structural thermal analysis. 4: Incompressible/low-Mach CFD analysis only, 5: Coupled incompressible fluid-structure interaction. (Not currently used.)

*CONTROL_SPH

Purpose: Provide controls for computing SPH particles

Card Format

	1	2	3	4	5	6	7	8
Variable	NCBS	BOXID	DT	DIM				
Type	I	I	F	I				
Default	1	none	1.e20	none				

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: 3 for 3D Problems 2 for 2D Problems -2 for 2D Axisymmetric When a value is not specified LS-DYNA determines the space dimension automatically by checking the use of 3D, 2D or 2D asisymmetric elements.

*CONTROL_STRUCTURED_{*OPTION*}

Options include:

TERM

Purpose: Write out a LS-DYNA structured input deck for Version 960. The name of this structured file is “dyna.str”. This input deck will not support all capabilities that are available in Version 960. 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 Theoretical Manual, Section 20.2, 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 Format

	1	2	3	4	5	6	7	8
Variable	ENDTIM	ENDCYC	DTMIN	ENDENG	ENDMAS			
Type	F	I	F	F	F			
Default	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-DYNA3D 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".

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 Format

	1	2	3	4	5	6	7	8
Variable	REFMAX	TOL	DCP					
Type	I	F	F					
Default	10	-	1.0 / 0.5					

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

*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 Format

	1	2	3	4	5	6	7	8
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.

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.
SOLVER	Thermal analysis solver type: EQ.1: actol : symmetric direct solver, EQ.2: dactol : nonsymmetric direct solver, EQ.3: dscg : diagonal scaled conjugate gradient iterative (default), EQ.4: iccg : incomplete choleski conjugate gradient iterative.
CGTOL	Convergence tolerance for solver types 3 and 4. (eq.0: default 1.e-04)
GPT	Number of Gauss points to be used in the solid elements: EQ.0: the default is set to 8, EQ.1: one point quadrature is used.
EQHEAT	Mechanical equivalent of heat (e.g., 1 J / N m). (eq.0: default set to 1.)
FWORK	Fraction of mechnical work converted into heat. (eq.0: default set to 1.)
SBC	Stefan Boltzmann constant. Value is used with enclosure radiation surfaces, see *BOUNDARY_RADIATION_....

Remark:

1. Use of a direct solver (e.g., solver=1) is usually less efficient than an iterative solver. Solver 1 should be tried if convergence problems occur with an iterative solver.

*CONTROL_THERMAL_TIMESTEP

Purpose: Set timestep controls for the thermal solution in a thermal only or coupled structural/ thermal analysis. Also *CONTROL_SOLUTION, *CONTROL_THERMAL_SOLVER needed.

Card Format

	1	2	3	4	5	6	7	8
Variable	TS	TIP	ITS	TMIN	TMAX	DTEMP	TSCP	
Type	I	F	F	F	F	F	F	
Default	0	0.5	none	-	-	1.0	0.5	

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 timestep.
TMAX	Maximum thermal time step: EQ.0.0: set to 100 * structural explicit timestep.
DTEMP	Maximum temperature change in each time step above which the thermal timestep 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.

*CONTROL_TIMESTEP

Purpose: Set structural time step size control using different options.

Card Format

	1	2	3	4	5	6	7	8
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

Card Format (This card is optional).

Card 2	1	2	3	4	5	6	7	8
Variable	DT2MSF							
Type	F							
Default	not used							

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).

VARIABLE	DESCRIPTION
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). 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: timestep size is based on the maximum eigenvalue. This option is okay for structural applications where the material sound speed changes slowly. The calculational cost to determine 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. 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. The DT2MS option below applies to all materials and element classes and may be preferred.
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”.
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.
ERODE	Erosion flag for solid and solid shell elements when DTMIN (see *CONTROL_TERMINATION)is reached. If this flag is not set the calculation will terminate: EQ.0: no, EQ.1: yes.

VARIABLE	DESCRIPTION
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.

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.

*DAMPING

The Keyword options in this section in alphabetical order are:

***DAMPING_GLOBAL**

***DAMPING_PART_MASS**

***DAMPING_PART_STIFFNESS**

***DAMPING_RELATIVE**

*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 Format

	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 node system damping: 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 (this option is bypassed if the load curve number defined above is non zero).
STX	Scale factor on global x translational damping forces.

VARIABLE	DESCRIPTION
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 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. Note that this damping applies to both translational and rotational degrees of freedom.

   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

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 Format

	1	2	3	4	5	6	7	8
Variable	PID	LCID	SF	FLAG				
Type	I	I	F	I				
Default	0	0	1.0	0				

Card Format (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	STR	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	Part ID, see *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.
STX	Scale factor on global x translational damping forces.

VARIABLE	DESCRIPTION
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

Purpose: Assign Rayleigh stiffness damping coefficient by part ID.

Card Format

	1	2	3	4	5	6	7	8
Variable	PID	COEF						
Type	I	F						
Default								

VARIABLE	DESCRIPTION
PID	Part ID, see *PART.
COEF	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.

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:

$$\phi_i^t C \phi_i = \phi_i^t (\alpha M + \beta K)\phi_i = \alpha + \beta\omega_i^2 = 2\omega_i \xi_i \delta_{ij}$$

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 significanly. 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.

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 acummulated 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 Format

	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.

*DAMPING

*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_BINARY_*OPTION

DATABASE_CROSS_SECTION_*OPTION

DATABASE_EXTENT_*OPTION

***DATABASE_FORMAT**

DATABASE_HISTORY_*OPTION

***DATABASE_NODAL_FORCE_GROUP**

***DATABASE_SPRING_FORWARD**

***DATABASE_SUPERPLASTIC_FORMING**

***DATABASE_TRACER**

The ordering of the database definition cards in the input file is competely 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
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.
DEFORC	Discrete elements.
ELOUT	Element data. See *DATABASE_HISTORY_*OPTION*.
GCEOUT	Geometric contact entities.
GLSTAT	Global data. Always obtained if SSSTAT file is activated.
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	Nodal point data. See *DATABASE_HISTORY_*OPTION*.
RBDOUT	Rigid body data. See Remark 2 below.
RCFORC	Resultant interface forces.
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 (spotwelds and rivets).
TPRINT	Thermal output from a coupled structural/thermal or thermal only analysis.
TRHIST	Tracer particle history information. See *DATABASE_TRACER.

Card Format

	1	2	3	4	5	6	7	8
Variable	DT							
Type	F							
Default	0.							

VARIABLE	DESCRIPTION
DT	Time interval between outputs. If DT is zero, no output is printed.

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

	I/O UNIT #	FILE NAME
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 (spotwelds/rivets)
Thermal output	i/o unit #73	TPRINT
Tracer particles	i/o unit #70	TRHIST

Output Components for ASCII Files

ABSTAT	BNDOUT	DEFORC
volume	x,y,z force	x,y,z force
pressure		
internal energy		
input mass flow rate		
output mass flow rate		
mass		
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

Translational Components	Rotational Components
x force	x moment
y force	y moment
z force	z moment

GLSTAT	JNTFORC	MATSUM
kinetic energy	x,y,z force	kinetic energy
internal energy	x,y,z moment	internal energy
total energy		hourglass energy
ratio		x,y,z momentum
stonewall-energy		x,y,z rigid body velocity
spring & damper energy		total kinetic energy
hourglass energy		total internal energy
damping energy		total hourglass energy
sliding interface energy		
external work		
x,y,z velocity		
time step		
element id controlling time step		

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		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			
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 rotational kinetic energy 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-POST. 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.

***DATABASE_BINARY**_*OPTION*

Options for binary output files with the default names given include:

D3DRLF	Dynamic relaxation database.
D3DUMP	Binary output restart files. Define output frequency in cycles.
D3MEAN	Averaging interval and statistics level for mean value database.
D3PART	Dt for partial output states See also *DATABASE_EXTENT_BINARY.
D3PLOT	Dt for complete output states. See also *DATABASE_EXTENT_BINARY.
D3THDT	Dt for time history data of element subsets. See *DATABASE_HISTORY.
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.
D3CRCK	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. 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 timestep.

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-POST. 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-POST. 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), 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. The default file size of 7000000 octal words may be much to small to hold one complete output state when models are very large, and an excessive number of files may be created. The limit of LS-DYNA to create files is 99 family members. Therefore, it is recommended that the file size be adjusted on the execution line with the X=scl (scl is a scale factor to enlarge the family member size). 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 Format

	1	2	3	4	5	6	7	8
Variable	DT/CYCL	LCDT	BEAM	NPLTC	PSETID	ISTATS	TSTART	IAVG
Type	F	I	I	I	I	I	F	I
Default	-	-	-	-	-	0	0.0	100
Remarks						1		

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.
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, 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.

ISTATS	Set the level of statistics to collect. This applies to D3MEAN only, and is also restricted to the incompressible CFD solver variables. EQ.0: don't collect any statistics (default), EQ.1: generate mean quantities, EQ.2: generate second moments in addition to the mean quantities, EQ.3: generate higher-order moments in addition to all other moments.
TSTART	Set the simulation time at which collection of the time-averaged statistics will begin (D3MEAN only). TSTART=0.0 is the default.
IAVG	Set the interval to write out the time-averaged statistics (D3MEAN only). The time-averaged statistics are re-initialized and collection of new statistics starts after the time-averaged data is written to the database. EQ.0: IAVG=100 (default).

Remarks:

1. When positive, this option creates the D3MEAN binary database containing the mean field values and correlations according to the level of statistics requested. Note that the time-averaged statistics are only available for analyses that solve the time-dependent Navier-Stokes equations.

   For ISTATS=1, the time averages of the following variables are placed in the database:

   X-velocity, Y-velocity, Z-velocity, Temperature, Pressure, X-vorticity, Y-vorticity, Z-vorticity, Stream Function, Density, Species-1 Concentration, ... , Species-10 Concentration.

   For ISTATS=2, the database includes the time average quantities specified with ISTATS=1, as well as X-velocity, Y-velocity, and Z-velocity correlations with the following variables:

   X-velocity, Y-velocity, Z-velocity, Temperature, Pressure, Species-1 Concentration, ... , Species-10 Concentration.

   For ISTATS=3, the database includes the time average quantities specified with ISTATS=1 and ISTATS=2, as well as time average of the following variables:

   u_x^3, u_y^3, u_z^3, u_x^4, u_y^4, and u_z^4.

   LS-POST derives the following additional quantities for each level of statistics:

   For ISTATS=1, velocity magnitude, enstrophy, and helicity are added.

   For ISTATS=2, turbulent kinetic energy, Reynolds Stresses, and fluctuations of other velocity correlation quantities are added.

   For ISTATS=3, velocity skewness and velocity flatness are added.

   For further details on these mean statistical quantities, see Chapter 8 (Flow Statistics) in LS-DYNA's Incompressible Flow Solver User's Manual.

DATABASE_CROSS_SECTION_*OPTION

Options include:

PLANE

SET

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 9.1. If the SETS option is used, just one card is needed. In this latter case the forces in the elements belonging to the set are summed up to form the section forces.

Format (1 of 2) for the PLANE option

	1	2	3	4	5	6	7	8
Variable	PSID	XCT	YCT	ZCT	XCH	YCH	ZCH	
Type	I	F	F	F	F	F	F	
Default	0	0.	0.	0.	0.	0.	0.	

Format (2 of 2) for the PLANE option

	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	

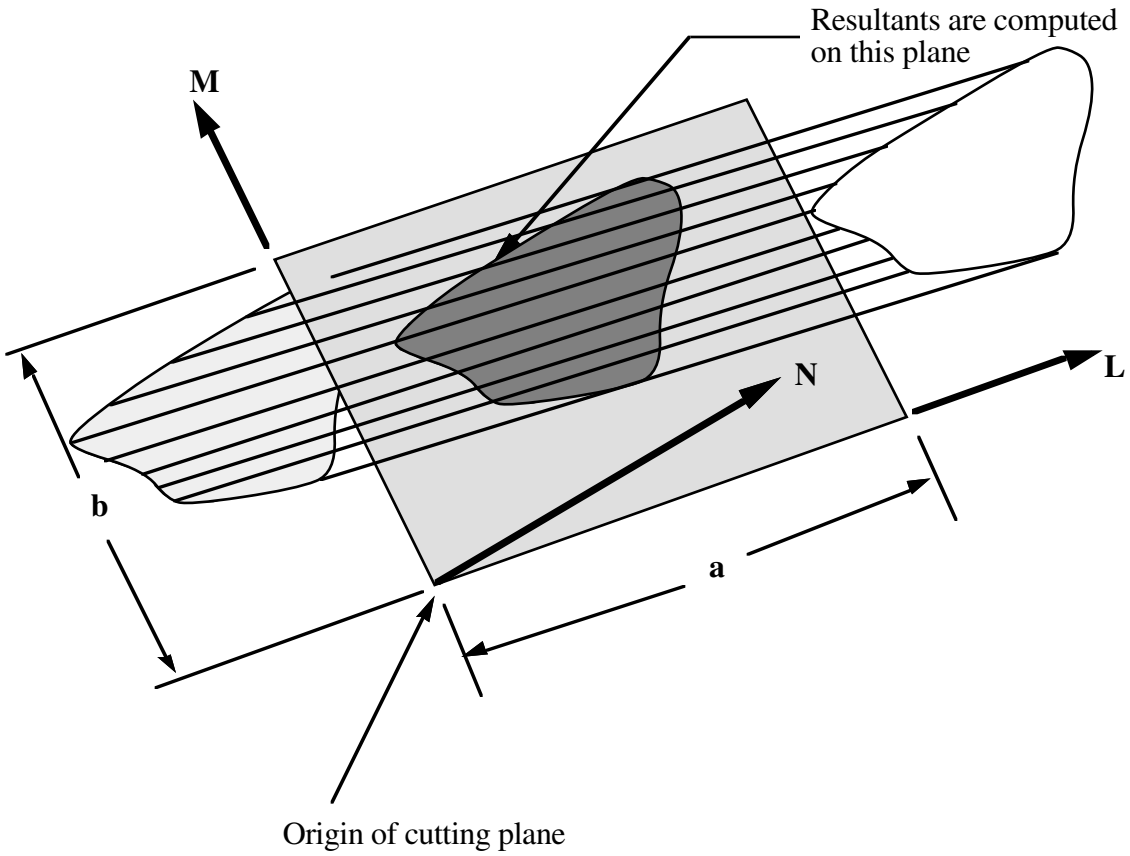


Figure 9.1. Definition of cutting plane for automatic definition of interface for cross-sectional forces. The automatic definition does not check for springs and dampers in the section. For best results the cutting plane should cleanly pass through the middle of the elements, distributing them equally on either side.

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.

Format (1 of 1) for the SET option

	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
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 9.1.
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**.
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**.
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.

VARIABLE	DESCRIPTION
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) or accelerometer ID (see *ELEMENT_ SEATBELT_ACCELEROMETER). The force resultants are output in the updated local system of the rigid body or accelerometer.
ITYPE	Flag for local system type: EQ. 0: rigid body, EQ. 1: accelerometer.

DATABASE_EXTENT_*OPTION

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 Format

	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 9.1, VTYPE.EQ.1: Table 9.2, VTYPE.EQ.2: not supported, VTYPE.EQ.3: Table 9.3, 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 connectivities 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 9.1. Nodal Quantities

Component ID	Quantity
1	x, y, z-displacements
2	x, y, z-velocities
3	x, y, z-accelerations

Table 9.2. 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 9.3. 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
17	outer surface z-stress
18	outer surface xy-stress
19	outer surface yz-stress
20	outer surface zx-stress

Table 9.3. Shell and Thick Shell Element Quantities (cont.).

Component ID	Quantity
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-nyy (4-node shell)
29	normal resultant-nxy (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	midsuface 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 9.4. 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 Format

Card 1

Variable	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	CMPFLG	IEVERP	BEAMIP	DCOMP	SHGE	STSSZ	N3THDT	
Type	I	I	I	I	I	I	I	
Default	0	0	0	0	0	0	2	
Remarks			2					

VARIABLE	DESCRIPTION
NEIPH	Number of additional integration point history variables written to the LS-TAURUS 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 LS-TAURUS database for both shell and thick shell elements for each integration point, see NEIPH above.

VARIABLE	DESCRIPTION
MAXINT	Number of shell integration points written to the LS-DYNA database, see also *INTEGRATION_SHELL. If the default value of 3 is used then results are output for the outrtmost (top) and innermost (bottom) integration points together with results for the neutral axis. If MAXINT is set to 3 and the 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.
STRFLG	Set to 1 to dump strain tensors for solid, shell and thick shell elements for plotting by LS-POST 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.
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 internal energy and thickness in the LS-DYNA database: EQ.1: include (default), EQ.2: exclude.
CMPFLG	Orthotropic and anisotropic material stress output in local coordinate system for shells and thick shells. Currently, this option does not apply to solid elements with the exception of material, MAT_COMPOSITE_DAMAGE. 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 100 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 data compression, EQ.2: on.

VARIABLE	DESCRIPTION
SHGE	Output shell hourglass energy: 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, 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.

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, the added mass is output. Also, see the control card *CONTROL_TIMESTEP.

For the SSSTAT option the following card(s) apply:

Define as many cards as necessary.

Card Format (Define one part set ID for each subsystem. Use as many cards as necessary.)

	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 Format

	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_HISTORY_*OPTION

Options include:

BEAM
BEAM_SET
NODE
NODE_LOCAL
NODE_SET
NODE_SET_LOCAL
SHELL
SHELL_SET
SOLID
SOLID_SET
SPH
SPH_SET
TSHELL
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*.

Card Format for all options except NODE_LOCAL and NODE_SET_LOCAL

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, SHELL/SHELL_SET, SOLID/SOLID_SET, or TSHELL/ TSHELL_SET. The contents of the files are given in Table 9.1 for nodes, Table 9.2 for solid elements, Table 9.3 for shells and thick shells, and Table 9.4 for beam elements. In the binary file, D3THDT, the contents may be extended or reduced with the *DATABASE_EXTENT_BINARY definition.

Card Format options NODE_LOCAL and NODE_SET_LOCAL

Cards 1,2,...	1	2	3	4	5	6	7	8
Variable	ID	CID	REF					
Type	I	I	I					

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.

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_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 Format

	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 absense of body forces, interior nodes would always yield a null force resultant vector. In general this option would be used for surface nodes.

*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 Format

Cards 1	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, EQ.2: output element nodal force vector for materials, subset for NIKE3D interface file.

*DATABASE_SUPERPLASTIC_FORMING

Purpose: Specify the output intervals to the superplastic forming output files. The option *LOAD_SUPERPLASTIC_FORMING must be active.

Card Format

Cards 1	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 pressure file may be plotted in Phase 3 of LS-TAURUS using the SUPERPL option.

*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.

Card Format

	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_BOX**

***DEFINE_BOX_ADAPTIVE**

***DEFINE_BOX_COARSEN**

***DEFINE_BOX_DRAWBEAD**

***DEFINE_COORDINATE_NODES**

***DEFINE_COORDINATE_SYSTEM**

***DEFINE_COORDINATE_VECTOR**

***DEFINE_CURVE**

***DEFINE_CURVE_FEEDBACK**

***DEFINE_CURVE_SMOOTH**

***DEFINE_CURVE_TRIM**

***DEFINE_SD_ORIENTATION**

***DEFINE_TABLE**

***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_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 Format

	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 Format

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.

VARIABLE	DESCRIPTION
PID	Part ID. If zero, all active element 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 Format

	1	2	3	4				
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 outside box can not be coarsened EQ.1: elements inside box can not 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-shaped volume around a drawbead. The box will contain the drawbead nodes and elements between the bead and the outer edge of the blank. Elements directly under the bead are also included.

Card Format

	1	2	3	4				
Variable	BOXID	PID	NSID	IDIR				
Type	I	F	F	F				
Default	0	0.0	0.0	0.0				
Remarks								

VARIABLE	DESCRIPTION
BOXID	Box ID. Define unique numbers.
PID	Part ID of blank.
NSID	Node set ID defining nodes that lie along the drawbead.
IDIR	Direction of tooling movement: EQ.1: tooling moves in x-direction, EQ.2: tooling moves in y-direction, EQ.3: tooling moves in z-direction.

*DEFINE_COORDINATE_NODES

Purpose: Define a local coordinate system with three node numbers. The local cartesian coordinate system is defined in the following steps. The z-axis is computed from the cross product of x and $\bar{y}$, (see Figure 10.1), $z = x \times \bar{y}$, then the y-axis is computed via $y = z \times x$.

Card Format

	1	2	3	4	5	6	7	8
Variable	CID	N1	N2	N3	FLAG			
Type	I	I	I	I	I			
Default	0	0	0	0	0			

VARIABLE	DESCRIPTION
CID	Coordinate system ID. A unique number has to be defined.
N1	Number of node located at local origin.
N2	Number of node located along local x-axis.
N3	Number of node located in local x-y plane.
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

Remark:

1. The nodes N1, N2, and N3 must be separated by a reasonable distance and not colinear to avoid numerical inaccuracies.

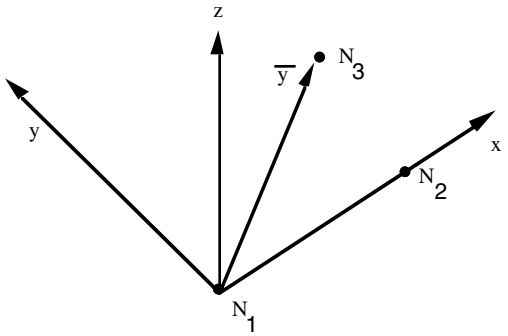


Figure 10.1. Definition of local coordinate system using three nodes.

*DEFINE_COORDINATE_SYSTEM

Purpose: Define a local coordinate system with three points. The same procedure as described in Figure 10.1, 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.

	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.

	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
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

Remark:

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 10.2. The vector cross product, $xy \times x = z$, determines the z-axis. The y-axis is then given by $y = z \times x$.

Card Format

	1	2	3	4	5	6	7	8
Variable	CID	XX	YX	ZX	XV	YV	ZV	
Type	I	F	F	F	F	F	F	
Default	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

Remark:

1. These vectors should be separated by a reasonable included angle to avoid numerical inaccuracies.

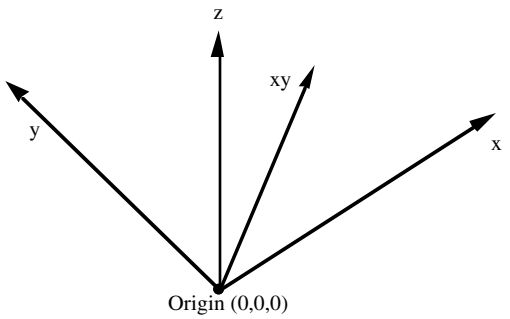


Figure 10.2. Definition of the coordinate system with two vectors.

*DEFINE_CURVE

Purpose: Define a curve [for example, load (ordinate value) versus time (abcissa value)], often referred to as a load curve.

Card Format

	1	2	3	4	5	6	7	8
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.)

