LS-DYNA
USER'S MANUAL

Nonlinear Dynamic Analysis of Structures

May 1999
Version 950

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

Support Address:
Livermore Software Technology Corporation
97 Rickenbacker Circle
Livermore, California 94550-7612

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

Table of Contents

ABSTRACT..I.1

BACKGROUND..I.1

MATERIAL MODELS I.11

SPATIAL DISCRETIZATION..I.13

SLIDING INTERFACES I.13

INTERFACE DEFINITIONS FOR COMPONENT ANALYSIS I.14

CAPACITY I.17

CODE ORGANIZATION..I.17

SENSE SWITCH CONTROLS...I.17

PRECISION I.18

EXECUTION... I.18

VDA DATABASE I.21

MESH GENERATION I.26

POST-PROCESSING...I.26

EXECUTION SPEEDS..I.27

LS-DYNA User’s Guide 1.1

1. Title Card 1.1

2. Control Cards... 2.1

Card 1. Model Size—General.. 2.1

Card 2. Model Size—Boundary Conditions... 2.2

Card 3. Model Size—Loading.. 2.3

Card 4. Model Size—Constraints and Contact... 2.4

Card 5. Model Size—Rigid Body Parameters 2.5

Card 6. Model Size—Discrete Elements and Seat Belts.................................... 2.6

Card 7. Model Size—Output Control.. 2.8

Card 8. Computation Options—Termination... 2.9

Card 9. Computation Options—Time Step Size Control 2.10

Card 10. Computation Options—Loading..2.12

Card 11. Computation Options—Input Control 2.14

Card 12. Computation Options—Beams and Shells 2.15

Card 13. Computation Options—Material Related Input....................................2.17

Card 14. Computation Options—Damping/Dynamic Relaxation..........................2.19

Card 15. Computation Options—Contact 2.21

Card 16. Computation Options—Parallel and Subcycling 2.24

Table of Contents

Card 17. Computation Options—Coupling 2.25

Card 18. Computation Options—Output Control 2.27

Card 19. Computation Options—Output Energy 2.29

Card 20. Computation Options—LS-TAURUS Database I 2.30

Card 21. Computation Options—LS-TAURUS Database II 2.31

Card 22. Computation Options—ASCII Output Control I 2.34

Card 23. Computation Options—ASCII Output Control II 2.35

Card 24. Computation Options—ASCII Output Control III 2.36

Card 25. Computation Options—Arbitray Lagrangian Eulerian 2.39

Card 26. Computation Options—Arbitray Lagrangian Eulerian 2.40

Card 27. Thermal Boundary Conditions 2.41

Card 28. Thermal Solver and Output Controls 2.42

Card 29. Thermal Time Step Controls 2.43

Card 30. Thermal Nonlinear Problem Controls 2.44

3. Material/Part Definitions 3.1

Control Parameters for User Defined Material 3.1

Load/Table Definitions (NLCUR<0) 3.2

Default Parameters for Automatic Contact 3.2

Input Section for IARB=0 (Control Card 11, Column 15) 3.3

Part Definition Control Cards (IARB=0) 3.3

Rayleigh Damping by Material (IARB=0) 3.9

Part/Material Heading (IARB=0) 3.10

Material Parameter Cards (IARB=0) 3.10

Solid Element ALE Section Heading (IARB=0) 3.11

Solid Element ALE Section Parameter Cards (IARB=0) 3.11

Solid Element Equation of State Heading (IARB=0) 3.12

Solid Element Equation of State Parameters (IARB=0) 3.12

Structural Beam and Shell Section Heading Card 3.12

Section Parameters for Beam Elements (IARB=0) 3.13

Section Parameters for Shell Elements (IARB=0) 3.20

Material Angle cards 3.22

VDA Surface for Rigid Part (IARB=0) 3.23

Contact Entity Mesh for Rigid Part (IARB=0) 3.24

Input Section for IARB=1 (Control Card 11, Column 15) 3.28

Part Definition Control Cards (IARB=1) 3.28

Rayleigh Damping by Material (IARB=1) 3.31

Part/Material Heading (IARB=1)....3.32
Material Parameter Cards (IARB=1) 3.32
Equation of State Definitions (IARB=1)....3.33
Solid Element Equation of State Heading (IARB=1)....3.33
Solid Element Equation of State Parameters (IARB=1) 3.33
Cross Section Definitions (IARB=1)....3.34
Section Heading Card (IARB=1)....3.34
Solid Element ALE Section Card (IARB=1) 3.34
Beam Element Section Card (IARB=1) 3.35
Shell Element Section Card (IARB=1)....3.38
Material Angle Cards (IARB=1) 3.40
Part Set Definitions (IARB=1)....3.41
VDA Surface for Rigid Part (IARB=1)....3.43
Contact Entity Mesh for Rigid Part (IARB=1) 3.44