	1	2	3	4	5	6	7	8
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-DYNA3D allows load curve ID's and table ID's to be used interchangeably. A unique number has to be defined. **Note: <u>The magnitude of LCID is restricted to 5 significant digits. This limitation will be removed in a future release of LS-DYNA3D</u>.**
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
SFA	Scale factor for abcissa 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 abcissa 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 abcissa values do not increase monotonically. Generally, DATTYP=0 for time dependent curves, force versus displacement curves, and stress strain curves.
A1, A2,...	Abcissa 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.:

$$Abcissa\ value = SFA \cdot (Defined\ value + OFFA)$$
$$Ordinate\ value = SFO \cdot (Defined\ value + OFFO)$$

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 abcissa simply shifts the abcissa values without creating additional points.

4. Load curves are not extrapolated by LS-DYNA for applied loads such as pressures, concentrated forces, displacement boundary condtions, 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 abcissa 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_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 Format

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 10.3. If the product of *FSL* and the ordinate value of the maximum principal strain is exceeded the scale factor for flow, *SF*, is active.
FSL	If the strain ratio, $\varepsilon_{major_{workpiece}} / \varepsilon_{major_{fld}}$, is exceeded the scale factor for flow, *SF*, is active.

VARIABLE	DESCRIPTION
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).
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.

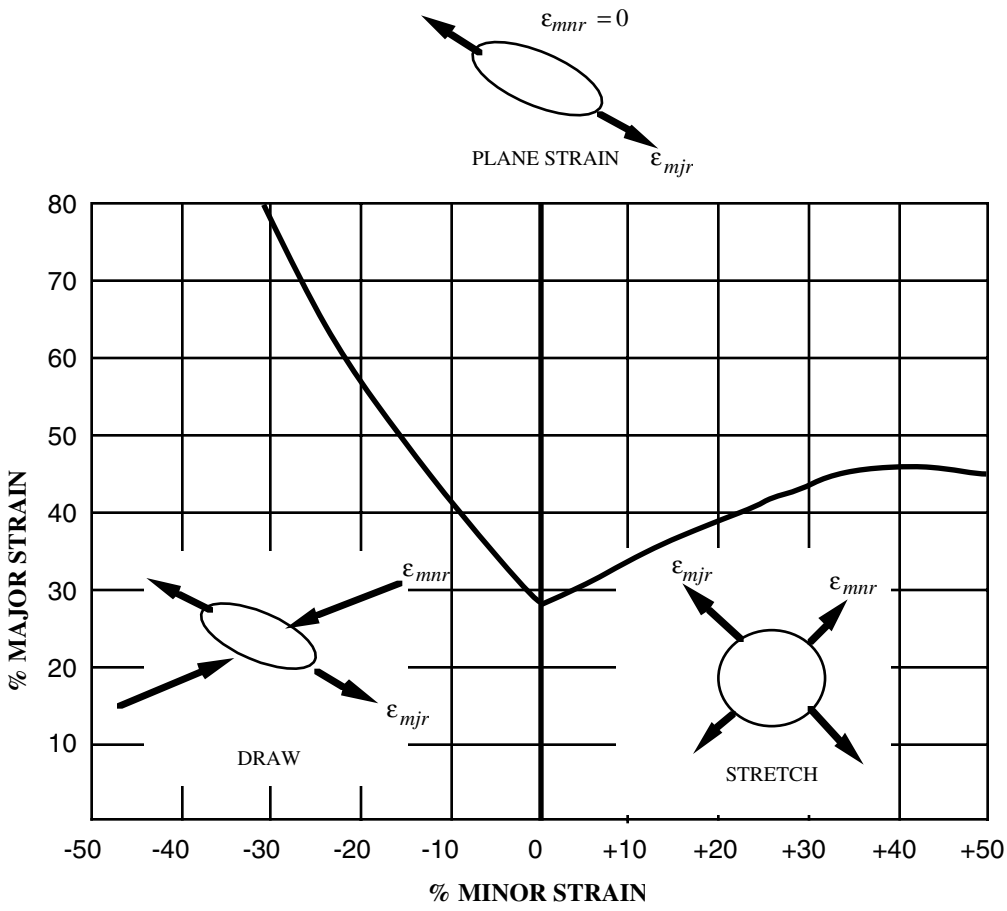


Figure 10.3. Flow limit diagram.

*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 Format

	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:

See Figure 10.4.

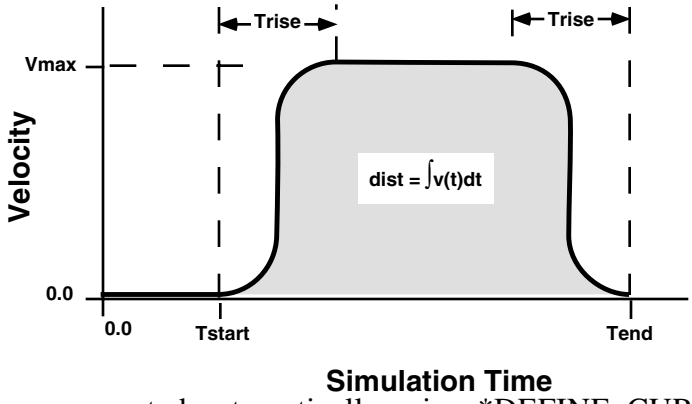


Figure 10.4. Smooth curve created automatically using *DEFINE_CURVE_SMOOTH. This shape is commonly used to control velocity of tools in metal forming applications as shown in the above graph, but can be used for other applications in place of any standard load curve.

*DEFINE_CURVE_TRIM

Purpose: Define a curve for trimming. Also, see *INTERFACE_SPRINGBACK.

Card Format

	1	2	3	4	5	6	7	8
Variable	TCID	TCTYPE	TFLG	TDIR	TCTOL			
Type	I	I	I	I	F			
Default	none	none	none	none	0.25			
Remarks	1,2,3			figure 10.5	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.

	1	2	3	4	5	6	7	8
Variable	CX		CY					
Type	F		F					
Default	0.0		0.0					

Card 2 defined if and only if TCTYPE=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 10.5). 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 10.6). LT.0: "simple" trimming, producing jagged edge mesh
C X	x-coordinate of trim curve Defined if and only if TCTYPE=1.
CY	y-coordinate of trim curve Defined if and only if TCTYPE=1.
FILENAME	Name of IGES database containing trim curve(s). Defined 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 proceduce 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.

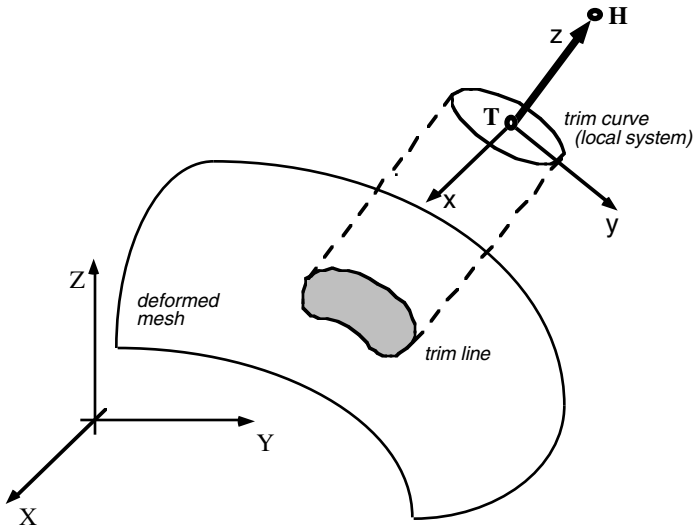
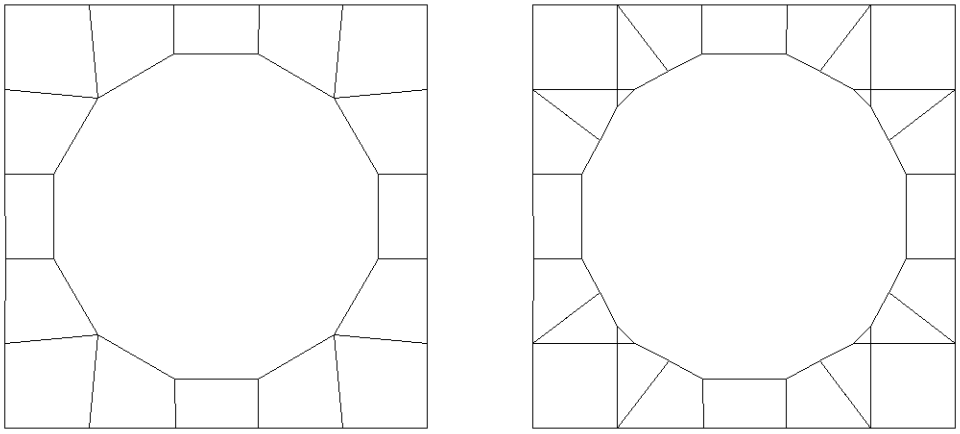


Figure 10.5. Trimming Orientation Vector. The tail (**T**) and head (**H**) points define a local coordinate system (x,y,z). The local x-direction is constructed in the Xz plane. Trim curve data is input in the x-y plane, and projected in the z-direction onto the deformed mesh to obtain the trim line.



tol = 0.25 (default) *tol = 0.01*

Figure 10.6 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_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 optiona 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 Format

	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.

VARIABLE	DESCRIPTION
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_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. There are no rules for defining the n curves, i.e., each curve may have a different origin, spacing, and number of points in their definition. (Load curve ID's defined for the table may be referenced elsewhere in the input.) 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 Format

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

Card 2, 3, 4, etc. Put one point per card (E20.0). Input is terminated when a "*DEFINE_CURVE" card is found.

	1	2	3	4	5	6	7	8
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-DYNA3D allows load curve ID's and table ID's to be used interchangeably.
VALUE	Load curve will be defined corresponding to this value, e.g., this value could be a strain rate, see purpose above.

Remark:

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.

*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.

Card Format 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	1	2	3	4	5	6	7	8
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 below for the three available options..

FORMAT (A10,7F10.0)		
OPTION	ENTITIES + ATTRIBUTES	FUNCTION
SCALE	a1, a2, a3	Scale the 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 a4, a5, and a6.
TRANSL	a1, a2, a3	Translate the x, y, and z coordinates of a point by a1, a2, and a3, respectively.

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 *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)

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
      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 Format

	1	2	3	4	5	6	7	8
Variable	VID	XT	YT	ZT	XH	YH	ZH	
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
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

Remark:

1. The coordinates should differ by a certain margin to avoid numerical inaccuracies.

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$  *DEFINE_BOX
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Define box number eight which encloses a volume defined by two corner
$  points: (-20.0, -39.0, 0.0) and (20.0, 39.0, 51.0).  As an example, this
$  box can be used as an input for the *INITIAL_VELOCITY keyword in which
$  all nodes within this box are given a specific initial velocity.
$
*DEFINE_BOX
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$    boxid       xmm       xmx       ymn       ymx       zmn       zmx
         8     -20.0      20.0     -39.0      39.0       0.0      51.0
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$  *DEFINE_COORDINATE_NODES
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Define local coordinate system number 5 using three nodes: 10, 11 and 20.
$  Nodes 10 and 11 define the local x-direction.  Nodes 10 and 20 define
$  the local x-y plane.
$
$  For example, this coordinate system (or any coordinate system defined using
$  a *DEFINE_COORDINATE_option keyword) can be used to define the local
$  coordinate system of a joint, which is required in order to define joint
$  stiffness using the *CONSTRAINED_JOINT_STIFFNESS_GENERALIZED keyword.
$
*DEFINE_COORDINATE_NODES
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$      cid        n1        n2        n3
         5        10        11        20
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$   *DEFINE_COORDINATE_SYSTEM
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Define local coordinate system number 3 using three points.  The origin of
$  local coordinate system is at (35.0, 0.0, 0.0).  The x-direction is defined
$  from the local origin to (35.0, 5.0, 0.0).  The x-y plane is defined using
$  the vector from the local origin to (20.0, 0.0, 20.0) along with the local
$  x-direction definition.
$
*DEFINE_COORDINATE_SYSTEM
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$     cid        Xo        Yo        Zo        Xl        Yl        Zl
        3      35.0       0.0       0.0      35.0       5.0       0.0
$
$      Xp        Yp        Zp
     20.0       0.0      20.0
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$   *DEFINE_COORDINATE_VECTOR
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Define local coordinate system number 4 using two vectors.
$    Vector 1 is defined from (0.0, 0.0, 0.0) to (1.0, 1.0, 0.0)
$    Vector 2 is defined from (0.0, 0.0, 0.0) to (1.0, 1.0, 1.0)
$  See the corresponding keyword command for a description.
$
*DEFINE_COORDINATE_VECTOR
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$     cid        Xx        Yx        Zx        Xv        Yv        Zv
        4       1.0       1.0       0.0       1.0       1.0       1.0
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$   *DEFINE_CURVE
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Define curve number 517.  This particular curve is used to define the
$  force-deflection properties of a spring defined by a *MAT_SPRING_INELASTIC
$  keyword.  The abscissa value is offset 25.0 as a means of modeling a gap
$  at the front of the spring.  This type of spring would be a compression
$  only spring.
$
*DEFINE_CURVE
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$     lcid      sidr      scla      sclo      offa      offo
       517                                    25.0
$
$          abscissa            ordinate
                0.0                 0.0
               80.0                58.0
               95.0                35.0
              150.0                44.5
              350.0                45.5
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$   *DEFINE_SD_ORIENTATION
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  A discrete spring is defined with two nodes in 3-D space.  However, it is
$  desired to have the force of that spring to act only in the z-direction.
$  The following definition makes this happen.  Additionally, vid = 7
$  must be specified in the *ELEMENT_DISCRETE keyword for this spring.
$
*DEFINE_SD_ORIENTATION
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$      vid       iop        xt        yt        zt      nid1      nid2
         7         0       0.0       0.0       1.0
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$   *DEFINE_VECTOR
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Define vector number 5 from (0,0,0) to (0,1,1).  As an example, this vector
$  can be used to define the direction of the prescribed velocity of a node
$  using the *BOUNDARY_PRESCRIBED_MOTION_NODE keyword.
$
*DEFINE_VECTOR
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$      vid        xt        yt        zt        xh        yh        zh
         3       0.0       0.0       0.0       0.0       1.0       1.0
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

*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

*DEFORMABLE_TO_RIGID

Purpose: Define materials to be switched to rigid at the start of the calculation.

Card Format

	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 Format

Card 1

	1	2	3	4	5	6	7	8
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

	1	2	3	4	5	6	7	8
Variable	NRBF	NCSF	RWF	DTMAX	D2R	R2D		
Type	I	I	I	F	I	I		
Default	0	0	0	0.	0	0		
Remark	4	4	4					

*DEFORMABLE_TO_RIGID

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.
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/spotweld 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.

VARIABLE	DESCRIPTION
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.

Remarks:

1. Only surface to surface and node to surface contacts can be used to activate an automatic part switch.

2. Contact surface and 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.

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 consistant 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).

4. If the delete switch is activated, ALL corresponding constraints are deactivated regardless of their relationshiop 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         2                                       3        20        -1
$     nrbf      ncsf       rwf     dtmax       D2R       R2D
                                                      1
```

*DEFORMABLE_TO_RIGID

Define D2R cards below:

Card Format

	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 Format

	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 Format

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

Card 2	1	2	3	4	5	6	7	8
Variable	XC	YC	ZC	TM				
Type	F	F	F	F				

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	0.0	none	0.0	none		

*DEFORMABLE_TO_RIGID

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}

*ELEMENT

The element cards in this section are defined in alphabetical order:

***ELEMENT_BEAM_**{*OPTION*}

***ELEMENT_DIRECT_MATRIX_INPUT**

***ELEMENT_DISCRETE**

***ELEMENT_INERTIA**

***ELEMENT_MASS**

***ELEMENT_SEATBELT**

***ELEMENT_SEATBELT_ACCELEROMETER**

***ELEMENT_SEATBELT_PRETENSIONER**

***ELEMENT_SEATBELT_RETRACTOR**

***ELEMENT_SEATBELT_SENSOR**

***ELEMENT_SEATBELT_SLIPRING**

***ELEMENT_SHELL_**{*OPTION*}

***ELEMENT_SOLID_**{*OPTION*}

***ELEMENT_SPH**

***ELEMENT_TRIM**

***ELEMENT_TSHELL**

The ordering of the element cards in the input file is competely arbitrary. An arbitrary number of element blocks can be defined preceeded by a keyword control card.

***ELEMENT_BEAM_{*OPTION*}**

Available options include:

<BLANK>

THICKNESS

PID

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 option is provided for the user to override the *SECTION_BEAM data which is taken as the default if the THICKNESS option is not used. End release conditions are imposed used constraint equations, and caution must be used with this option as discussed in remark 2 below.

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 segments. 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 spotweld elements only one node is needed to define the weld, and two nodes are optional.

Card Format (10I8)

	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)

	1	2	3	4	5	6	7	8	9	10
Variable	PARM1		PARM2		PARM3		PARM4		PARM5	
Type	F		F		F		F		F	
Remarks	4		5		5		5		6	

Optional Card (Required if PID is specified after the keyword)

	1	2	3	4	5	6	7	8	9	10
Variable	PID1	PID2								
Type	I	I								
Default	none	none								
Remarks										

VARIABLE	DESCRIPTION
EID	Element ID. A unique number has to be specified.
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 nonconfliciting nodal point ID. Nodes N1 and N2 are automatically positioned for the spot weld beam element.
N3	Nodal point 3. The third node, N3, is optional for beam types 3, 6, 7, 8, and 9 if the latter, type 9, has a non-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.

VARIABLE	DESCRIPTION
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 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 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: not used 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: volume 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

VARIABLE	DESCRIPTION
PARM3	Based on beam type: Type.EQ.1: beam thickness, t direction at node 1 Type.EQ.2: I_{tt} Type.EQ.3: not used 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 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: not used
PID1	Optional part ID for spot weld element type 9.
PID2	Optional part ID for spot weld element type 9.

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 12.1.

2. This option applies to all three-dimensional beam elements. The released degrees-of-freedom can be either global, the default, or local relative to the local beam coordinate system, see Figure 12.1. 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 infinitestimal.

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.

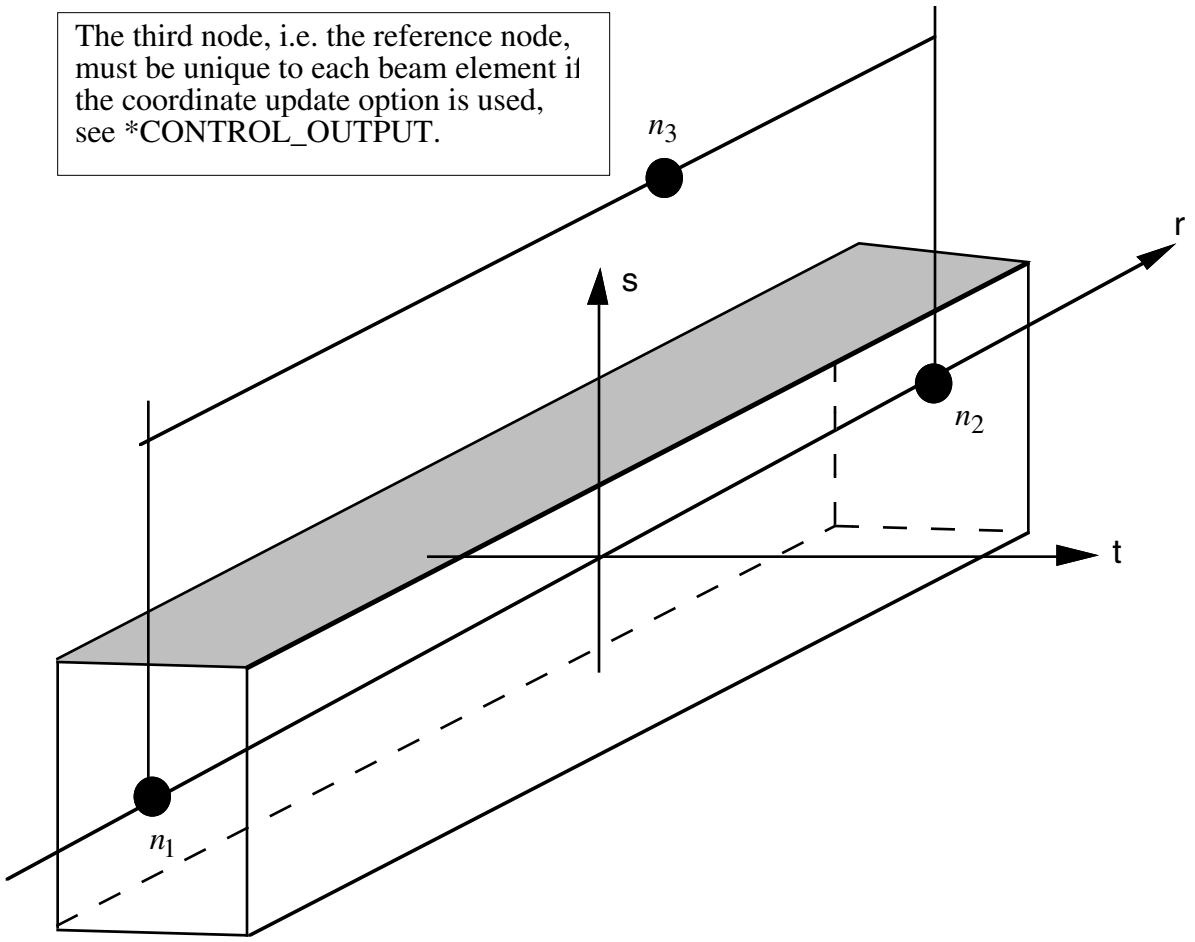


Figure 12.1. LS-DYNA beam elements. Node n_3 determines the initial orientation of the cross section.

*ELEMENT_DIRECT_MATRIX_INPUT

Purpose: Define a an element consisting of mass, damping, and stiffness matrices in a specified file which follows the formats used in the direct matrix input, DMIG, of NASTRAN. Currently, one file format is supported corresponding to the type 6 symmetric matrix in real double-precision. The damping matrix is optional. The following three cards are required for each super element. Multiple super elements can be contained in the same file, or each superelement may be contained in a separate file. The mass matrix must contain the same number of degrees-of-freedom as the stiffness matrix, and in the explicit integration scheme for which this element is implemented, the mass matrix must also be positive definite. This element is assumed to have an arbitrary number of degrees-of-freedom and the no assumptions are made about the sparse matrix structure of the matricies that comprise this element. The degrees-of-freedom for this element may consist of generalized coordinates as well as nodal point quantities.

Card Format (I8)

Card 1	1	2	3	4	5	6	7	8
Variable	EID							
Type	I							

Card Format (A80)

Card 2

Variable	FILENAME
Type	C

Card Format (3A8)

Card 3	1	2	3	4	5	6	7	8
Variable	MASS	DAMP	STIF					
Type	C	C	C					

VARIABLE	DESCRIPTION
EID	Super element ID.
FILENAME	Path and name of a file which containes the input matrices for this element.
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.
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.

*ELEMENT_DISCRETE

Purpose: Define a discrete (spring or damper) element between two nodes or a node and ground. 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 be developed and extended.

Note: These 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)

	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	

VARIABLE	DESCRIPTION
EID	Element ID. A unique number has to be used.
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.
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.

***ELEMENT_INERTIA**

Purpose: Define a lumped inertia element assigned to a nodal point.

Card Format (10I8)

	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 (8I10)

	1	2	3	4	5	6	7	8
Variable	IXX	IXY	IXZ	IYY	IYZ	IZZ		
Type	F	F	F	F	F	F		
Default	0.0	0.0	0.0	0.0	0.0	0.0		
Remarks		2	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.

VARIABLE	DESCRIPTION
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.

Remarks:

1. The coordinate system cannot be defined using the option *DEFINE_COORDINATE_NODE for this element.

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**

Purpose: Define a lumped mass element assigned to a nodal point .

Card Format (2I8,E16.0)

	1	2	3	4	5	6	7	8	9	10
Variable	EID	NID	MASS							
Type	I	I	F							
Default	none	none	0.							
Remarks										

VARIABLE	DESCRIPTION
EID	Element ID. A unique number must be used.
NID	Node ID. Node to which the mass is assigned.
MASS	Mass value

*ELEMENT_SEATBELT

Purpose: Define a seat belt element.

Card Format (5I8,E16.0)

	1	2	3	4	5	6	7	8	9	10
Variable	EID	PID	N_1	N_2	SBRID	SLEN				
Type	I	I	I	I	I	F				
Default	none	none	none	none	none	0.0				
Remarks										

VARIABLE	DESCRIPTION
EID	Element ID. A unique number has to be used.
PID	Part ID
N1	Node 1 ID
N2	Node 2 ID
SBRID	Retractor ID, see *ELEMENT_SEATBELT_RETRACTOR.
SLEN	Initial slack length

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.

*ELEMENT_SEATBELT_ACCELEROMETER

Purpose: Define seat belt accelerometer. The accelerometer is fixed to a rigid body containing the three nodes defined below.

Card Format

	1	2	3	4	5	6	7	8
Variable	SBACID	NID1	NID2	NID3	IGRAV			
Type	I	I	I	I	I			
Default	0	0	0	0	0			
Remarks								

VARIABLE	DESCRIPTION
SBACID	Accelerometer ID. A unique number has to be used.
NID1	Node 1 ID
NID2	Node 2 ID
NID3	Node 3 ID
IGRAV	Gravitational accelerations due to body force loads. EQ.0: included in acceleration output EQ.1: removed from acceleration output

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 ($\mathbf{z} = \mathbf{x} \times \mathbf{a}$), where **a** is from node 1 to node 3),
- local $\mathbf{y} = \mathbf{z} \times \mathbf{x}$.

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 Format

	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					

Second Card

	1	2	3	4	5	6	7	8
Variable	SBRID	TIME	PTLCID					
Type	I	F	I					
Default	0	0.0	0					
Remarks								

VARIABLE	DESCRIPTION
SBPRID	Pretensioner ID. A unique number has to be used.
SBPRTY	Pretensioner type: EQ.1: pyrotechnic retractor, EQ.2: pre-loaded spring becomes active, EQ.3: lock spring removed.

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) 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).

Remarks:

1. At least one sensor should be defined.

   Pretensioners allow modeling of three types of active devices which tighten the belt during the initial stages of a crash. The first type represents 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. The remaining types 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.

   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.

*ELEMENT_SEATBELT_RETRACTOR

Purpose: Define seat belt retractor.

Card Format

	1	2	3	4	5	6	7	8
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				

Second Card

	1	2	3	4	5	6	7	8
Variable	TDEL	PULL	LLCID	ULCID	LFED			
Type	F	F	I	I	F			
Default	0.0	0.0	0	0	0.0			
Remarks			3	4				

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

VARIABLE	DESCRIPTION
SID2	Sensor ID 2
SID3	Sensor ID 3
SID4	Sensor ID 4
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 12.3.
ULCID	Load curve for unloading (Pull-out, Force), see Figure 12.3.
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. At least one sensor should be defined.

3. 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}.

4. 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 × 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 11.2, 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 × 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 12.3. The final section can be very steep (but must have a finite slope).

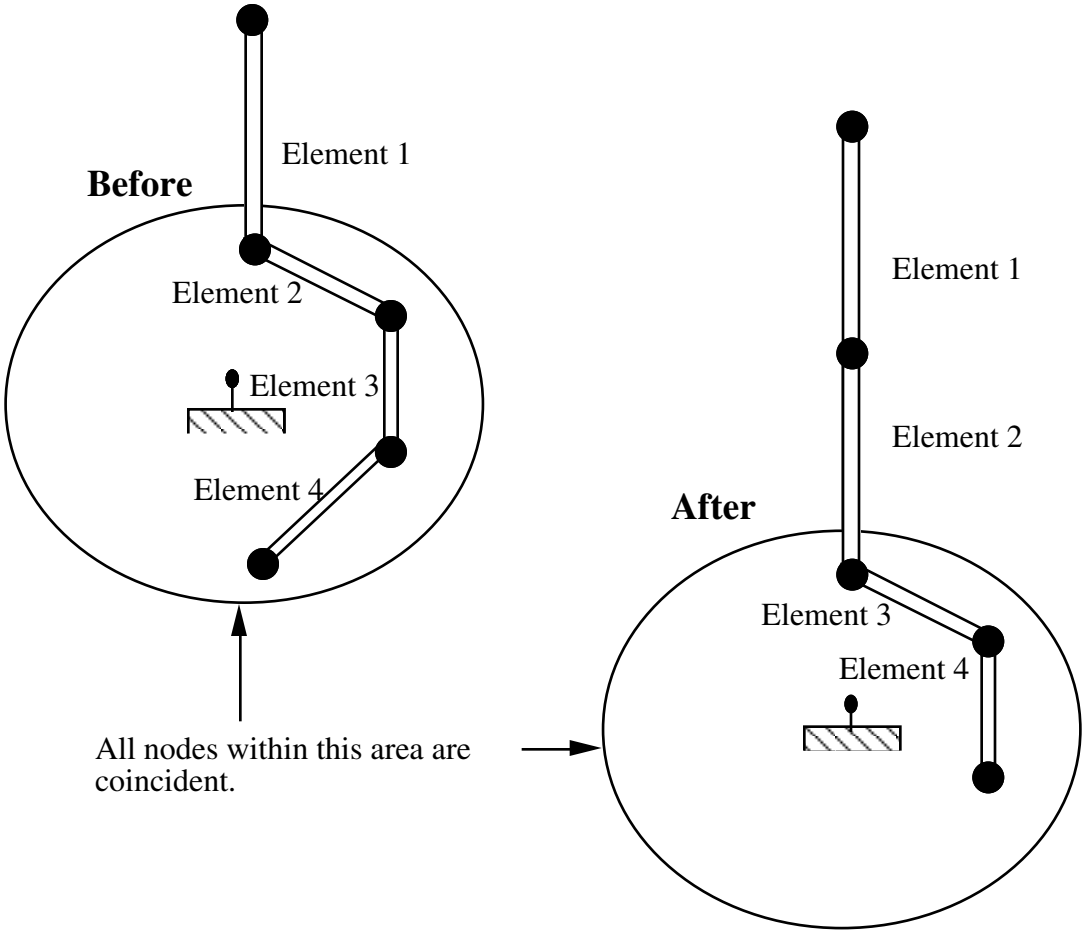


Figure 12.2. Elements in a retractor.

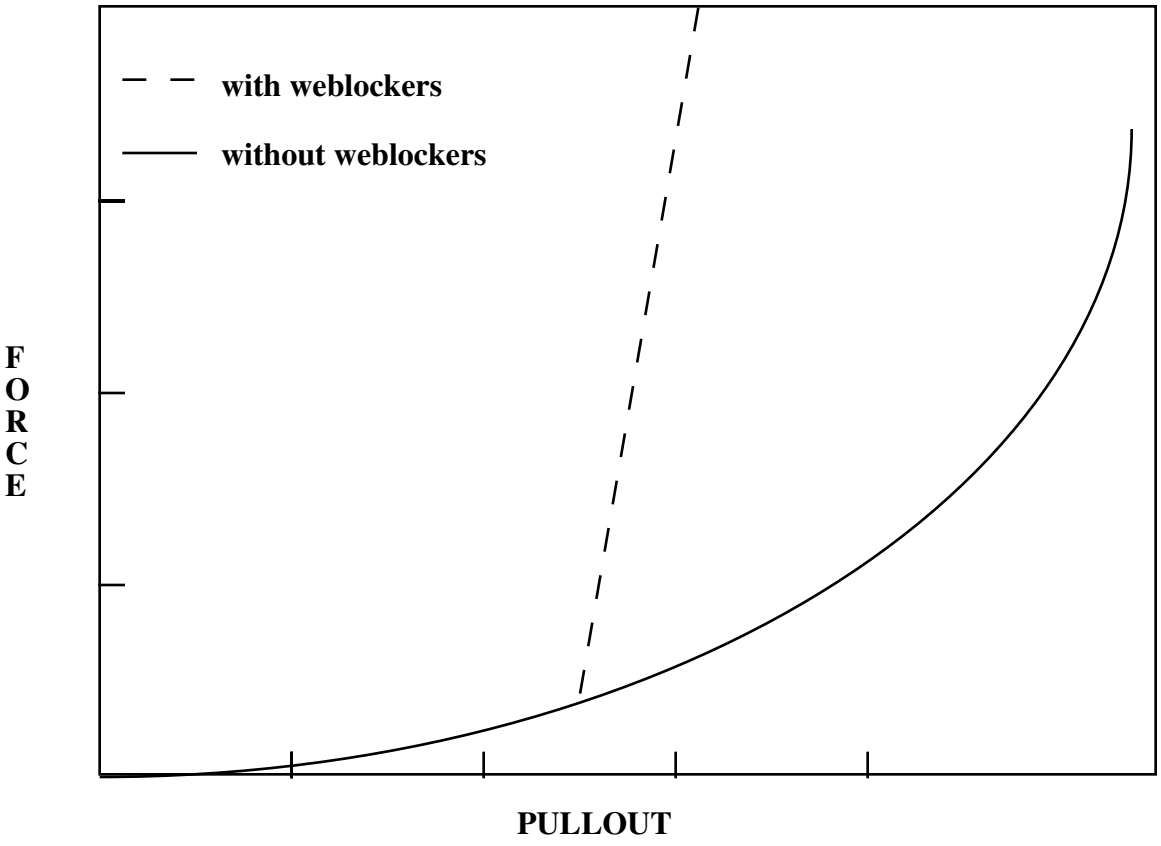


Figure 12.3. Retractor force pull characteristics.

*ELEMENT_SEATBELT_SENSOR

Purpose: Define seat belt sensor. Four types are possible, see explanation below.

Card Format

	1	2	3	4	5	6	7	8
Variable	SBSID	SBSTYP	SBSFL					
Type	I	I	I					
Default	0	0	0					
Remarks								

Second Card if SBSTYP=1

	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							

Second Card if SBSTYP=2

	1	2	3	4	5	6	7	8
Variable	SBRID	PULRAT	PULTIM					
Type	I	F	F					
Default	0	0.0	0.0					
Remarks								

Second Card if SBSTYP=3

	1	2	3	4	5	6	7	8
Variable	TIME							
Type	F							
Default	0.0							
Remarks								

Second Card if SBSTYP=4

	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

*ELEMENT_SEATBELT_SLIPRING

Purpose: Define seat belt slip ring.

Card Format

	1	2	3	4	5	6	7	8
Variable	SBSRID	SBID1	SBID2	FC	SBRNID			
Type	I	I	I	F	I			
Default	0	0	0	0.0	0			
Remarks	yes	yes	yes	yes	yes			

VARIABLE	DESCRIPTION
SBSRID	Slipring ID. A unique number has to be used.
SBID1	Seat belt element 1 ID
SBID2	Seat belt element 2 ID
FC	Coulomb friction coefficient
SBRNID	Slip ring node, NID

Remarks:

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 12.4) 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 timestep 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, μ. 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 $e^{\mu\Theta}$. 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 12.4). 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.

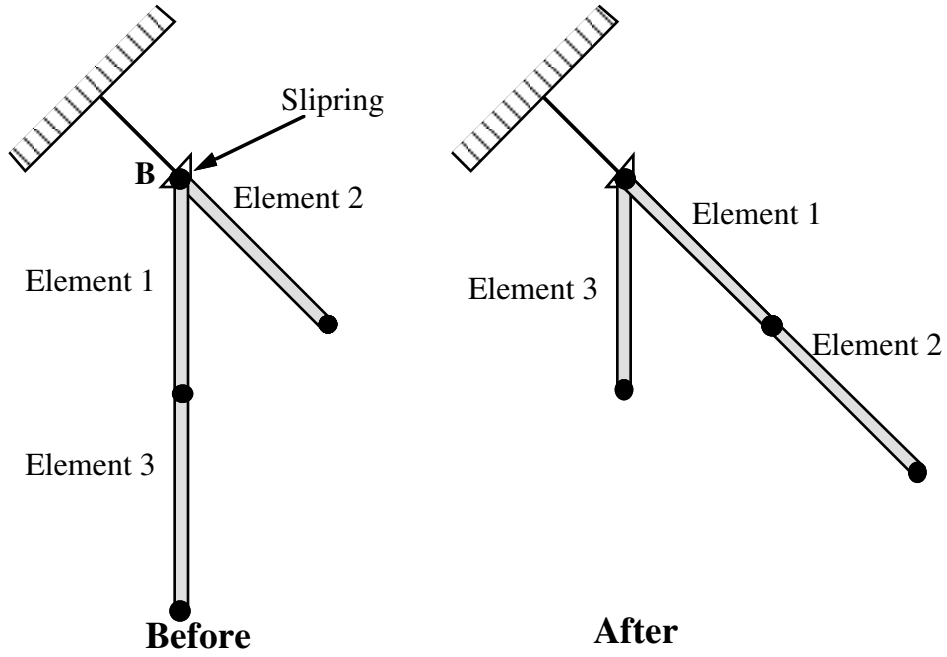


Figure 12.4. Elements passing through slipring.

*ELEMENT

***ELEMENT_SHELL_{*OPTION*}**

Available options include:

<BLANK>

THICKNESS

BETA

Purpose: Define three and four noded 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.

Card Format (10I8)

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				

Optional Card (Required if THICKNESS or BETA is specified after the keyword) (5E16.0)

	1	2	3	4	5	6	7	8	9	10
Variable	THIC1		THIC2		THIC3		THIC4		PSI	
Type	F		F		F		F		F	
Default	0.		0.		0.		0.		0.	
Remarks	1								2	

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
THIC1	Shell thickness at node 1
THIC2	Shell thickness at node 2
THIC3	Shell thickness at node 3
THIC4	Shell thickness at node 4
PSI	Orthotropic material angle offset measured from the reference (1-2 element side) axis, see remark 6 below. The angle is given in degrees.

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. PSI is defined only for orthotropic and anisotropic materials.

3. Counterclockwise node numbering determines the top surface, see Figure 12.5.

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 has a unique material orientation angle which measures the offset from some reference in the element. Each integration point through the shell thickness, typically though not limited to one point per ply, requires the definition of the orientation angle at that point. The reference is determined by the angle ψ, which can be defined for each element on the element card, and is measured from the 1-2 element side. Figures 12.6 and 12.7 depict these angles.

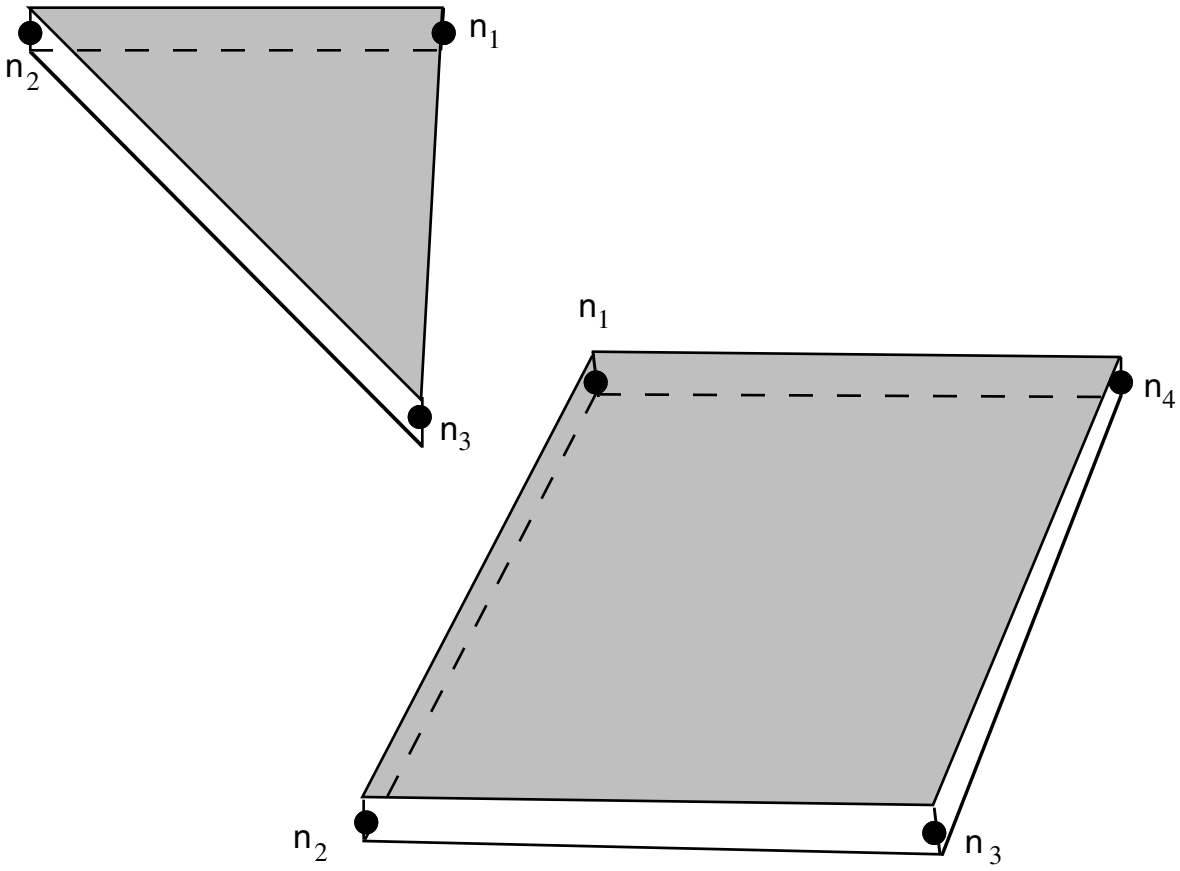


Figure 12.5. LS-DYNA shell elements. Counterclockwise node numbering determines the top surface.

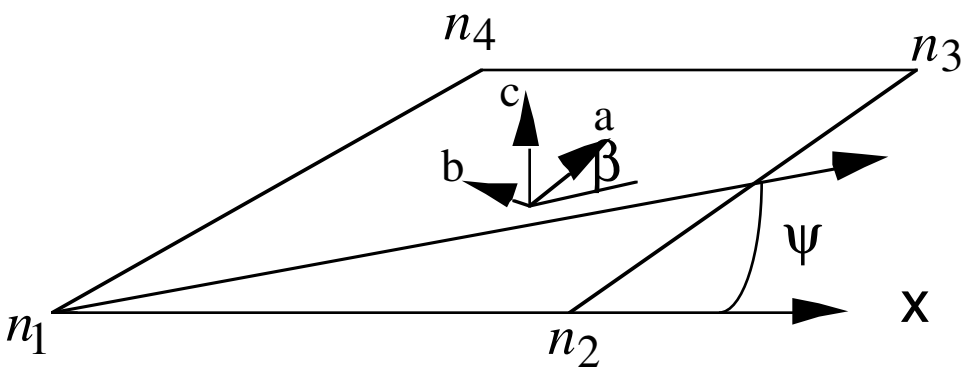


Figure 12.6 Orientation of material directions relative to the 1-2 side.

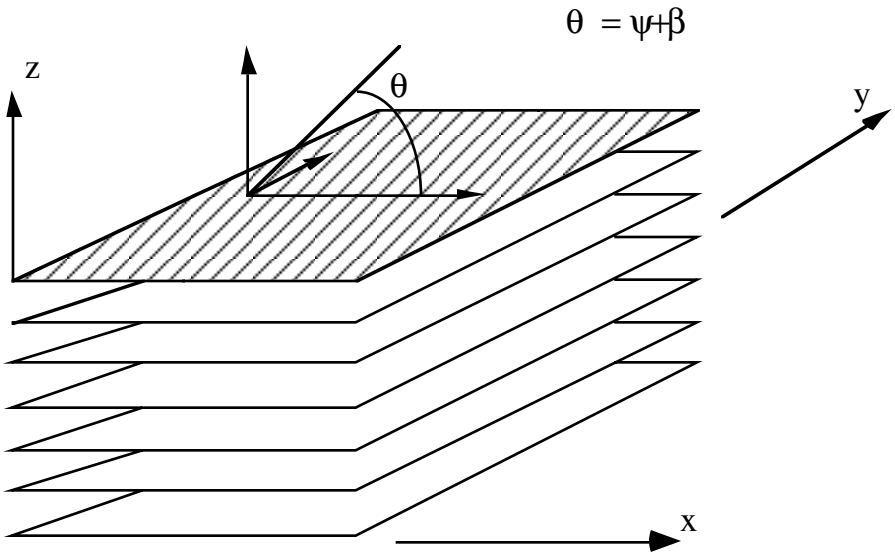


Figure 12.7. A multi-layer laminate can be defined. The angle β_i is defined for the ith lamina (integration point), see *SECTION_SHELL.

***ELEMENT_SOLID_{*OPTION*}**

Available options include:

<BLANK>

ORTHO

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.

Card Format (10I8)

	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									

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	2									

Optional card 2	1	2	3	4	5	6	7	8	9	10
Variable	D1		D2		D3					
Type	F		F		F					
Default	0.		0.		0.					
Remarks	2									

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
.	.
.	.
N8	Nodal point 8
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.

Remarks:

1. Four, six, and eight node elements are depicted in Figure 12.8 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

   6-noded pentahedron N1, N2, N3, N4, N5, N5, N6, N6

   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 tetrahedrons and pentahedrons elements, respectively.

2. 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 12.9):

$$\underset{\sim}{c} = \underset{\sim}{a} \times \underset{\sim}{d} \text{ and } \underset{\sim}{b} = \underset{\sim}{c} \times \underset{\sim}{a}.$$

   These vectors are internally normalized within LS-DYNA.

3. Stress output for solid elements is in the global coordinate system by default.

4. If vector **d** is input as a zero length vector, then A1 is interpreted as a rotation angle in degrees which is used for AOPT=3 on various orthotropic material cards such as *MAT_*OPTION* TROPIC_ELASTIC.

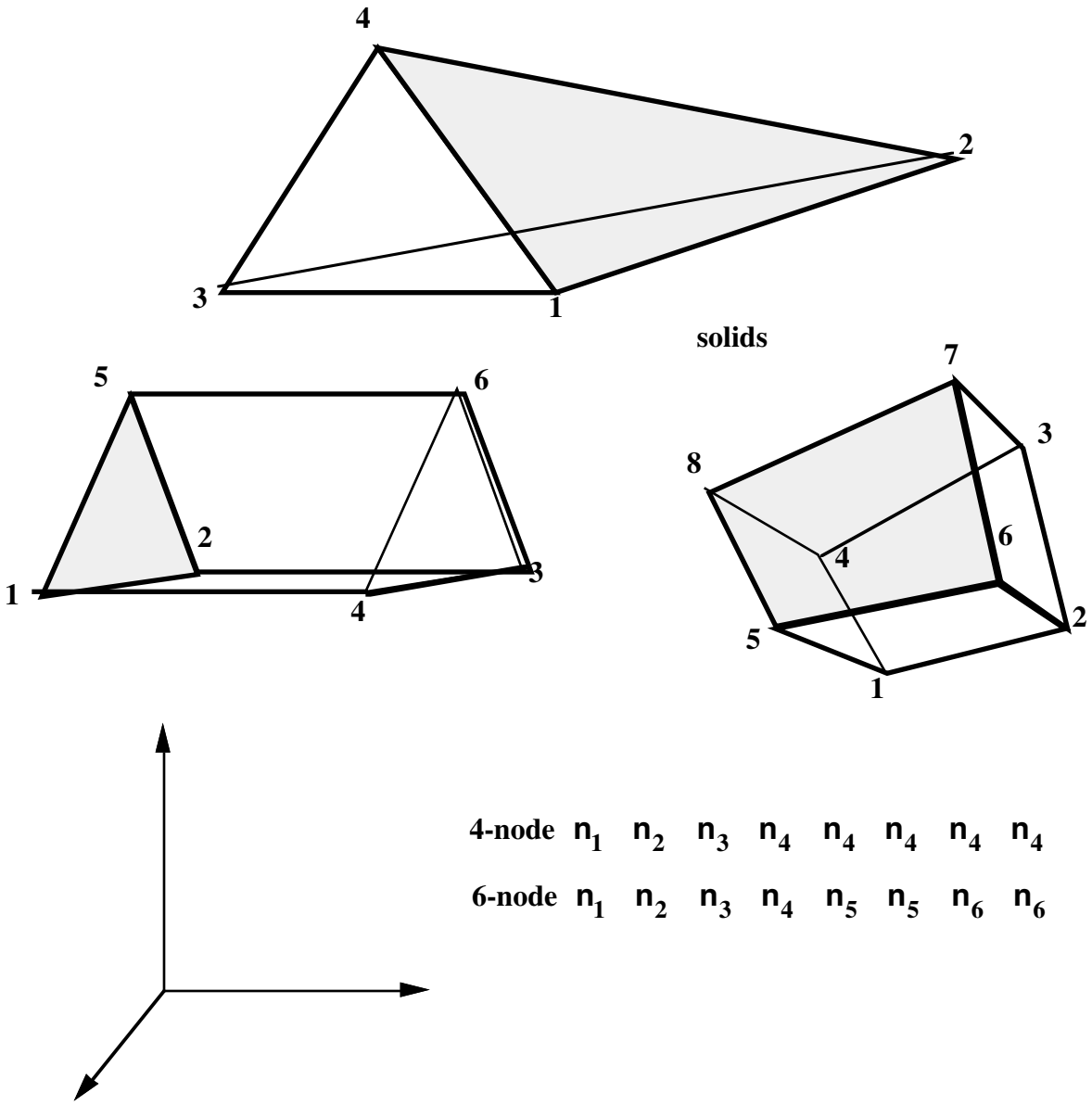


Figure 12.8. Four, six, and eight node solid elements. Nodes 1-4 are on the bottom surface.

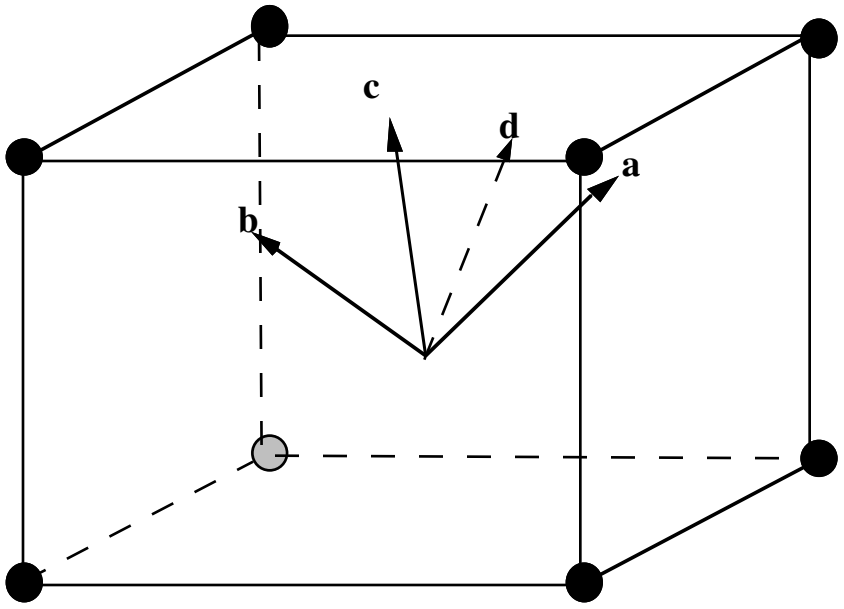


Figure 12.9 Two vectors **a** and **d** are defined and the triad is computed and stored. Vectors **b** and **d** lie in the same plane.

*ELEMENT_SPH

Purpose: Define a lumped mass element assigned to a nodal point .

Card Format (2I8,E16.0)

	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

Card 1	1	2	3	4	5	6	7	8	9	10
Variable	PSID									
Type	I									
Default	none									
Remarks	1,2,3									

VARIABLE	DESCRIPTION
PSID	Part set ID for trimming, see *SET_PART.

Remarks:

1. This command in combination with *DEFINE_CURVE_TRIM trims the requested parts before the job starts.

2 In case this command does not exist and only *DEFINE_CURVE_TRIM is available in the input, the related 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 proceduce outlined in the Remarks in the section *INTERFACE_SPRINGBACK.

*ELEMENT_TSHELL

Purpose: Define an eight node thick shell element which is available with either fully reduced or selectively reduced integration rules. This element can be used as an alternative to the 4 node shell elements. The major use is for transition between shell and solid regions or for modelling thick shells. The definition is completed by the *PART and *SECTION_TSHELL cards. The behavior of this shell exhibits excessive stiffness for large radius/thickness ratios

Card Format (10I8)

	1	2	3	4	5	6	7	8	9	10
Variable	EID	PID	N_1	N_2	N_3	N_4	N_5	N_6	N_7	N_8
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 12.10. 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.

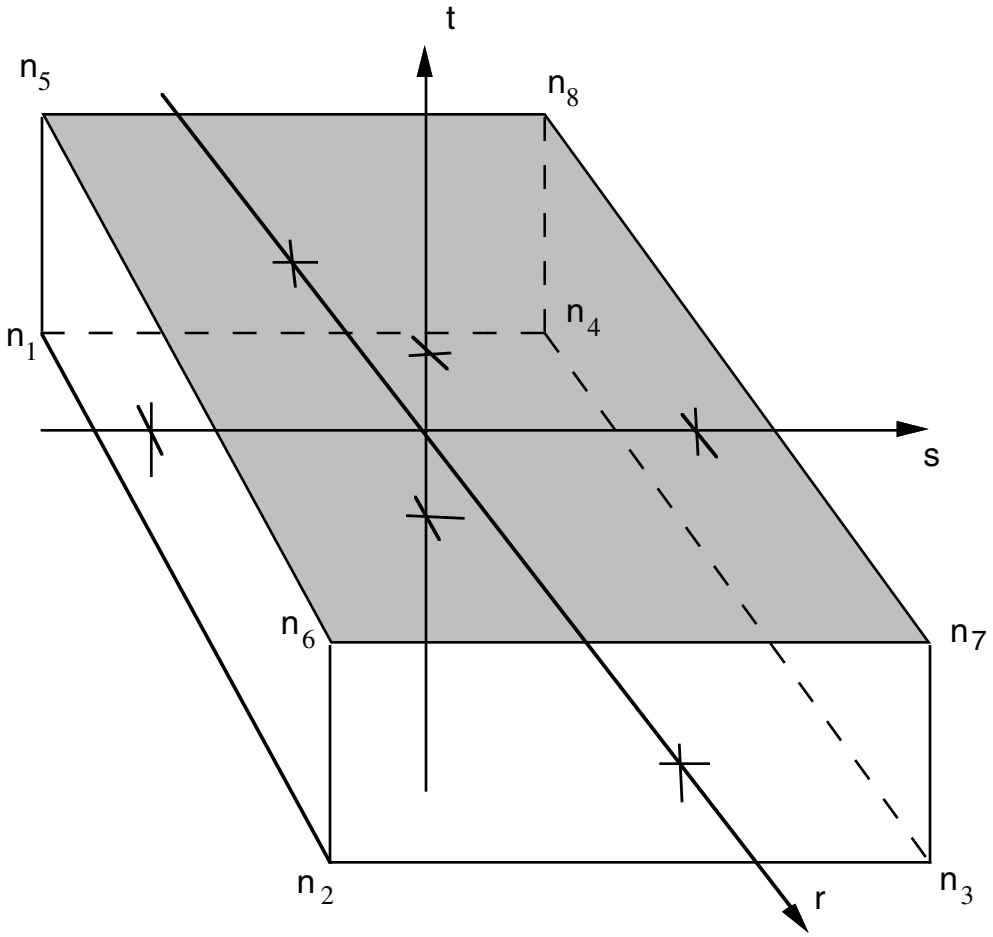


Figure 12.10. Solid 8-node Shell Element.

*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 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 14: ***EOS_JWLB**

An additional option **_TITLE** may be appended to all the ***EOS** 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 equation of state. At present LS-DYNA does make use of the title. Inclusion of titles gives greater clarity to input decks.

*EOS_LINEAR_POLYNOMIAL

Purpose: Define coefficients for linear polynomial EOS.

Card Format

Card 1

Variable	EOSID	C0	C1	C2	C3	C 4	C5	C6
Type	I	F	F	F	F	F	F	F

Card 2

Variable	E0	V0						
Type	F	F						

VARIABLE	DESCRIPTION
EOSID	Equation of state label
C0	
C1	
C2	
C3	
C4	
C5	
C6	
E0	Initial internal energy
V0	Initial relative volume

Remarks:

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 initial density.

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 γ is the ratio of specific heats. The pressure is then given by:

$$p = (\gamma - 1)\frac{\rho}{\rho_0} E$$

The units of E are the units of pressure.

*EOS_JWL

This is Equation of state Form 2.

Card Format

	1	2	3	4	5	6	7	8
Variable	EOSID	A	B	R1	R2	OMEG	E0	VO
Type	I	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
EOSID	Equation of state label
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 Format

	1	2	3	4	5	6	7	8
Variable	EOSID	A1	A2	A3	B1	B2	E0	V0
Type	I	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
EOSID	Equation of state label
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 Format

	1	2	3	4	5	6	7	8
Variable	EOSID	C	S1	S2	S3	GAMAO	A	E0
Type	I	F	F	F	F	F	F	F

Card 2

Variable	V0							
Type	F							

VARIABLE	DESCRIPTION
EOSID	Equation of state ID
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	I

Card 2 1 2 3 4

Variable	A10	A11	A12	A13
Type	F	F	F	F

Card 3 1 2 3 4

Variable	A20	A21	A22	A23
Type	F	F	F	F

Card 4 1 2 3 4

Variable	A30	A31	A32	A33
Type	F	F	F	F

Card 5 1 2 3 4

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	1	2	3	4
Variable	A60	A61	A62	A63
Type	F	F	F	F

Card 8	1	2	3	4
Variable	A70	A71	A72	A73
Type	F	F	F	F

Card 9	1	2		
Variable	A14	A24		
Type	F	F		

Card 10	1	2	3	4
Variable	ALPH	BETA	E0	V0
Type	F	F	F	F

VARIABLE	DESCRIPTION
EOSID	Equation of state ID
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	
A71	

VARIABLE	DESCRIPTION
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

$$p = \frac{F_1 + F_2 E + F_3 E^2 + F_4 E^3}{F_5 + F_6 E + F_7 E^2}\left(1 + \alpha\mu\right)$$

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 Format

Card 1

Variable	EOSID	C0	C1	C2	C3	C4	C5	C6
Type	I	F	F	F	F	F	F	F

Card 2

Variable	E0	V0	LCID					
Type	F	F	I					

VARIABLE	DESCRIPTION
EOSID	Equation of state label
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 Format

Card 1

Variable	EOSID	A	B	XP1	XP2	FRER	G	R1
Type	F	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 label
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

$$P_e = r_1 e^{-r5Ve} + r_2 e^{-r6Ve} + r_3 \frac{Te}{Ve} \left(r_3 = \omega_e \,\mathrm{cvr}\right)$$

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

$$P_p = a\, e^{-xp1Vp} + be^{-xp2Vp} + \frac{gTp}{Vp} \left(g = \omega_p \,\mathrm{cvp}\right)$$

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 (Te=Tp, pe=pp) and the relative volumes are additive, i.e.,

$$V = (1\text{-}F)\ Ve + Vp$$

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

$$\frac{\partial F}{\partial t} = \mathrm{freq}\left(1-F\right)^{\mathrm{frer}} \left(Ve^{-1} - 1 - \mathrm{ccrit}\right)^{\mathrm{eetal}} \qquad \text{(Ignition)}$$

$$+ \mathrm{grow1}(1-F)^{\mathrm{esl}}\ F^{\mathrm{arl}}\ p^{\mathrm{em}} \qquad \text{(Growth)}$$

$$+ \mathrm{grow2}(1-F)^{\mathrm{es2}} F^{\mathrm{ar2}}\ p^{\mathrm{en}} \qquad \text{(Completion)}$$

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. 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 85 and 89 are temperature and burn fraction, respectively. See *DATABASE_EXTENT_BINARY if these output variables are desired in the databases for post-processing.

*EOS_TABULATED_COMPACTION

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	EOSID	GAMA	E0	V0				
Type	I	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 label
εV1,εV2,..εVN	ln V
C1,C2,..CN	
T1,T2,..TN	
K1,K2,..KN	
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. 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 13.1. 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.

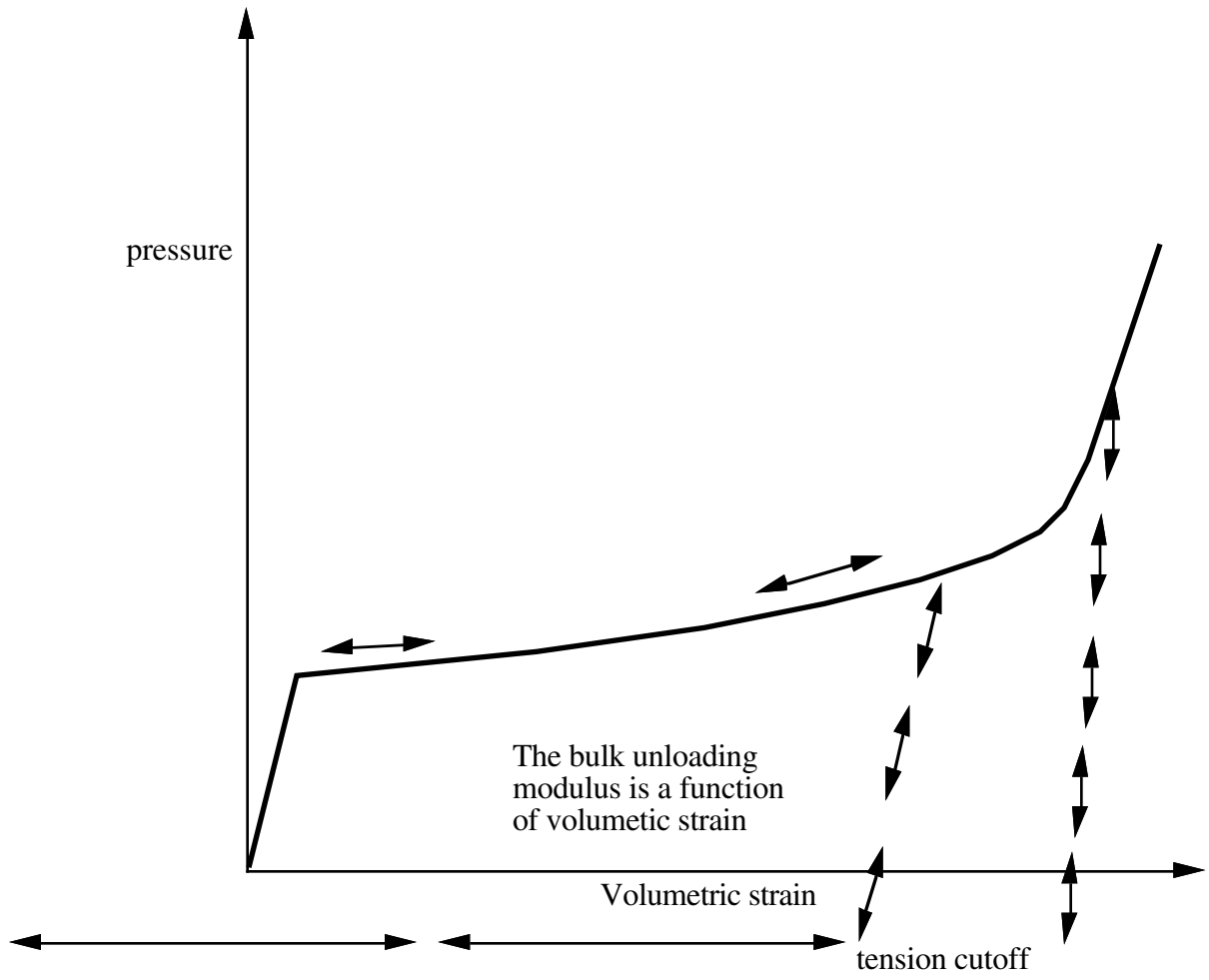


Figure 13.1. Pressure versus volumetric strain curve for Equation of state Form 8 with compaction. In the compacted states the bulk unloading modulus depends on the peak volumetric strain.

*EOS_TABULATED

This is Equation of state Form 9.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	EOSID	GAMA	E0	VO				
Type	I	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 and T_i. A total of 7 cards must be defined.

VARIABLE	DESCRIPTION
EOSID	Equation of state label
εV1,εV2,..εVN	ln V
C1,C2,..CN	
T1,T2,..TN	

VARIABLE	DESCRIPTION
GAMA	γ
E0	Initial internal energy
V0	Initial relative volume

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. 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 Format

Card 1

Variable	EOSID	A	B	XP1	XP2	FRER		
Type	F	F	F	F	F	F		

Card 2

Variable	g	R1	R2	R3	R5			
Type	F	F	F	F	F			

Card 3

Variable	R6	FMXIG	FREQ	GROW1	EM			
Type	F	F	F	F	F			

Card 4

Variable	AR1	ES1	CVP	CVR	EETAL	CCRIT	ENQ	TMP0
Type	F	F	F	F	F			

Card 5

Variable	GROW2	AR2	ES2	EN	FMXGR	FMNGR		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
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
TMP0	Initial Temperature (298°K)
GROW2	Second burn rate coefficient

VARIABLE	DESCRIPTION
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 modelled 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:

$$p = A e^{-R_1 V} + B e^{-R_2 V} + \frac{\omega C_v T}{V - d}$$

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 dependance 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 dependences 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 Format

Card 1	1	2	3	4	5	6	7	8
Variable	EOSID	NLD	NCR	MU1	MU2	IE0	EC0	
Type	I	F	F	F	F	F	F	

Repeat Cards 2, etc. as required for ECC_i and PC_i.

VARIABLE	DESCRIPTION
EOSID	Equation of state label
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 13.2. 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.

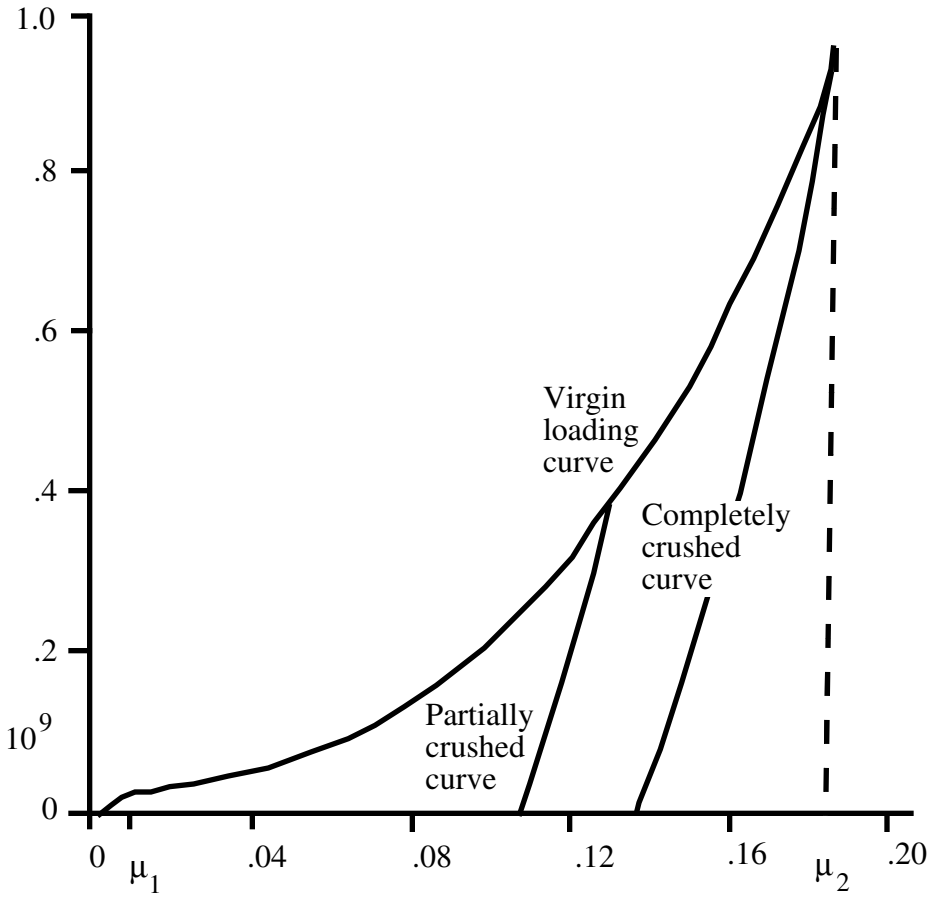


Figure 13.2. Pressure versus compaction curve.

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,

$$K = \frac{\partial P_{cc}(\mu_a)}{\partial \mu}(1+\mu_a)$$

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_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 Format

Card 1

Variable	EOSID	A1	A2	A3	A4	A5		
Type	I	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 label
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.
AL4	$A_{\lambda 4}$, equation of state coefficient, see below.

VARIABLE	DESCRIPTION
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 realtive volume.

Remarks:

The JWLB equation-of-state defines the pressure as

$$p = \sum_{i=1}^{5} A_i \left(1 - \frac{\lambda}{R_i V}\right) e^{-R_i V} + \frac{\lambda E}{V} + C\left(1 - \frac{\lambda}{\omega}\right) V^{-(\omega+1)}$$

$$\lambda = \sum_{i=1}^{5} A_i \left(A_{\lambda i} V + B_{\lambda i}\right) e^{-R_{\lambda i} V} + \omega$$

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

*HOURGLASS

*HOURGLASS

Purpose: Define hourglass and bulk viscosity properties. Using the *PART definition this specification is connected to the elements.

An additional option **_TITLE** may be appended to ***HOURGLASS** 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.

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	HGID	IHQ	QM	IBQ	Q2	Q1	QB	QW
Type	I	I	F	I	F	F	F	F
Default	0	1	.10	0	1.5	0.06	QM	QM
Remark		1	2&4	3	3	3	4	4

VARIABLE — **DESCRIPTION**

HGID — Hourglass ID. Unique numbers have to be specified.

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,
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,
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 is mandatory for the implicit options.

VARIABLE	DESCRIPTION
	EQ:8:Applicable to the type 16 fully integrated shell element. IHQ=8 activates the warping stiffness for accurate solutions. A speed penalty of 25% is common for this option.
	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 (for IHQ<6) may cause instabilities. The recommended default applies to all options. 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, QM=1.0 gives an accurate coarse mesh bending stiffness that does not lock in the incompressible limit. For type 6 values such as 0.001-0.01 will avoid an overly stiff response.
IBQ	Bulk viscosity type (See remark 3. below.): EQ.1: standard LS-DYNA.
Q2	Quadratic bulk viscosity coefficient.
Q1	Linear bulk viscosity coefficient.
QB	Hourglass coefficient for shell bending. The default: QB=QM. See remark 4. below.
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. 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

$$\dot{q}_\alpha^M = \tau_I \, \hat{\upsilon}_{\alpha I}$$

$$\dot{q}_\alpha^B = \tau_I \hat{\theta}_{\alpha I}$$

$$\dot{q}_3^W = \tau_I \, \hat{\upsilon}_{zI}$$

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.

*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*}

Generally, the *INCLUDE command is used without any options; however, two options are available:

***INCLUDE_STAMPED_PART**

***INCLUDE_TRANSFORM**

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. Between the stamped part and the crash part, note the following points:

1. The 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.

Card Format. The card is required.

Card 1 1

Variable	FILENAME
Type	C

If the STAMPED_PART option is active then define the following input.
Card Format for the STAMPED_PART option

Card 2	1	2	3	4	5	6	7	8
Variable	PID	THICK	PSTRN	STRAIN				
Type	I	I	I	I				
Default	none	0	0	0				

Card 3	1	2						
Variable	N1S	N2S	N3S	N1C	N2C	N3C		
Type	I	I	I	I	I	I		

If the TRANSFORM option is active then define the following input
Card Format for the TRANSFORM option

Card 2	1	2	3	4	5	6	7	8
Variable	IDNOFF	IDEOFF	IDPOFF	IDMOFF	IDSOFF	IDFOFF	IDDOFF	
Type	I	I	I	I	I	I	I	

Card 3	1	2						
Variable	IDROFF							
Type	I							

Card 4	1	2	3	4	5	6	7	8
Variable	FCTMAS	FCTTIM	FCTLEN	FCTTEM	INCOUT			
Type	F	F	F	F	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.
PID	Part ID of crash part for remapping.
THICK	Thickness remap: EQ.0: map thickness EQ.1: do not map thickness
PSTRN	Plastic strain remap: EQ.0: map plastic strain EQ.1: do not plastic strain
STRAIN	Strain remap: EQ.0: map strains EQ.1: do not map strains
N1S	First of 3 nodes need to reorient the stamped part.
N2S	Second of 3 nodes need to reorient the stamped part.
N3S	Third of 3 nodes need to reorient the stamped part.
N1C	First of 3 nodes need to reorient the crash model part.
N2C	Second of 3 nodes need to reorient the crash model part.

VARIABLE	DESCRIPTION
N3C	Third of 3 nodes need to reorient the crash model part.
IDNOFF	Offset to node ID.
IDEOFF	Offset to element ID.
IDPOFF	Offset to part ID, nodal rigid body ID, and constrained nodal set ID.
IDMOFF	Offset to material 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	Offset to section ID, hourglass ID, and any equation of state ID.
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.
FCTLEN	Length transformation factor.
FCTTEM	Temperature transformation factor: F to C (Farenheit to Centigrad), C to F, F to K, K to F, and so on.
INCOUT	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 tranformation will be applied. See the input DEFINE_TRANSFORM.

Remarks:

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, wheelhouse.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
```

The TRANSFORM option should be used cautiously, and the transformed quantities should be checked closely for correctness.

*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_CFD**

***INITIAL_DETONATION**

***INITIAL_FOAM_REFERENCE_GEOMETRY**

***INITIAL_MOMENTUM**

***INITIAL_STRAIN_SHELL**

***INITIAL_STRESS_BEAM**

***INITIAL_STRESS_SHELL**

***INITIAL_STRESS_SOLID**

***INITIAL_TEMPERATURE**_*OPTION*

***INITIAL_VEHICLE_KINEMATICS**

Two mutually exclusive methods are available for initial velocity generation:

***INITIAL_VELOCITY**

***INITIAL_VELOCITY_NODE**

and:

***INITIAL_VELOCITY_GENERATION**

The latter is convenient for specifying initial rotational velocities about arbitrary axes. These method for velocity generation must not be mixed in a single input deck.

***INITIAL_VOID**_*OPTION*

***INITIAL_VOLUME_FRACTION**

*INITIAL_CFD

Purpose: Specify initial conditions for all nodal variables in the incompressible CFD solver.

Card Format (1 of 3)

Card 1	1	2	3	4	5	6	7	8
Variable	U	V	W	T	H	RHO	Z1	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 Format (2 of 3)

Card 2	1	2	3	4	5	6	7	8
Variable	Z3	Z4	Z5	Z6	Z7	Z8	Z9	Z10
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 Format (3 of 3)

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

VARIABLE	DESCRIPTION
U	Initial x-velocity
V	Initial y-velocity
W	Initial z-velocity
T	Initial temperature
H	Initial enthalpy
RHO	Initial density
Z1	Initial Species-1 concentration
Z2	Initial Species-2 concentration
. . .	
Z10	Initial Species-10 concentration
K	Initial turbulent kinetic energy
EPS	Initial turbulent dissipation rate

*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 Format

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 16.1.
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 16.1.
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 possiblities 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 deonation 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-POST. 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:

$$p(t) = p_o e^{-\frac{t}{\tau}}.$$

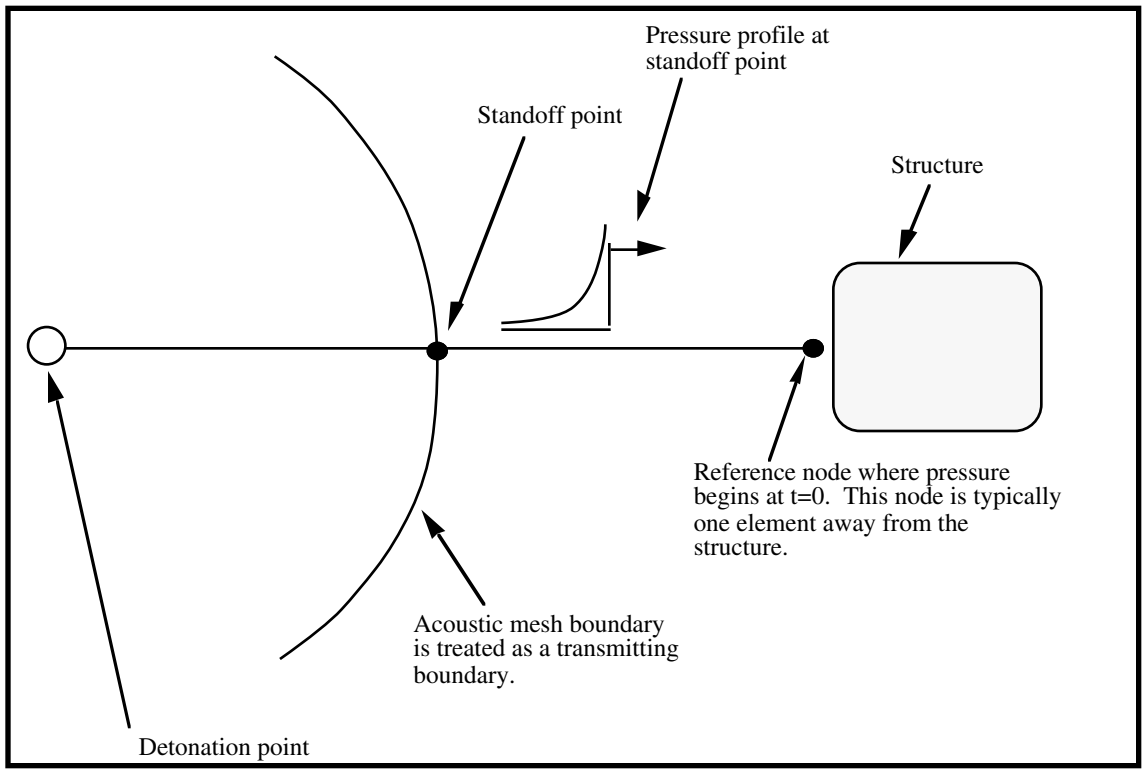


Figure 16.1 Initialization of the initial pressures due to an explosive disturbance is performed in the acoustic media. LS-DYNA automatically determines the acoustic mesh boundary and applies the pressure time history to the boundary. This option is only applicable to the acoustic element formulation, see *SECTION_SOLID.

*INITIAL_FOAM_REFERENCE_GEOMETRY

Purpose: The reference configuration allows stresses to be initialized in the material model, *MAT_LOW_DENSITY_FOAM 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:

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

where 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
Y	y coordinate
Z	z coordinate

*INITIAL_MOMENTUM

Purpose: Define initial momentum to be deposited in solid elements. This option is to crudely simulate an impulsive type of loading.

Card Format

	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_STRAIN_SHELL

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 Format

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
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 Format

Card 1	1	2	3	4	5	6	7	8
Variable	EID	RULE	NPTS					
Type	I	I	I					
Default	none	none	none					

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: truss element or discrete beam element, 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.

VARIABLE	DESCRIPTION
NPTS	Number of integration points output.
SIGIJ	Define the IJ stress component.
EPS	Effective plastic strain

*INITIAL_STRESS_SHELL

Purpose: Initialize stresses and plastic strains for shell elements.

Define as many 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 elments which are initialized. The data will be interpolated by LS-DYNA3D.

Card Format

Card 1

Variable	EID	NPLANE	NTHICK					
Type	I	I	I					
Default	none	none	none					

Define NPLANE X NTHICK cards below, one per integration point. 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:

$$\left(-\frac{\sqrt{3}}{3},-\frac{\sqrt{3}}{3}\right),\qquad \left(\frac{\sqrt{3}}{3},-\frac{\sqrt{3}}{3}\right),\qquad \left(\frac{\sqrt{3}}{3},\frac{\sqrt{3}}{3}\right),\qquad \left(-\frac{\sqrt{3}}{3},\frac{\sqrt{3}}{3}\right),$$

respectively.

Card 2,...

Variable	T	SIGxx	SIGyy	SIGzz	SIGxy	SIGyz	SIGzx	EPS
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
EID	Element ID
NPLANE	Number of in plane integration points being output.
NTHICK	Number of through thickness integration points.
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

*INITIAL_STRESS_SOLID

Purpose: Initialize stresses and plastic strains for solid elements.

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 Format

Card 1

Variable	EID	NINT						
Type	I	I						
Default	none	none						

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.

Card 2,...

Variable	SIG11	SIG22	SIG33	SIG12	SIG23	SIG31	EPS	
Type	F	F	F	F	F	F	F	

VARIABLE	DESCRIPTION
EID	Element ID
NINT	Number of integration points either 1 or 8.
SIGIJ	Define the IJ stress component.
EPS	Effective plastic strain

*INITIAL

INITIAL_TEMPERATURE_*OPTION

Available options are:

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 Format

Card 1	1	2	3	4	5	6	7	8
Variable	NSID/NID	TEMP						
Type	I	I						
Default	none	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.

Remark:

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 Format

Card 1	1	2	3	4	5	6	7	8
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	1	2	3	4	5	6	7	8
Variable	VX	VY	VZ	AAXIS	BAXIS	CAXIS		
Type	F	F	F	I	I	I		
Default	0.	0.	0.	0	0	0		

Card 3	1	2	3	4	5	6	7	8
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).
WB	Angular velocity component for the second axis (radian/second).
WC	Angular velocity component for the third axis (radian/second).

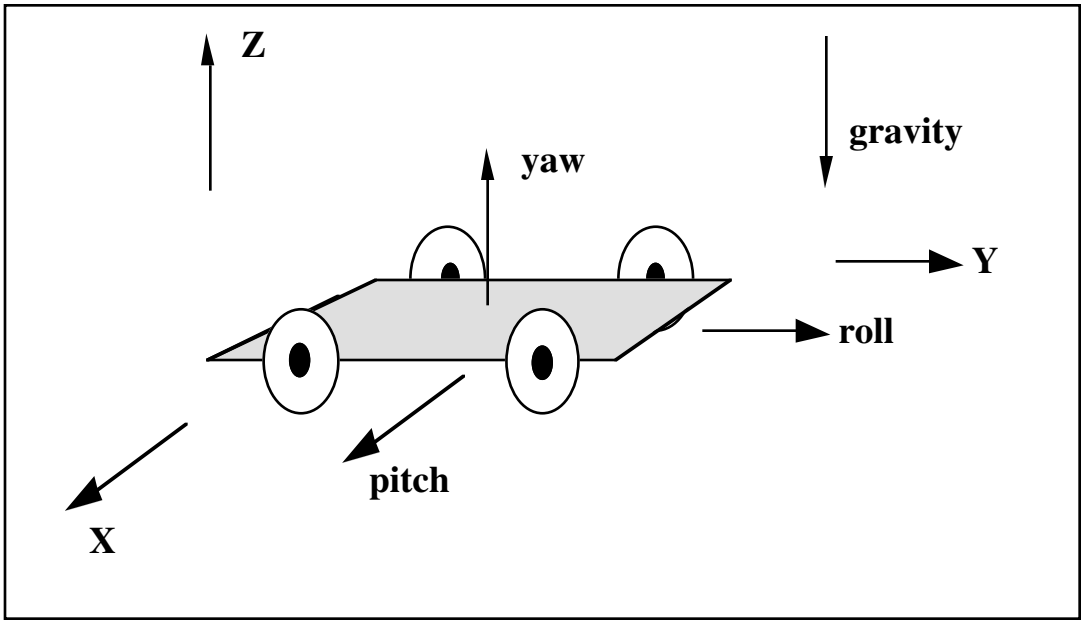


Figure 16.1. The vehicle pictured is to be oriented with a successive rotation sequence about the yaw, pitch, and roll axes, respectively. Accordingly, AAXIS=3, BAXIS=1, and CAXIS=2. The direction of gravity is given by GRAV=-3.

*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 Format

Card 1	1	2	3	4	5	6	7	8
Variable	NSID	NSIDEX	BOXID					
Type	I	I	I					
Default	none	0.	0.					
Remark	1							

Card 2	1	2	3	4	5	6	7	8
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.

	1	2	3	4	5	6	7	8
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:

VARIABLE	DESCRIPTION
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.
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. 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_NODE

Purpose: Define initial nodal point velocities for a node.

Card Format

	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	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

See remark on *INITIAL_VELOCITY card.

*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.

Card Format

Card 1

Variable	ID	STYP	OMEGA	VX	VY	VZ		
Type	I	I	F	F	F	F		
Default	none	none	0.	0.	0.	0.		

Card 2

Variable	XC	YC	ZC	NX	NY	NZ	PHASE	
Type	F	F	F	F	F	F	I	
Default	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.
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 rotational axis.
VX	Initial translational velocity in global x-direction.
VY	Initial translational velocity in global y-direction.
VZ	Initial translational velocity in global z-direction. .

VARIABLE	DESCRIPTION
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 applied immediately, EQ.1. Velocities applied after dynamic relaxation.

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_VOID**_*OPTION*

Available options are:

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 Format

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:

Remark:

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 fraction of different materials in multi-material ALE, or in single material and void, models.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	EID	VF1	VF2					
Type	I	F	F					
Default	none	0.0	0.0					

VARIABLE	DESCRIPTION
EID	Element ID.
VF1	Volume fraction of multi-material group 1
VF2	Volume fraction of multi-material group 2. Only needed in simulations with 3 material groups. Otherwise VF2=1-VF1.

*INTEGRATION

*INTEGRATION_BEAM

Purpose: Define user defined through the thickness integration rules for the beam element.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	IRID	NIP	RA	ICST				
Type	I	I	F	I				
Default	none	0	0.0	0				

Define the following card if and only if ICST>0.

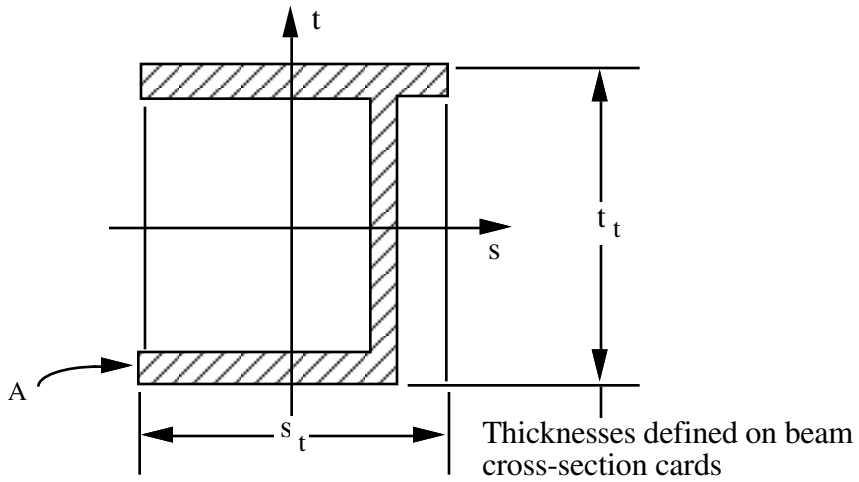
	1	2	3	4	5	6	7	8
Variable	W	TF	D	TW	SREF	TREF		
Type	F	F	F	F	F	F		
Default	none	none	none	none	1.0	0.0		

Define NIP cards below (Skip if NIP=0).

	1	2	3	4	5	6	7	8
Variable	S	T	WF					
Type	F	F	F					

*INTEGRATION

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 17.1.
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 Figures 17.3a and 17.3b. EQ.1: W-section, EQ.2: C-section, EQ.3: Angle section, EQ.4: T-section, EQ.5: Rectangular tubing, EQ.6: Z-section,. EQ.7: Trapezoidal section
W	w, flange width
TF	t_f, flange thickness
D	d, depth
TW	t_w, web thickness
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, see also *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, see also *SECTION_BEAM.
S	Normalized s coordinate of integration point, $-1 \le s \le 1$.
T	Normalized t coordinate of integration point, $-1 \le t \le 1$.
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 17.2.



$$\text{Relative Area} = \frac{A}{s_t \cdot t_t}$$

Figure 17.1. Definition of relative area for user defined integration rule.

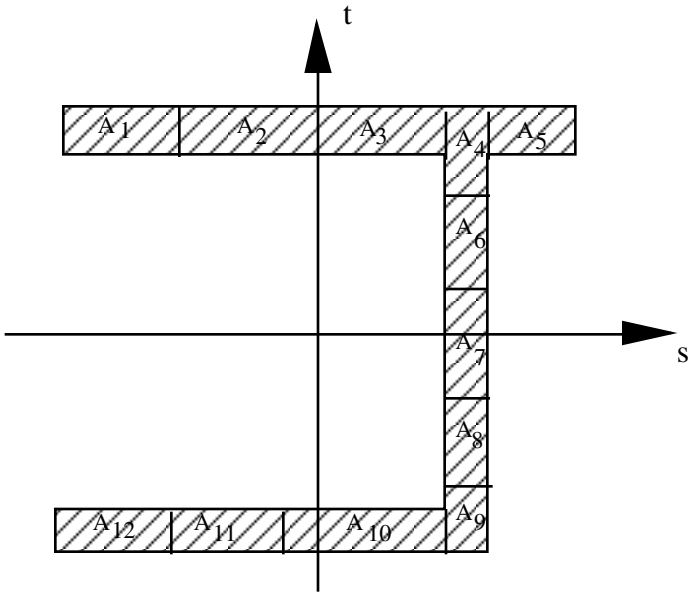


Figure 17.2. Definition of integration points for user defined integration rule.

Remarks:

The input for standard beam section types is defined below. In Figures 17.3a and 17.3b, 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:

w = flange width
t_f = flange thickness
d = depth
t_w = web thickness
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

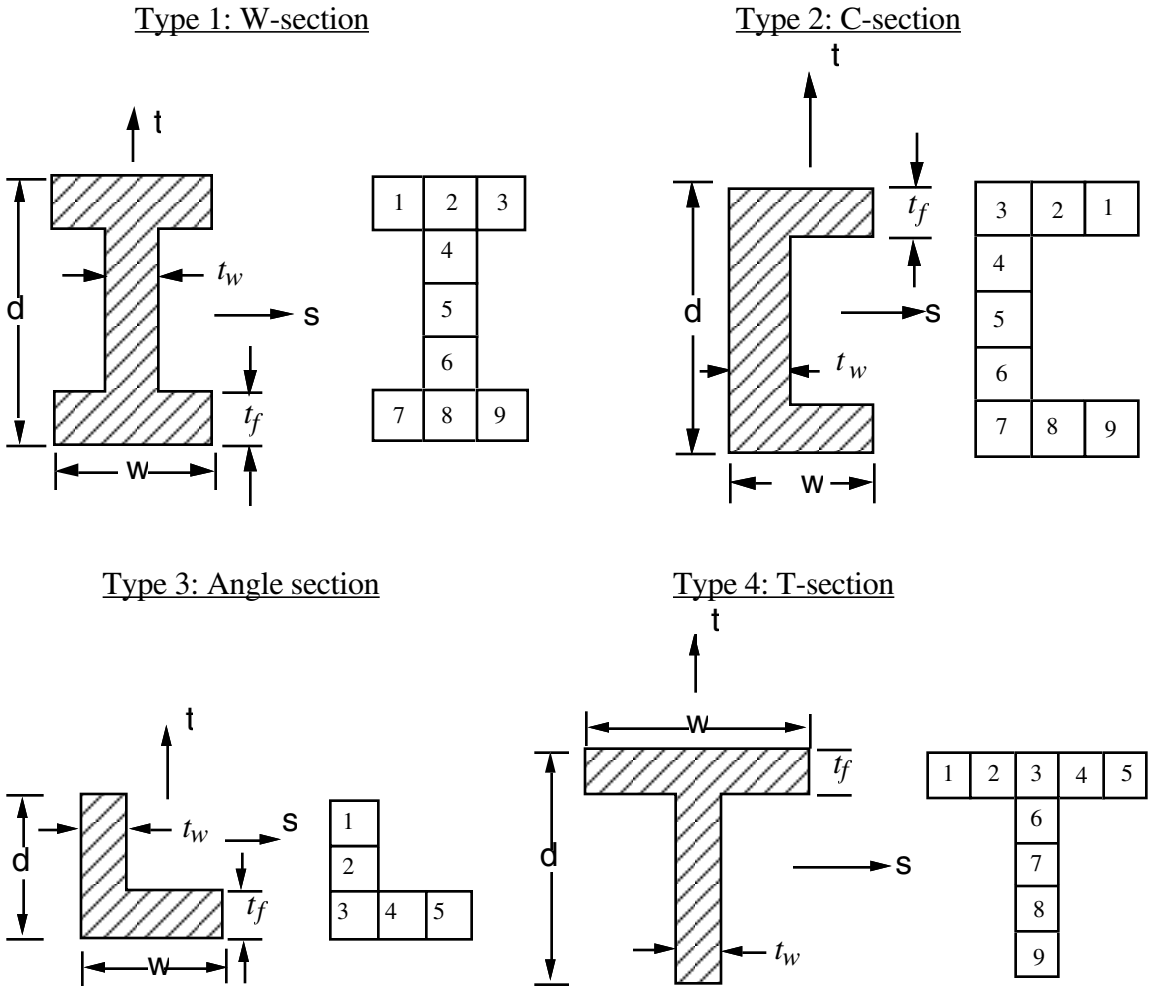
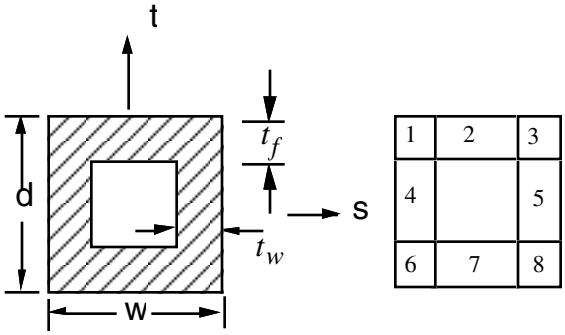
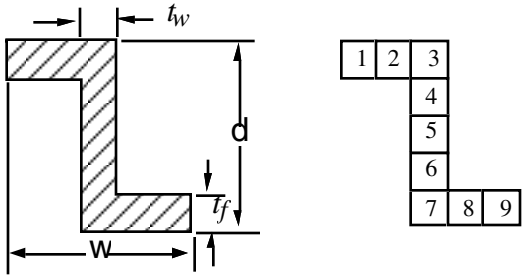


Figure 17.3a. Standard beam cross sections.

Type 5: Rectangular tubing



Type 6: Z-section



Type 7: Trapezoidal section.

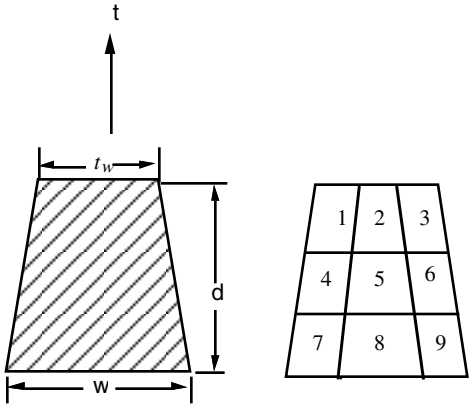


Figure 17.3b. Standard beam cross sections.

*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 nodel thick shell (ELEMENT_TSHELL).

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	IRID	NIP	ESOP					
Type	I	I	I					

Define NIP cards below if ESOP = 0.

	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.
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 17.4.

VARIABLE	DESCRIPTION
PID	Optional part ID if different from the PID specified on the element card. The material type is not allowed to change, see *PART. 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 throuigh 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.

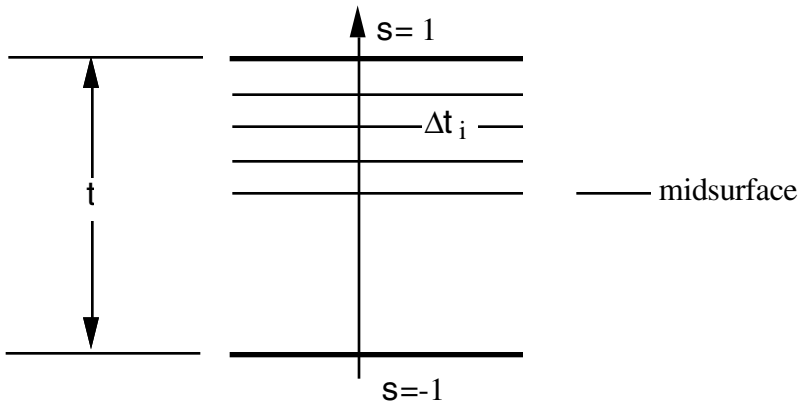


Figure 17.4. In the user defined shell integration rule the ordering of the integration points is arbitrary.

*INTERFACE

***INTERFACE_COMPONENT**_*OPTION*

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 Format

	1	2	3	4	5	6	7	8
Variable	SID							
Type	I							

VARIABLE	DESCRIPTION
SID	Set ID, see *SET_NODE or *SET_SEGMENT.

INTERFACE_LINKING_DISCRETE_NODE_*OPTION

Options include:

NODE

SET

Purpose: Define an interface for linking discrete nodes to an interface file. This link applies to spring and beam elements only.

Card Format

	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_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 Format

	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_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 Format

	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_JOY

Purpose: Define an interface for linking calculations by moving a nodal interface.

Card Format

	1	2	3	4	5	6	7	8
Variable	SID							
Type	I							

VARIABLE	DESCRIPTION
SID	Nodal set ID, see *SET_NODE.

INTERFACE_SPRINGBACK_*OPTION1_OPTION2

Options included for *OPTION1* are:

NIKE3D

DYNA3D

NASTRAN

SEAMLESS

and for *OPTION2:*

THICKNESS

NOTHICKNESS

See the remarks below.

Purpose: Define a material subset for an implicit springback calculation in LS-NIKE3D and any nodal constraints to eliminate rigid body degrees-of-freedom.

Card Format

	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 Format

	1	2	3	4	5	6	7	8
Variable	NID	TC	RC					
Type	I	F	F					

VARIABLE	DESCRIPTION
NID	Node ID, see *NODE.
TC	Tranlational 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 default is NIKE3D with the THICKNESS option for each element and the NOTHICKNESS option is not available for NIKE3D. The file name for the NIKE3D option is "nikin". The adaptive constraint option is not available for this option.

2. The NOTHICKNESS option is available for DYNA3D 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.

3. 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_ DYNA3D.

   (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_DYNA3D.

Remarks for Seamless Springback:

Seamless springback avoids the use of LS-NIKE3D for static springback analysis. Instead, LS-DYNA automatically and seamlessly switches from explicit 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_DYNA3D to generate a "dynain" file after forming, and then perform a second simulation running LS-DYNA in fully implicit mode for springback. See Appendix M for a description of how to run an implicit analysis using LS-DYNA.

The implict 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) are required 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.

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 *CONTROL_IMPLICIT_GENERAL, *CONTROL_IMPLICIT_AUTO and *CONTROL_IMPLICIT_STABILIZATION keywords.

*LOAD

The keyword *LOAD provides a way of defining applied forces. The keyword control cards in this section are defined in alphabetical order:

***LOAD_BEAM**_*OPTION*

***LOAD_BLAST**

***LOAD_BODY**_*OPTION*

***LOAD_BODY_GENERALIZED**

***LOAD_BRODE**

***LOAD_DENSITY_DEPTH**

***LOAD_HEAT_GENERATION**_*OPTION*

***LOAD_MASK**

***LOAD_NODE**_*OPTION*

***LOAD_RIGID_BODY**

***LOAD_SEGMENT**

***LOAD_SEGMENT_SET**

***LOAD_SHELL**_*OPTION*

***LOAD_SSA**

***LOAD_SUPERPLASTIC_FORMING**

***LOAD_THERMAL**_*OPTION*

LOAD_BEAM_*OPTION

Options include:

ELEMENT

SET

urpose: Apply the distributed traction load along any local axis of beam or a set of beams. The local axes are defined in Figure 19.1, see also *ELEMENT_BEAM.

Card Format

	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: along r-axis of beam, EQ.2: along s-axis of beam, EQ.3: along t-axis of beam.
LCID	Load curve ID, see *DEFINE_CURVE.
SF	Load curve scale factor. This is for a simple modification of the function values of the load curve.

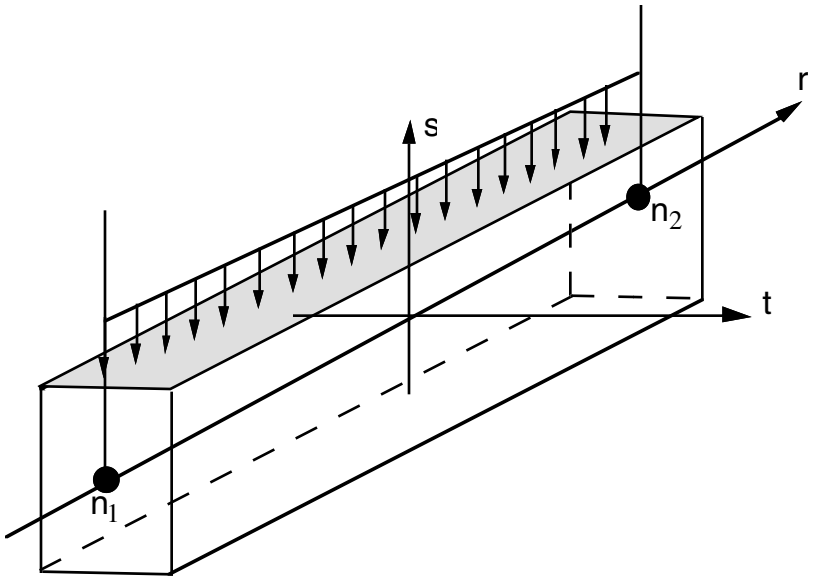


Figure 19.1. Applied traction loads are given in force per unit length. The s and t directions are defined on the *ELEMENT_BEAM keyword.

*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 pressue values when used in conjuntion with the keywords: *LOAD_SEGMENT, *LOAD_SEGMENT_SET, or *LOAD_SHELL.

Card Format

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, pounds, 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 - hemispherical charge situated on the surface EQ.2: air burst - spherical charge at least one charge diameter away from the surface (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.

***LOAD_BODY**_*OPTION*

Options incude for base accelerations:

X

Y

Z

for angular velocities:

RX

RY

RZ

and to specifiy 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.

Card Format for options: X, Y, Z, RX, RY, and RZ.

	1	2	3	4	5	6	7	8
Variable	LCID	SF	LCIDDR	XC	YC	ZC		
Type	I	F	I	F	F	F		
Default	none	1.	0	0.	0.	0.		

Card Format for option: PARTS.

	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.
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

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.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	N1	N2	LCID	DRLCID	XC	YC	ZC	
Type	I	I	I	I	F	F	F	
Default	none	none	none	0	0.	0.	0.	
Remarks								

Card 2	1	2	3	4	5	6	7	8
Variable	AX	AY	AZ	OMX	OMY	OMZ		
Type	F	F	F	F	F	F		
Default	0.	0.	0.	0.	0.	0.		
Remarks	1, 2	1, 2	1, 2	3, 4, 5	3, 4, 5	3, 4, 5		

VARIABLE	DESCRIPTION
N1	Beginning node ID for body force load.
N2	Ending node ID for body force load.
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.

VARIABLE	DESCRIPTION
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

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_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 Format

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.

VARIABLE	DESCRIPTION
TALC	Load curve number giving time of arrival versus range relative to Brode origin (space, time), see *DEFINE_CURVE and remark below.
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.

Remark:

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 Format

	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 abcissa of the curve and must vary (increase) with depth, i.e., an infinite slope is not allowed.

LOAD_HEAT_GENERATION_*OPTION

Available options are:

SET

SOLID

Purpose: Define solid elements or solid element set with heat generation.

Card Format

	1	2	3	4	5	6	7	8
Variable	SID	LCID	CMULT					
Type	I	I	F					
Default	none	none	1.0					

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.

*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 Format

	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					

	1	2	3	4	5	6	7	8
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.

VARIABLE	DESCRIPTION
VID1	Vector ID normal to the suface 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 19.3.
OFF	Pressure loads will be discontinued if $\left\|VID1 \cdot n_{shell}\right\| < OFF$ where n_{shell} is the normal vector to the shell element.
BOXID	Only elements inside the box with part ID, SSID, are considered. If no ID is given all elements of part ID, SSID, 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 10.4.
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 lcoal system. This curve must be defined if LCIDM is nonzero.
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 appropiate for all problems.

Remarks:

1 The part ID must consist of 3D shell elements.

LOAD_NODE_*OPTION

Options include:

POINT

SET

Purpose: Apply a concentrated nodal force to a node or a set of nodes.

Card Format

	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.
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 19.2.

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).

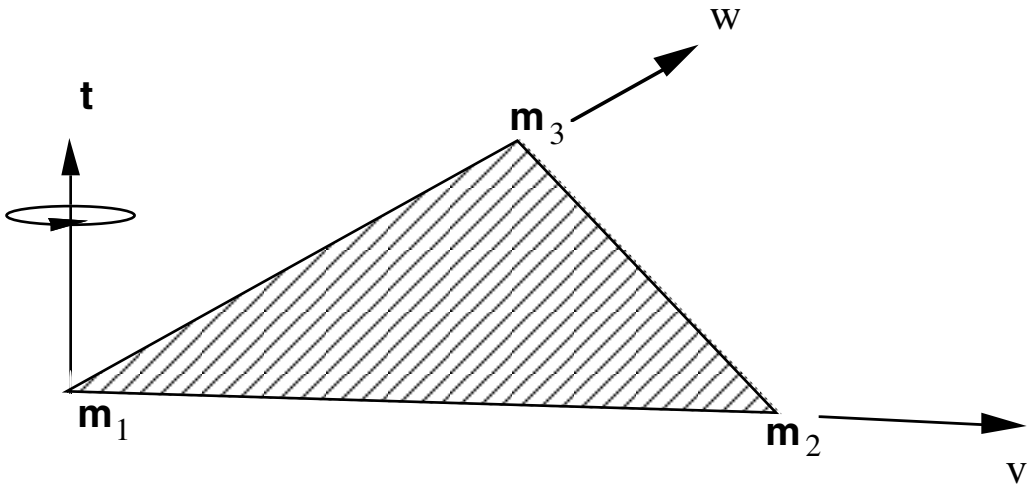


Figure 19.2. Follower force and moment acting on a plane defined by nodes m_1, m_2, and m_3. In this case, the load is applied to node m_1; i.e., m=m_1. A positive force acts in the positive t-direction, and a positive moment puts a counterclockwise torque about the normal vector. The positive t-direction 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_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 Format

	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 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, see remark 2.
LCID	Load curve ID, see *DEFINE_CURVE. GT.0: force as a function of time, LT.0: force as a function of the absolute value of the rigid body displacement.
SF	Load curve scale factor
CID	Coordinate system ID
M1	Node 1 ID. Only necessary if DOF.EQ.4 or 8, see remark 2 on next page.

VARIABLE	DESCRIPTION
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 19.2. 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.

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$   *LOAD_RIGID_BODY
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  From a sheet metal forming example.  A blank is hit by a punch, a binder is
$  used to hold the blank on its sides.  The rigid holder (part 27) is held
$  against the blank using a load applied to the cg of the holder.
$
$  The direction of the load is in the y-direction (dof=2) but is scaled
$  by sf = -1 so that the load is in the correct direction.  The load
$  is defined by load curve 12.
$
*LOAD_RIGID_BODY
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$      pid       dof      lcid        sf       cid        m1        m2        m3
        27         2        12      -1.0
$
$
*DEFINE_CURVE
$     lcid      sidr      scla      sclo      offa      offo
        12
$
$          abscissa           ordinate
          0.000E+00          8.000E-05
          1.000E+04          8.000E-05
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

*LOAD_SEGMENT

Purpose: Apply the distributed pressure load over one triangular or quadrilateral segment defined by four nodes. The pressure convention follows Figure 19.3.

Card Format

	1	2	3	4	5	6	7	8
Variable	LCID	SF	AT	N1	N2	N3	N4	
Type	I	F	F	I	I	I	I	
Default	none	1.	0.	none	none	none	none	
Remarks	1	2	3	4				

VARIABLE	DESCRIPTION
LCID	Load curve ID, see *DEFINE_CURVE.
SF	Load curve scale factor
AT	Arrival time for pressure or birth time of pressure.
N1	Node Number
N2	Node Number
N3	Node Number. Repeat N2 for two dimensional geometries.
N4	Node Number. Repeat N2 for two dimensional geometries.

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.

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$  *LOAD_SEGMENT
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  A block of solid elements is pressed down onto a plane as it moves along
$  that plane.  This pressure is defined using the *LOAD_SEGMENT keyword.
$
$  The pressure is applied to the top of the block.  This top is defined
$  by the faces on top of the appropriate solid elements.  The faces are
$  defined with segments.  For example, nodes 97, 106, 107 & 98 define
$  a top face on one of the solids (and thus, one of the faces to apply the
$  pressure too).  This "face" is referred to as a single segment.
$
$  The load is defined with load curve number 1.  The curve starts at zero,
$  ramps to 100 in 0.01 time units and then remains constant.  However,
$  the curve is then scaled by sclo = 2.5.  Thus, raising the load to 250.
$  Note that the load is NOT scaled in the *LOAD_SEGMENT keyword, but
$  could be using the sf variable.
$
*LOAD_SEGMENT
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$     lcid        sf        at        n1        n2        n3        n4
         1      1.00       0.0        97       106       107        98
         1      1.00       0.0       106       115       116       107
         1      1.00       0.0        98       107       108        99
         1      1.00       0.0       107       116       117       108
$
$
*DEFINE_CURVE
$
$     lcid      sidr      scla      sclo      offa      offo
         1         0       0.0       2.5
$
$           abscissa           ordinate
              0.000                0.0
              0.010              100.0
              0.020              100.0
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

*LOAD_SEGMENT_SET

Purpose: Apply the distributed pressure load over each segment in a segment set. The pressure convention follows Figure 19.3.

Card Format

	1	2	3	4	5	6	7	8
Variable	SSID	LCID	SF	AT				
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.
SF	Load curve scale factor
AT	Arrival time for pressure or birth time of pressure.

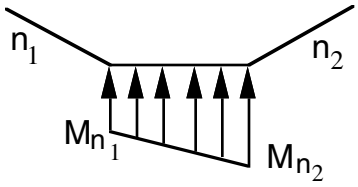
Remarks:

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

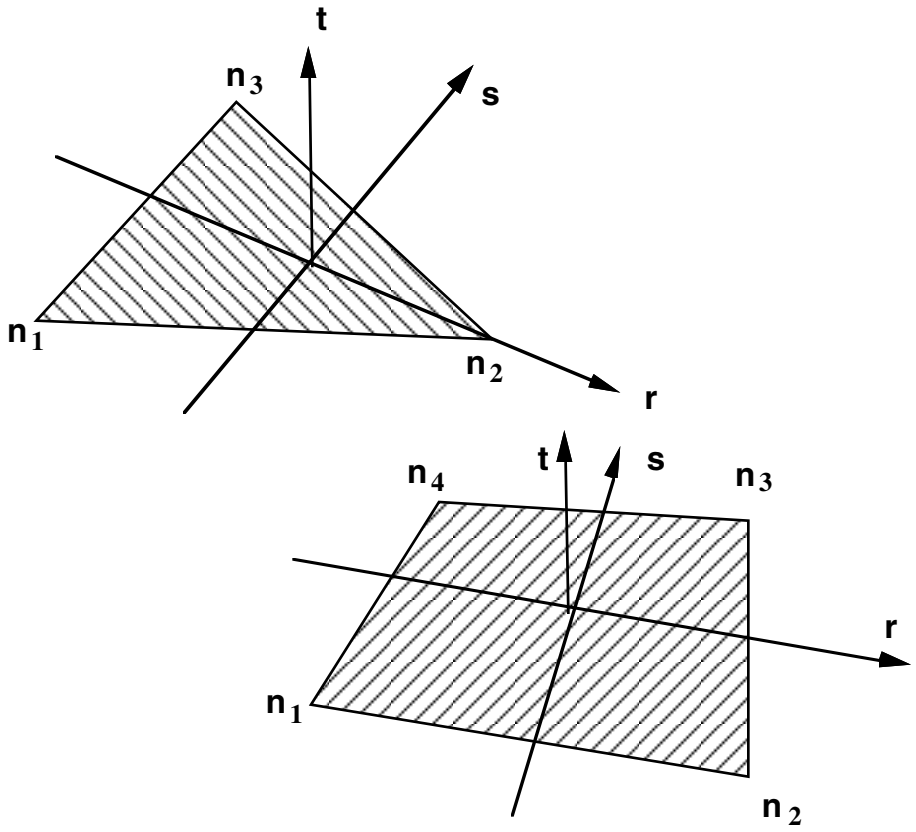


Figure 19.3. Nodal numbering for pressure cards. Positive pressure acts in the negative t-direction. For two dimensional problems repeat the second node for the third and fourth nodes in the segment definitions.

***LOAD_SHELL**_*OPTION*

Options include:

ELEMENT

SET

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 19.3. This option applies to the three-dimensional shell elements only.

Card Format

	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.
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	1	2	3	4	5	6	7	8
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 there outward normals pointing into the fluid. See Figure 19.4. 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
W	Weight of charge

VARIABLE	DESCRIPTION
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.

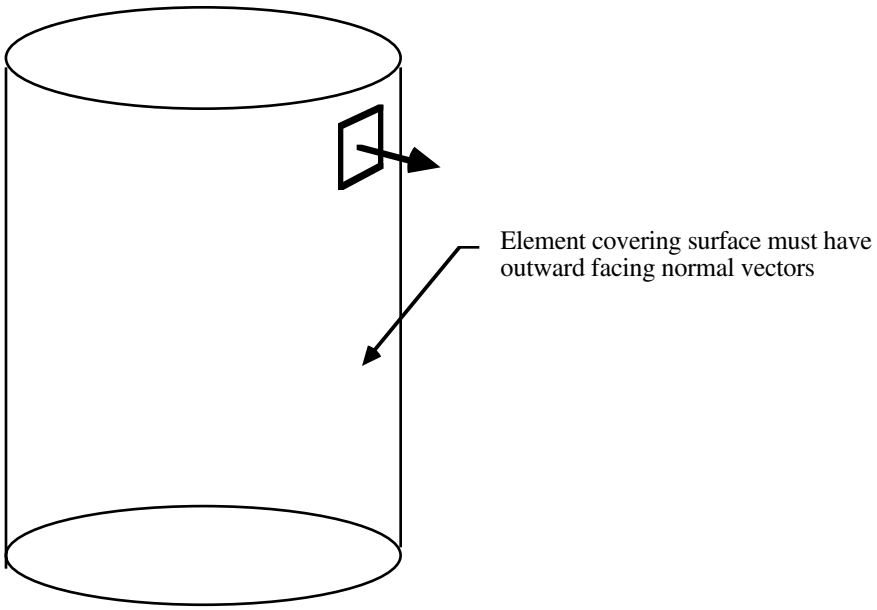


Figure 19.4. The shell elements interacting with the fluid must be numbered such that their outward normal vector points into the fluid media.

*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.

Card Format

	1	2	3	4	5	6	7	8
Variable	LCP1	CSP1	NCP1	LCP2	CSP2	NCP2		
Type	I	I	F	I	I	F		
Default	none	none	none.	none	none	none		
Remarks				1	1	1		

	1	2	3	4	5	6	7	8
Variable	ERATE	SCMIN	SCMAX	NCYL				
Type	F	F	F	I				
Default	none	none	none.	0				
Remarks				2				

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.
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.

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. The output files named: “pressure”, “curve1”, and “curve2”, may be ploted by LS-TAURUS in PHS3 using the SUPERPL command. 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.

LOAD_THERMAL_*OPTION

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 Format

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_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 Format

	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.

Remark:

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 Format

	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 INTRODUCTION.

*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 Format

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.

Remark:

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 are 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 Format

	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:

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

T_{base} is the base temperature

*NODE

Two keywords are defined in this section.

***NODE**

***NODE_RIGID_SURFACE**

*NODE

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.

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
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.

VARIABLE	DESCRIPTION
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_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.

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

*PART

Three keywords are used in this section.

***PART_{*OPTION1*}_{*OPTION2*}_{*OPTION3*}_{*OPTION4*}**

***PART_MODES**

***PART_MOVE**

*PART

***PART**_{*OPTION1*}_{*OPTION2*}_{*OPTION3*}_{*OPTION4*}

For *OPTION1* the available choices are

<BLANK>

INERTIA

REPOSITION

For *OPTION2* the available choices are

<BLANK>

CONTACT

For *OPTION3* the available choices are

<BLANK>

PRINT

For *OPTION4* the available choices 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 component 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 setction.card.

The PRINT option allows user control over whether output data is written into the ASCII files MATSUM and RBDOUT. See *DATABASE_ASCII.

Card Format

Card 1

Variable	HEADING	
Type	C	
Default	none	
Remarks	1	

Card 2

Variable	PID	SECID	MID	EOSID	HGID	GRAV	ADPOPT	TMID
Type	I	I	I	I	I	I	I	I
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	1	2	3	4	5	6	7	8
Variable	IXX	IXY	IXZ	IYY	IYZ	IZZ		
Type	F	F	F	F	F	F		

Card 5	1	2	3	4	5	6	7	8
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.
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	

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 a 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: EQ.0: no adaptivity, EQ.1: H-adaptive for 3-D shells. EQ.2: R-adaptive remeshing for 2-D shells.

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
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
IYZ	I_{yz}, yz component of inertia tensor (see Remark 4)
IZZ	I_{zz}, zz component of inertia tensor
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
VRY	initial rotational velocity of rigid body about y axis
VRZ	initial rotational velocity of rigid body about z axis

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 vector in local x-y plane
YLIP	y-coordinate of vector in local x-y plane
ZLIP	z-coordinate of vecotr 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 $\mu_c = FD + (FS - FD)e^{-DC\cdot\lvert v_{rel}\rvert}$.
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.

VARIABLE	DESCRIPTION
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. In case of SMUG post processing place PSHELL (or PBAR, or PSOLID) in columns 1-8 and Property name in columns 34-41.

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 specifiying 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. Note that the off-diagonal terms of the inertia tensor are opposite in sign from the products of inertia.

*PART_MODES

Purpose: Define mode shapes for a flexible rigid body. Currently, flexible bodies cannot be merged into other flexible bodies or rigid bodies; however, interconnections to other rigid/flexible 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 recommeded. A lump mass matrix is assumed in the implementation. Also see the keyword control card: *CONTROL_RIGID.

Card Format

Card 1

Variable	PID	NMFB	FORM	ANSID	FORMAT	KMFLAG	NUPDF	SIGREC
Type	I	I	I	I	I	I	I	

Card 2

Variable	FILENAME
Type	C
Default	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, ...

Variable	MODE1	MODE2	MODE3	MODE4	MODE5	MODE6	MODE7	MODE8
Type	I	I	I	I	I	I	I	I
Default	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	nont	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: LSTC eigout file 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.
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.
NUPDF	Nodal update flag. If active, an attachment node set, ANSID, must be defined. EQ.0: all nodes of 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.

VARIABLE	DESCRIPTION
SIGREC	Stress recovery flag. If active, attachment nodes should not be used. EQ.0: no stress recovery EQ.1: recover stresses.
FILENAME	The path and name of a file which containes 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_MOVE**

Purpose: Translate shell part by an increment. This option currently applies only to shell elements.

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				
Type	I	F		F		F				
Default	none	0.		0.		0.				

VARIABLE	DESCRIPTION
PID	Part identification
XMOV	Move shell part ID, PID, in the x-direction by the incremental distance, XMOV.
YMOV	Move shell part ID, PID, in the y-direction by the incremental distance, YMOV.
ZMOV	Move shell part ID, PID, in the z-direction by the incremental distance, ZMOV.

*RIGIDWALL

Two keywords are used in this section to define rigid surfaces:

***RIGIDWALL_GEOMETRIC_*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

***RIGIDWALL_GEOMETRIC**_*OPTION*_{*OPTION*}

Available forms include (one is mandatory):

RIGIDWALL_GEOMETRIC_FLAT

RIGIDWALL_GEOMETRIC_PRISM

RIGIDWALL_GEOMETRIC_CYLINDER

RIGIDWALL_GEOMETRIC_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**

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.

Card Format:

- 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					
Type	I	I	I					
Default	none	0	0					
Remarks								

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.

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 22.1.
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 22.1. 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 22.1.
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 22.1.

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 22.1.
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 valure defines an infinite size plane.
LENM	Length of **m** edge. A zero valure defines an infinite size plane.
LENP	Length of prism in the direction negative to **n**, see Figure 22.1.

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 22.1.

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 22.1. Only if a valure 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 22.1.

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

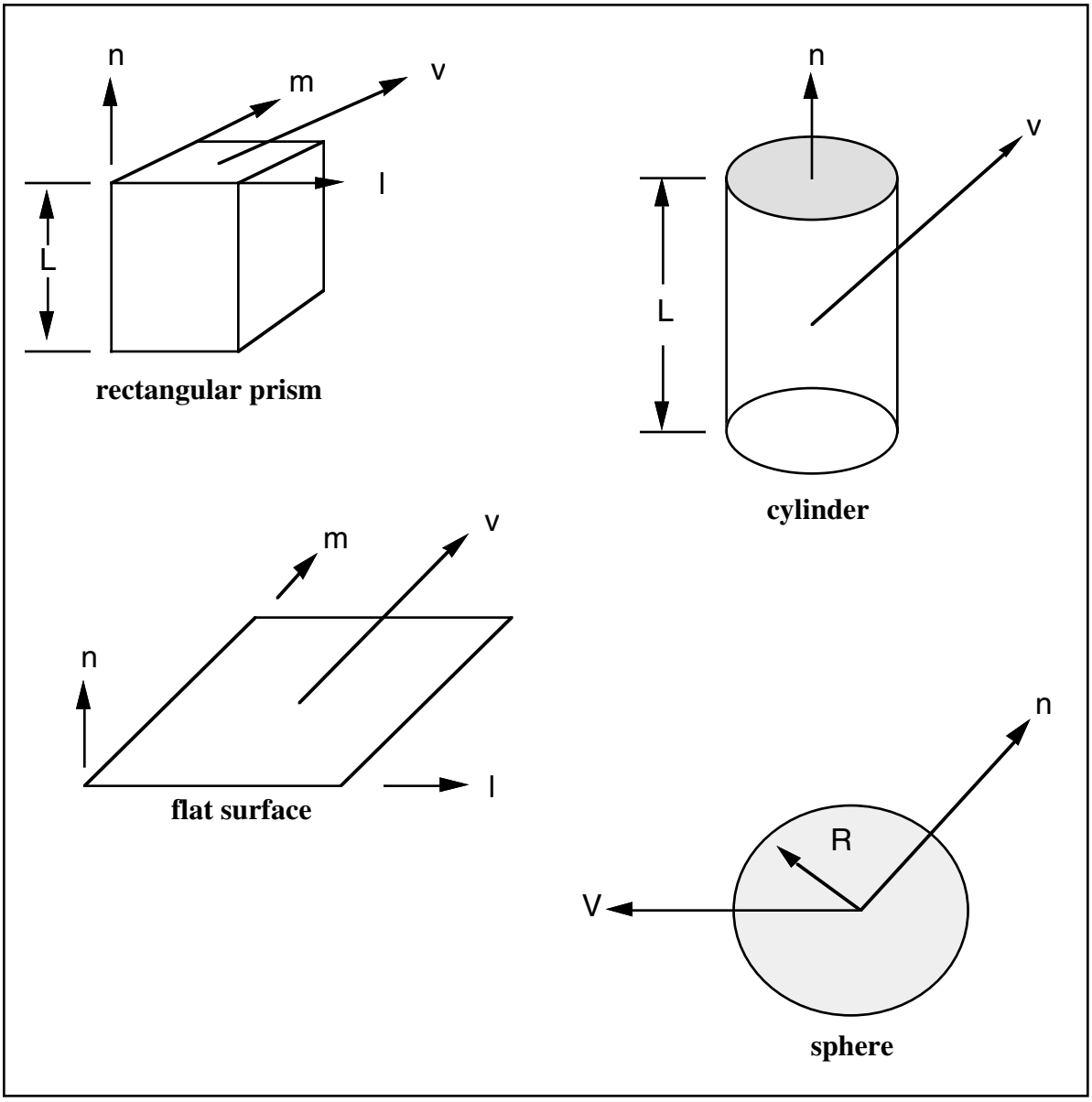


Figure 22.1. Vector **n** determines the orientation of the generalized stonewalls. For the prescribed motion options the wall can be moved in the direction **V** as shown.

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$  *RIGIDWALL_GEOMETRIC_SPHERE_MOTION
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Define a rigid sphere:
$     - with a radius of 8
$     - centered at (x,y,z) = (20,20,9)
$     - that moves in the negative z-direction with a specified displacement
$          given by a load curve (load curve: lcid = 5)
$     - which prevents all nodes within a specified box from penetrating the
$          sphere (box number: boxid = 3), these nodes can slide on the sphere
$          without friction
$
*RIGIDWALL_GEOMETRIC_SPHERE_MOTION
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$     nsid    nsidex     boxid
                             3
$
$       xt        yt        zt        xh        yh        zh      fric
      20.0      20.0       9.0      20.0      20.0       0.0       0.0
$
$   radsph
       8.0
$
$     lcid       opt        vx        vy        vz
         5         1       0.0       0.0      -1.0
$
$
*DEFINE_BOX
$    boxid       xmn       xmx       ymn       ymx       zmn       zmx
         3       0.0      40.0       0.0      40.0      -1.0       1.0
$
$
*DEFINE_CURVE
$     lcid      sidr      scla      sclo      offa      offo
         5
$           abscissa            ordinate
                 0.0                 0.0
              0.0005                15.0
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

*RIGIDWALL

*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 preceed the **MOVING** definition card, and the **FORCES** definition card should be last. The **ORTHO** option does not apply if the **MOVING** option is used.

Purpose: Define planar rigid walls with either finite or infinte 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.

Card Format:

- 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.

Card 1 - Required.

Card 1	1	2	3	4	5	6	7	8
Variable	NSID	NSIDEX	BOXID	OFFSET				
Type	I	I	I	F				
Default	none	0	0	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.

Card 2 - 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 22.3.
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 23.2 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	1	2	3	4	5	6	7	8
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 22.2
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 22.2. With the node definition the direction changes if the nodal pair rotates.
NODE2	Node 2

VARIABLE	DESCRIPTION
D1	d_1, x-component of vector, alternative to definition with nodes above, see Figure 23.2. This vector is fixed as a funtion of time.
D2	d_2, y-component of vector
D3	d_3, z-component of vector

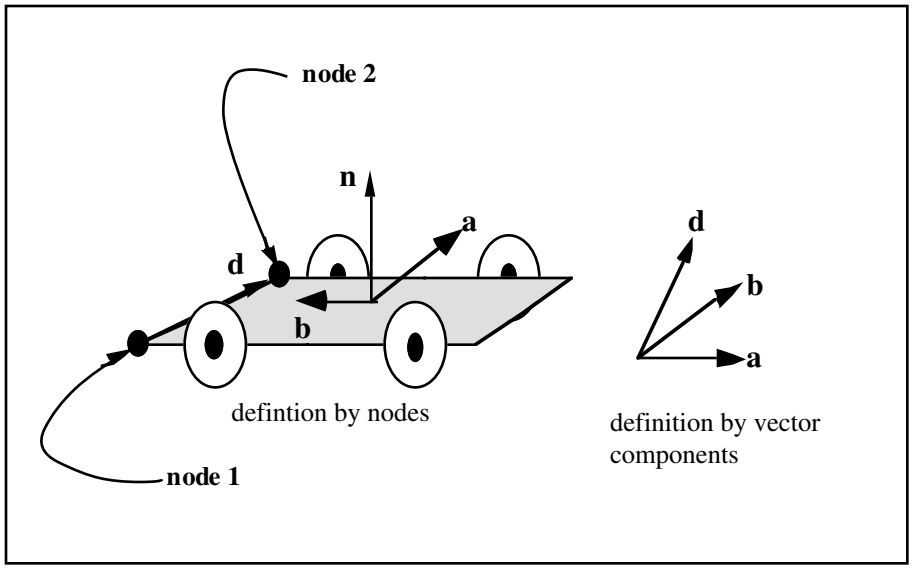


Figure 22.2. Definition of orthotropic friction vectors. The two methods of defining the vector, **d** are shown. If vector **d** is defined by nodes 1 and 2, the local coordinate system may rotate with the body which contains the nodes; otherwise, **d** is fixed in space, thus on the rigid wall, and 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 23.3. 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 22.3.
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-POST.

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	Optional nodal point for visualization in LS-DYNA database, see remark 2 below.
N2-N4	Optional nodal points 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-POST 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.

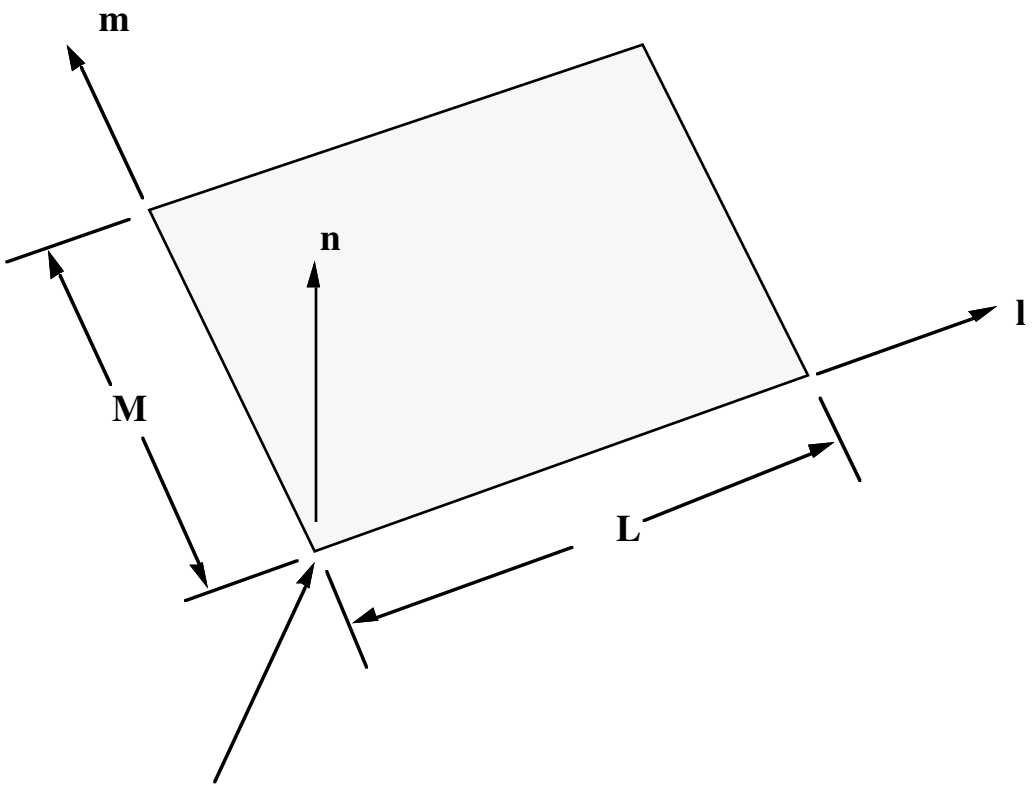


Figure 22.3. Vector **n** is normal to the stonewall. An optional vector **l** can be defined such that **m**=**n**× **l**. The extent of the stonewall is limited by defining **L** (LENL) and **M** (LENM). A zero value for either of these lengths indicates that the stonewall is infinite in that direction.

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$   *RIGIDWALL_PLANAR_MOVING_FORCES
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Define a moving planar rigid wall:
$     - that is parallel to the y-z plane starting at x = 250 mm
$     - with an initial velocity of 8.94 mm/ms in the negative z-direction
$     - that has a mass of 800 kg
$     - which prevents all nodes in the model from penetrating the wall
$     - with a friction coefficint for nodes sliding along the wall of 0.1
$     - track the motion of the wall by creating a node (numbered 99999)
$         at the tail of the wall and assigning the node to move with the wall
$
*RIGIDWALL_PLANAR_MOVING_FORCES
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$     nsid    nsidex     boxid
         0         0         0
$
$       xt        yt        zt        xh        yh        zh      fric
     250.0       0.0       0.0       0.0       0.0       0.0       0.1
$
$  SW mass    SW vel
    800.00      8.94
$
$     soft      ssid     node1     node2     node3     node4
         0         0     99999
$
$
*NODE
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$    nid               x               y               z      tc      rc
   99999           250.0             0.0             0.0       0       0
$
$
*DATABASE_HISTORY_NODE
$   Define nodes that output into nodout
$      id1       id2       id3
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
     99999
$
*DATABASE_NODOUT
$       dt
       0.1
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

*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_BEAM**

***SECTION_DISCRETE**

***SECTION_SEATBELT**

***SECTION_SHELL**_{*OPTION*}

***SECTION_SOLID**_{*OPTION*}

***SECTION_SPH**

***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

*SECTION_BEAM

Purpose: Define cross sectional properties for beam, truss, discrete beam, and cable elements.

Card Format

Card 1

Variable	SECID	ELFORM	SHRF	QR/IRID	CST	SCOOR		
Type	I	I	F	F	F	F		
Default	none	1	1.0	2.0	0.0	0.0		

Define the appropriate card format depending on the value of ELFORM (1-9) above.

Card 2

Integrated spotweld 1,4,5,7,8,9	TS1	TS2	TT1	TT2	NSLOC	NTLOC		
Resultant 2,3	A	ISS	ITT	IRR	SA			
Discrete 6	VOL	INER	CID	CA	OFFSET	RRCON	SRCON	TRCON
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
SECID	Section ID. SECID is referenced on the *PART card and must be unique.
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.

VARIABLE	DESCRIPTION
	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.
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, 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 infinitestimal separation between the nodes in the deformed state. For large separations or nonzero length beams set |SCOOR| to 2 or 3.*

VARIABLE	DESCRIPTION
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.
TT1	Beam thickness (CST=0.0, 2.0) or inner diameter (CST = 1.0) in t direction at node n_1.
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: 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: 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 23.1.
ISS	I_{ss}. The definition on *ELEMENT_BEAM_THICKNESS overrides the value defined here, see Figure 23.1.
ITT	I_{tt}. The definition on *ELEMENT_BEAM_THICKNESS overrides the value defined here, see Figure 23.1.
IRR	I_{rr} (J) polar inertia. The definition on *ELEMENT_BEAM_THICKNESS overrides the value defined here, see Figure 23.1. If IRR is zero, then IRR is reset to the sum of ISS+ITT as an approximation.
SA	Shear area. The definition on *ELEMENT_BEAM_THICKNESS overrides the value defined here, see Figure 23.1.
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.

VARIABLE	DESCRIPTION
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, materials type ID (66-69, 93 and 95), see *DEFINE_COORDINATE_SYSTEM. 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

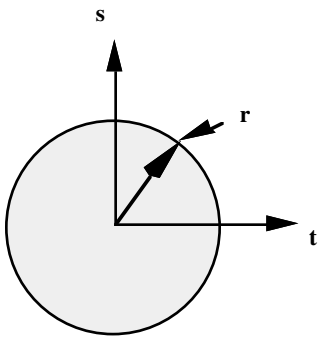
Remarks:

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_SYSTEM_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.

```
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$  *SECTION_BEAM
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Define a Belytschko-Schwer resultant beam (elform = 2) with the following
$  properties.  This beam models the connection/stiffening beams of a medium
$  size roadside sign.
$
$    cross sectional area:                        a =     515.6 mm2
$    2nd moment of area about s-axis:           iss =  99,660.0 mm4
$    2nd moment of area about t-axis:           iss =  70,500.0 mm4
$    2nd polar moment of area about beam axis: irr = 170,000.0 mm4
$
*SECTION_BEAM
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$      sid    elform      shrf   qr/irid      cst
       111         2
$
$        a       iss       itt       irr        sa
     515.6   99660.0   70500.0  170000.0
$
*SECTION_BEAM_TITLE
    Main beam member
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$      sid    elform      shrf   qr/irid      cst
       111         2
$
$        a       iss       itt       irr        sa
     515.6   99660.0   70500.0  170000.0
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

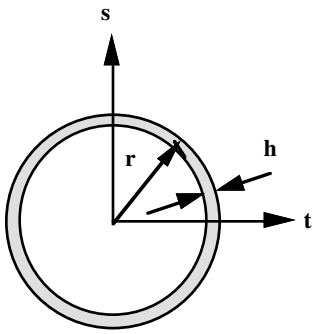


$$I_{tt} = I_{ss} = \frac{\pi r^4}{4}$$

$$J = \frac{\pi r^4}{2}$$

$$f_{tt} = f_{ss} = \frac{10}{9}$$

$$A = \pi r^2$$

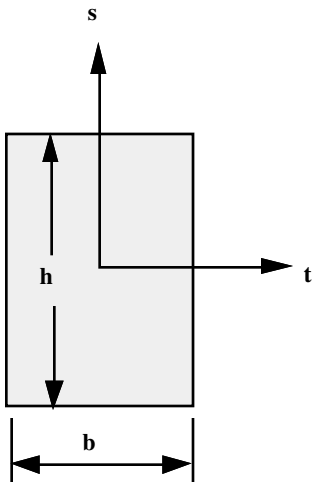


$$I_{tt} = I_{ss} \cong \pi r^3 h$$

$$J \cong 2\pi r^3 h$$

$$f_{tt} = f_{ss} = 2$$

$$A \cong 2\pi r h$$



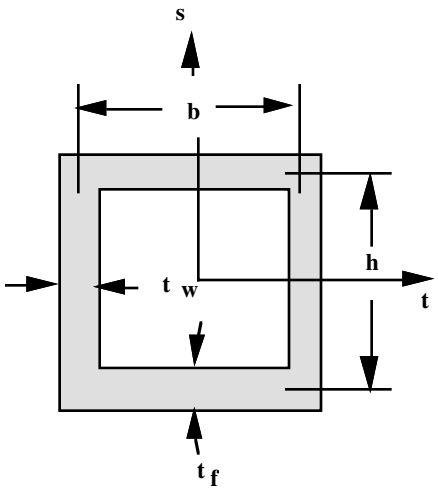
$$I_{tt} = \frac{bh^3}{12}$$

$$I_{ss} = \frac{b^3 h}{12}$$

$$J \cong \left[\frac{1}{3} - .21\left(\frac{b}{h}\right)\left(1 - \frac{b^4}{12h^4}\right)\right] hb^3$$

$$f_{tt} = f_{ss} = \frac{6}{5}$$

$$A = bh$$



$$I_{tt} \cong \left(\frac{h^2}{6}\right)(ht_w + 3bt_f)$$

$$I_{ss} \cong \left(\frac{b^2}{6}\right)(bt_f + 3ht_w)$$

$$J \cong \frac{2b^2h^2(t_w t_f)}{(bt_w + ht_f)}$$

$$f_{tt} = \frac{A}{[2(b + t_w)t_f]}$$

$$f_{ss} = \frac{A}{[2(h + t_f)t_w]}$$

$$A = 2(bt_f + ht_w)$$

$$\text{Shear Area} = \frac{A}{f} = \mu A$$

Figure 23.1. Properties of beam cross section for several common cross sections.

*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 Format

Card 1	1	2	3	4	5	6	7	8
Variable	SECID	DRO	KD	V0	CL	FD		
Type	I	I	F	F	F	F		

Card 2	1	2	3	4	5	6	7	8
Variable	CDL	TDL						
Type	F	F						

VARIABLE	DESCRIPTION
SECID	Section ID. SECID is referenced on the *PART card and must be unique.
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).
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_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 Format

Card 1	1	2	3	4	5	6	7	8
Variable	SECID							
Type	I							

VARIABLE	DESCRIPTION
SECID	Section ID

Remarks:

1. Seatbelt elements are not implicit 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*}

Options include:

<BLANK>

ALE

such that the keyword cards appear:

***SECTION_SHELL**

***SECTION_SHELL_ALE**

Purpose: Define section properties for shell elements.

Card Format

Card 1

1	2	3	4	5	6	7	8
SECID	ELFORM	SHRF	NIP	PROPT	QR/IRID	ICOMP	SETYP
I	I	F	F	F	F	I	I
none	0	1.0	2	0.0	0.0	0	1
	1						

Card 2

1	2	3	4	5	6	7	8
T1	T2	T3	T4	NLOC	MAREA		
F	F	F	F	F	F		
0.0	0.0	0.0	0.0	0.0	0.0		

Optional Section Cards if ICOMP=1. Define NIP angles putting 8 on each card.

Cards 3,4,..	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

Optional Section Card for ALE option.

Also see *CONTROL_ALE and *ALE_SMOOTHING.

Card 3	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

VARIABLE	DESCRIPTION
SECID	Section ID. SECID is referenced on the *PART card and must be unique.
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, 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 C0 shell (See remarks). EQ.31: 1 point Eulerian Navier-Stokes EQ.32: 8 point Eulerian Navier-Stokes

VARIABLE	DESCRIPTION
	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 corection 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 used in some of the constitutive model.
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. 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 11-15 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: 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/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.
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 (Hughes-Liu shell only): 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.
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
.	.
.	.
.	.
B8	β_8, material angle at eigth integration point
.	.
Bnip	β_{nip}, material angle at nipth integration point

VARIABLE	DESCRIPTION
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

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	-.9602896	-.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			+.9602896	+.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. Element formulations 31 and 32 are used exclusively with the CFD option which requires ISOLTYP=4 on the *CONTROL_SOLUTION card. In this case, ELFORM=31 is used with INSOL=1 and ELFORM=32 is used with INSOL=3 on the *CONTROL_CFD_GENERAL card. Note that selection of the element formulation is automatic based on the value of INSOL for the CFD solver.

2. For implicit calculations the following element choices are implemented:
   EQ.1: Hughes-Liu,
   EQ.2: Belytschko-Tsay (default),
   EQ.6: S/R Hughes-Liu ,
   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.31: 1 point Eulerian Navier-Stokes
EQ.32: 8 point Eulerian Navier-Stokes

If another element formulation is requested, LS-DYNA will substitute one of the above in place of the one chosen.

3. 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.

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$   *SECTION_SHELL
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Define a shell section that specifies the following:
$     elform = 10  Belytschko-Wong-Chiang shell element formulation.
$        nip = 3   Three through the shell thickness integration points.
$    t1 - t4 = 2.0 A shell thickness of 2 mm at all nodes.
$
*SECTION_SHELL
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$      sid    elform      shrf       nip     propt   qr/irid     icomp
         1        10              3.0000
$
$       t1        t2        t3        t4      nloc
       2.0       2.0       2.0       2.0
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

*SECTION

***SECTION_SOLID_**{*OPTION*}

Options include:

<BLANK>

ALE

such that the keyword cards appear:

***SECTION_SOLID**

***SECTION_SOLID_ALE**

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	I	I	I					
Remark		1, 2						

Card 2 define only for the ALE option.

Also see *ALE_SMOOTHING for the smoothing definition.

Cards 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	

VARIABLE	DESCRIPTION
SECID	Section ID. SECID is referenced on the *PART card and must be unique.
ELFORM	Element formulation options, (see remark 3 below): EQ.0: 1 point corotational for *MAT_MODIFIED_HONEYCOMB. See remark 4 below. EQ.1: constant stress solid element (default), EQ.2: fully integrated S/R solid. See remark 5 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 4 below. 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 for bulk forming. EQ.14: 8 point acoustic EQ.15: 2 point pentahedron element. EQ.18: 8 point enhanced strain solid element for linear statics only, EQ.31: 1 point Eulerian Navier-Stokes EQ.32: 8 point Eulerian Navier-Stokes
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)
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

Remarks:

1. Element formulations 31 and 32 are used exclusively with the CFD option which requires ISOLTYP=4 on the *CONTROL_SOLUTION card. In this case, ELFORM=31 is used with INSOL=1 and ELFORM=32 is used with INSOL=3 on the *CONTROL_CFD_GENERAL card. Note that selection of the element formulation is automatic based on the value of INSOL for the CFD solver.

2. 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.

3. 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.10: 1 point tetrahedron.
   EQ.15: 2 point pentahedron element.
   EQ.18: 8 point enhanced strain solid element for linear statics only,
   EQ.31: 1 point Eulerian Navier-Stokes
   EQ.32: 8 point Eulerian Navier-Stokes

   If another element formulation is requested, LS-DYNA will substitute, when possible, one of the above in place of the one chosen.

4. 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.

5. 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 strain formulations.

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$   *SECTION_SOLID
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  A bolt modeled with solids was found to have excessive hourglassing.
$  Thus, the section (sid = 116) associated with the bolt part was used
$  to specify that a fully integrated Selectively-Reduced solid element
$  formulation be used to totally eliminate the hourglassing (elform = 2).
$
*SECTION_SOLID
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$      sid    elform
       116         2
$
*PART
bolts
$      pid       sid       mid     eosid      hgid    adpopt
        17       116         5
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

*SECTION

*SECTION_SPH

Purpose: Define section properties for SPH particles.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	SECID	CSLH	HMIN	HMAX				
Type	I	F	F	F				
Default	none	1.2	0.2	2.0				

VARIABLE	DESCRIPTION
SECID	Section ID. SECID is referenced on the *PART card and must be unique.
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)

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.

*SECTION_TSHELL

Purpose: Define section properties for thick shell elements.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	SECID	ELFORM	SHRF	NIP	PROPT	QR	ICOMP	
Type	I	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.

Cards 2,3,..	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 and must be unique.
ELFORM	Element formulation: EQ.1: one point reduced integration (default), EQ.2: selective reduced 2×2 in plane integration. EQ.3: assumed strain 2×2 in plane integration, see remark below.
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 eigth 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 type 3 uses a full three-dimensional stress update rather than the two-dimensional plane stress update of types 1 and 2. The type 3 element is 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 type 3 two are recommended to avoid excessive softness.

2. These elements are available for implicit applications.

*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 condtions, 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*

***SET_DISCRETE**_{*OPTION*}

***SET_NODE**_*OPTION*

***SET_PART**_*OPTION*

***SET_SEGMENT**_{*OPTION*}

***SET_SHELL**_*OPTION*

***SET_SOLID**_{*OPTION*}

***SET_TSHELL**_{*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.

SET_BEAM_*OPTION

Available options include:

GENERATE

GENERAL

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 Format

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.)

	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.)

	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.

	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_DISCRETE_{*OPTION*}**

Available options include:

<BLANK>

GENERATE

GENERAL

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 Format

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.)

	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.)

	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.

	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_NODE_*OPTION

Available options include:

LIST

COLUMN

LIST_GENERATE

GENERAL

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 Format

	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.			
Remark		1	1	1	1			

Cards 2, 3, 4, ... (*OPTION*=LIST) (The next "*" card terminates the input.)

	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.)

	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.)

	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.

	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

VARIABLE	DESCRIPTION
DA4	Fourth nodal attribute default value
NIDN	Node ID n
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.

SET_PART_*OPTION

Available options include:

LIST

COLUMN

LIST_GENERATE

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 Format

	1	2	3	4	5	6	7	8
Variable	SID	DA1	DA2	DA3	DA4			
Type	I	F	F	F	F			
Default	none	0.						
Remark		1	1	1	1			

Card 2, 3, 4, ... (*OPTION*=LIST) (The next "*" card terminates the input.)

	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.)

	1	2	3	4	5	6	7	8
Variable	PID	A1	A2	A3	A4			
Type	I	F	F	F	F			
Remark		1	1	1	1			

Cards 2, 3, 4, ... (*OPTION*=LIST_GENERATE) (The next "*" card terminates the input.)

	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

VARIABLE	DESCRIPTION
A4	Fourth part attribute
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.

***SET_SEGMENT_**{*OPTION*}

Available options include:

<BLANK>

GENERAL

Purpose: Define a set of quadrilateral and triangular segments with optional identical or unique attributes.

Card Format

	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			

Cards 2, 3, 4, ... (No option is specified) (The next "*" card terminates the input.)

	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.

	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.

***SET_SHELL**_*OPTION*

Available options include:

LIST

COLUMN

LIST_GENERATE

GENERAL

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 Format

	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) (The next "*" card terminates the input.)

	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.)

	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.)

	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.

	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

VARIABLE	DESCRIPTION
EID1	First shell element ID, see remark 2.
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_SOLID_**{*OPTION*}

Available options include:

<BLANK>

GENERATE

GENERAL

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 Format

	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.)

	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.)

	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.

	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.

VARIABLE	DESCRIPTION
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.

***SET_TSHELL_{*OPTION*}**

Available options include:

<BLANK>

GENERATE

GENERAL

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 Format

	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.)

	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.)

	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.

	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.

VARIABLE	DESCRIPTION
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_NODE**

***TERMINATION_BODY**

***TERMINATION_CONTACT**

*TERMINATION

*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 (stop 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 Format

	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_BODY

Purpose: Terminate calculation based on rigid body displacements. For *TERMINATION_BODY the analysis terminates when the centre 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 centre 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 Format

	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 Format

	1	2	3	4	5	6	7	8
Variable	CID	ACTIM	DUR					
Type	I	I	F					
Default	none	none	-	-				

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.

*TITLE

***TITLE**

Purpose: Define job title.

Card Format

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.

*TRANSLATE

***TRANSLATE_ANSYS**_*OPTION*

Available options include:

4

5

corresponding to ANSYS version numbers 4 and 5.

Purpose: Provide a convenient route to read in ANSYS input decks as part of the LS-DYNA keyword input. This keyword can appear more than once anywhere in the input. It is a direct interface to ANSYS file28 keyword files.

Card Format

	1	2	3	4	5	6	7	8
Variable	FILE							
Type	A							
Default	none							

VARIABLE	DESCRIPTION
FILE	Filename of file created by ANSYS, see remarks below.

The supported options include:

Version	ANSYS Keyword	LS-DYNA Keyword
All	N,Type,NODE,Val1,Val2,Val3	*NODE
All	EN,Type,I1,I2,I3,I4,I5,I6,I7,I8	*ELEMENT
All	MPDATA, R5.0, LENGTH, Lab, MAT, STLOC, VAL1, VAL2, VAL3	*MAT_ELASTIC

*TRANSLATE

Version	ANSYS Keyword	LS-DYNA Keyword
All	ET, Type	*PART&*SECTION
All	R,R5.0,NSET,Type,STLOC,VAL1,VAL2,VAL3	*PART&*SECTION
5	DFLAB,NODF,LabD,LabF	
5	NDOF.eq.Ui,ROTi; LabD.eq.0	*BOUNDARY_SPC_*OPTION*
5	NODF.eq.Vi; LabD.eq.0	*INITIAL_VELOCITY_NODE
5	NODF.eq.Ui,ROTi,Ai,Vi,;LabD.eq.lcid;	
	LabF.eq.val	*BOUNDARY_PRESCRIBED_ MOTION_NODE
5	NDOF.eq.Fi; LabF.eq.lcid	*LOAD_NODE_POINT
5	SFE,ELEM,LKEY,Lab,KEY,R5.0	
5	LKEY.eq.lcid; Lab.eq.pressure	*LOAD_SEGMENT

Remarks:

1. Supported keywords as described in the SASI ANSYS Manual chapter on "Exporting a Finite Element Model."

2. Solid elements and shell elements cannot have the same R value in reference to the ET and R ANSYS keywords.

3. Supported element types include: 63.eq.shells, 45.eq.solids, 73.eq.solids, 4.eq.beams, 16.eq.pipes, and 21.eq.lumped masses.

***TRANSLATE_IDEAS_{*OPTION*}**

Available options include:

<BLANK>

MASTER

Purpose: Provide a convenient route to read in files created by IDEAS/SUPERTAB as part of the LS-DYNA keyword input. This keyword can appear more than once in the input. It is a direct interface to IDEAS universal files.

Card Format

	1	2	3	4	5	6	7	8
Variable	FILE							
Type	A							
Default	none							

VARIABLE	DESCRIPTION
FILE	Filename of the IDEAS universal file.

The following table lists supported IDEAS keywords:

Version	SDRC IDEAS Universal File	LS-DYNA Keyword
All	N,Type,NODE,Val1,Val2,Val3	*NODE
All	EN,Type,I1,I2,I3,I4,I5,I6,I7,I8	*ELEMENT
5	781	*NODE
MASTER	2411	*NODE
5	780	*ELEMENT
MASTER	2412	*ELEMENT
5	773	*MAT_ELASTIC
5	772	*PART&*SECTION
6	788	*PART&*SECTION

*TRANSLATE

Version	SDRC IDEAS Universal File	LS-DYNA Keyword
MASTER	2430	*PART&*SECTION
5	755	*BOUNDARY_SPC_NODE
MASTER	791	
	time variation set.le.0.0	*BOUNDARY_SPC_NODE
	time variation set.gt.0.0	*BOUNDARY_PRESCRIBED_ MOTION_NODE
MASTER	790	
	load type.eq.1	*LOAD_NODE

*TRANSLATE_NASTRAN

Purpose: Provide a convenient route to read in NASTRAN input deck as part of the LSDYNA keyword input. This keyword can appear more than once anywhere in the input. Also, see remarks below.

Card Format

	1	2	3	4	5	6	7	8
Variable	FILE							
Type	C							

VARIABLE	DESCRIPTION
FILE	Filename of the NASTRAN input deck.

The following table lists supported NASTRAN keywords:

Version	NASTRAN INPUT FILE	LS-DYNA Keyword
All	N,Type,NODE,Val1,Val2,Val3	*NODE
All	EN,Type,I1,I2,I3,I4,I5,I6,I7,I8	*ELEMENT
All	BEGIN BULK	
All	GRID	*NODE
All	CORD2R	*DEFINE_COORDINATE_SYSTEM
All	CHEXA, CPENTA, CTETRA	*ELEMENT_SOLID
All	PSOLID	*PART and *SECTION_SOLID
All	CQUAD4, CTRIA3	*ELEMENT_SHELL
All	PSHELL	*PART and *SECTION_SHELL
All	CBAR, CBEAM	*ELEMENT_BEAM
All	CELAS1, CVISC, CDAMP1	*ELEMENT_DISCRETE
All	CONM2	*ELEMENT_MASS
All	MAT1	*MAT_ELASTIC
All	SPC, SPC1	*BOUNDARY_SPC_*OPTION*

*TRANSLATE

Version	NASTRAN INPUT FILE	LS-DYNA Keyword
All	RBE2	*CONSTRAINED_NODE_SET or *CONSTRAINED_NODAL_RIGID_BODY_
All	ENDDATA	*END

Remarks:

1. Both small and large field fixed NASTRAN formats are supported.

2. The same keywords in LS-DYNA usually contain more options than the NASTRAN input. Therefore, to make it complete, we add some extra parameters to the NASTRAN keywords. For those extras we use the italics to distinguish from the standard ones. These additional parameters have to be added to the NASTRAN deck by the user to make the translation complete.

Card Format

For further explanation see *ELEMENT_DISCRETE.

	1	2	3	4	5	6	7	8	9
CELAS1	EID	PID	*N1*	*DFG*	*N2*	*VID*	*S*	*PF*	
Type	I	I	I	F	I	F	F	F	

For further explanation see *PART and *SECTION_SHELL.

PSHELL	PID	MID	*T*	*FORM*	*SHEAR*	*NIP*	*QR*	*HGID*	
Type	I	I	F	I	F	I	F	I	

3. Current **NASTRAN** only supports shell element with constant thickness T.

For further explanation see *PART and *SECTION_SOLID.

PSOLID	PID	MID	*SCID*	*EOSID*	*HGID*				
Type	I	I	I	I	I				

4. The **THRU** command for SPC, SPC1 is not supported in the current translation.

5. For RBE2 keyword, if any of the rotational DOF (4,5,6) appears in the constraint, LS-DYNA will treat it as nodal rigid body constraint. Otherwise, LS-DYNA will use nodal constraints to treat this RBE2.

*USER

USER_INTERFACE_*OPTION

Available options include:

CONTROL

FRICTION

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 D.

The **FRICTION** option may be used to modify the Coulomb friction coefficients according to contact information or to use a friction coefficient database. A sample subroutine for treating the friction in contact is provided in Appendix E.

Card Format

	1	2	3	4	5	6	7	8
Variable	IFID	NOC	NOCI					
Type	I	I	I					
Default	none	none	none					

Card Format (Use as many cards as necessary to define NOCI variables)

	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.

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.
NOCI	Initialize the first NOCI history variables in the input. NOCI must be smaller or equal to NOC.
UC1	First user defined input parameter
UC2	Second user defined input parameter
.	. .
.	. .
.	. .
UCNOCI	Last user defined input parameter
.	. .
.	. .
.	. .

*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.

Card Format (Insert as many cards as needed. The next * card terminates input.)

	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 parmeter.

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 termnination 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_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*

Available options are:

BOUNDARY_CONDITION

CONTACT_SMALL_PENETRATION

CURVE_DEFINITION

RIGID_BODY_CONSTRAINT

RIGID_BODY_STOPPER

STATUS_REPORT_FREQUENCY

THERMAL_PARAMETERS

VELOCITY

VELOCITY_NODE

VELOCITY_RIGID_BODY

VELOCITY_ZERO

Purpose: Change some solution options.

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 Format

	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 Format

	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 Format

	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 Format

	1	2	3	4	5	6	7	8
Variable	PID	TC	RT					
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_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 Formats

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 Format

	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 Format (Card 1 of 2)

	1	2	3	4	5	6	7	8
Variable	TS	DT	TMIN	TMAX	DTEMP	TSCP		
Type	I	F	F	F	F	F		

Card Format (Card 2 of 2)

	1	2	3	4	5	6	7	8
Variable	REFMAX	TOL						
Type	I	F						

VARIABLE	DESCRIPTION
TS	Thermal time step code: EQ.0: No change, EQ.1: Fixed timestep, EQ.2: variable timestep.
DT	Thermal time step on restart: EQ.0: No change.
TMIN	Minimum thermal timestep: EQ.0: No change.
TMAX	Maximum thermal timestep: EQ.0: No change.
DTEMP	Maximum temperature change in a thermal timestep: 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.

VARIABLE	DESCRIPTION
TOL	Non-linear convergence tolerance: EQ.0: No change.

The **VELOCITY_NODE** option allows the velocity of nodal points to be changed at restart. Termination of this input is when the next "*" card is read.

Card Format

	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. 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** option allows a new velocity field to be imposed at restart. Termination of this input is when the next "*" card is read.

Card Format

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

Card 2	1	2	3	4	5	6	7	8
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.

VARIABLE	DESCRIPTION
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 Format

	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 Format

	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 [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_TERMINATION

Purpose: Stop the job.

Card Format

	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 Format

	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.

Remark:

1. This a reduced version of the *CONTROL_TIMESTEP used in the initial analysis. The dummy fields are included to maintain compatability. If using free format input then a 0.0 should be entered for the dummy values.

*DAMPING_GLOBAL

Purpose: Define mass weigthed nodal damping that applies globally to the deformable nodes.

Card Format

	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 (spotwelds 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.

*RESTART

Card Format

	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.

***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 Format

	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

pose: Delete contact surfaces, 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 use their own numbering.

For **CONTACT**, **CONTACT_2DAUTO**, **ENTITY**, or **PART** option.

Card Format

	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/Part ID

For *DELETE_CONTACT a negative ID implies that the absoulute value gives the contact surface which is to be activated

For the four **ELEMENT** options. Termination of input is when the next "*" card is read.

Card Format

	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 Format

	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 Format

	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

VARIABLE	DESCRIPTION
TC	Tranlational 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.

***RIGID_DEFORMABLE**_*OPTION*

The *OPTIONS* available are:

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.

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

Card Format

	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/spotweld 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.

For the **D2R** option define the following card. Termination of this input is when the next "*" card is read.

Card Format

	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.

For the **R2D** option define the following card. Termination of this input is when the next "*" card is read.

Card Format

	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 a number of parts to be initialized on restart.

The options that are available with this kewyord are:

<BLANK>

DISCRETE

SEATBELT

*STRESS_INITIALIZATION

If this card is specified without further input then all parts in the new analysis are initialized from the corresponding part of the old analysis. Further all seatbelt and discrete parts are initialized.

If only a subset of parts are 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 Format

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 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: Stop 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 overides 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 Format

	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 Format

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.