3. Constitutive Models....3.1.1m
Material Type 1 (Elastic) 3.1.1m
Material Type 2 (Orthotropic Elastic)....3.2.1m
Material Type 3 (Kinematic/Isotropic Elastic-Plastic)....3.3.1m
Material Type 4 (Thermo-Elastic-Plastic)....3.4.1m
Material Type 5 (Soil and Crushable/Non-crushable Foam) 3.5.1m
Material Type 6 (Viscoelastic) 3.6.1m
Material Type 7 (Blatz - Ko Rubber)....3.7.1m
Material Type 8 (High Explosive Burn)....3.8.1m
Material Type 9 (Null Hydrodynamics)....3.9.1m
Material Type 10 (Isotropic-Elastic-Plastic-Hydrodynamic)....3.10.1m
Material Type 11 (Temperature Dependent, Elastoplastic, Hydrodynamic).... 3.11.1m
Material Type 12 (Isotropic-Elastic-Plastic)....3.12.1m
Material Type 13 (Elastic-Plastic with Failure Model)....3.13.1m
Material Type 14 (Soil and Crushable Foam with Failure Model)....3.14.1m
Material Type 15 (Johnson/Cook Strain and Temperature Sensitive Plasticity).... 3.15.1m
Material Type 16 (Pseudo TENSOR Concrete/Geological Model)....3.16.1m
Material Type 17 (Isotropic Elastic-Plastic Oriented Crack Model)....3.17.1m
Material Type 18 (Power Law Isotropic Plasticity) 3.18.1m
Material Type 19 (Strain Rate Dependent Isotropic Plasticity)....3.19.1m
Material Type 20 (Rigid) 3.20.1m

Table of Contents

Material Type 21 (Thermal Orthotropic Elastic) 3.21.1m
Material Type 22 (Composite Damage Model) 3.22.1m
Material Type 23 (Thermal Orthotropic Elastic with 12 Curves) 3.23.1m
Material Type 24 (Piecewise Linear Isotropic Plasticity) 3.24.1m
Material Type 25 (Inviscid Two Invariant Geologic Cap Model) 3.25.1m
Material Type 26 (Metallic Honeycomb) 3.26.1m
Material Type 27 (Compressible Mooney-Rivlin Rubber) 3.27.1m
Material Type 28 (Resultant Plasticity) 3.28.1m
Material Type 29 (FORCE LIMITED Resultant Formulation) 3.29.1m
Material Type 30 (Closed-Form Update Shell Plasticity) 3.30.1m
Material Type 31 (Frazer-Nash Rubber Model) 3.31.1m
Material Type 32 (Laminated Glass Model) 3.32.1m
Material Type 33 (Barlat's Anisotropic Plasticity Model) 3.33.1m
Material Type 34 (Fabric) 3.34.1m
Material Type 35 (Kinematic/Isotropic Elastic-Plastic Green-Naghdi Rate) 3.35.1m
Material Type 36 (Barlat's 3-Parameter Plasticity Model) 3.36.1m
Material Type 37 (Transversely Anisotropic Elastic-Plastic) 3.37.1m
Material Type 38 (Blatz-Ko Compressible Foam) 3.38.1m
Material Type 39 (Transversely Anisotropic Elastic-Plastic with FLD) 3.39.1m
Material Type 40 (Nonlinear Elastic Orthotropic Material) 3.40.1m
Material Types 41-50 (User Defined Material Models) 3.41-50.1m
Material Type 42 (Planar Anisotropic Plasticity Model) 3.42.1m
Material Type 51 (Temperature and Rate Dependent Plasticity) 3.51.1m
Material Type 52 (Sandia's Damage Model) 3.52.1m
Material Type 53 (Low Density Closed Cell Polyurethane Foam) 3.53.1m
Material Type 54 and 55 (Composite Damage Model) .... 3.54&55.1m
Material Type 57 (Low Density Urethane Foam) 3.57.1m
Material Type 59 (Composite Failure Model - Plasticity Based) 3.59.1m
Material Type 60 (Elastic With Viscosity) 3.60.1m
Material Type 61 (Maxwell/Kelvin Viscoelastic) 3.61.1m
Material Type 62 (viscous foam) 3.62.1m
Material Type 63 (Isotropic Crushable Foam) 3.63.1m
Material Type 64 (Strain Rate Sensitive Power-Law Plasticity) 3.64.1m
Material Type 65 (Modified Zerilli/Armstrong) 3.65.1m
Material Type 66 (Linear Stiffness/Linear Viscous 3D Discrete Beam) 3.66.1m
Material Type 67 (Nonlinear Stiffness/Viscous 3D Discrete Beam) 3.67.1m

Material Type 68 (Nonlinear Plastic/Linear Viscous 3D Discrete Beam)............ 3.68.1m
Material Type 69 (Side Impact Dummy Damper, SID Damper)........................ 3.69.1m
Material Type 70 (Hydraulic/Gas Damper)... 3.70.1m
Material Type 71 (Cable).. 3.71.1m
Material Type 72 (Concrete Damage Model)... 3.72.1m
Material Type 73 (Low Density Viscoelastic Foam).................................... 3.73.1m
Material Type 75 (Bilkhu/Dubois Foam Model)... 3.75.1m
Material Type 76 (General Viscoelastic)... 3.76.1m
Material Type 77 (Hyperviscoelastic Rubber).. 3.77.1m
Material Type 78 (Soil/Concrete) 3.78.1m
Material Type 79 (Hysteretic Soil)... 3.79.1m
Material Type 80 (Ramberg-Osgood Plasticity).. 3.80.1m
Material Type 81 (Plastic With Damage) 3.81.1m
Material Type 83 (Fu-Chang's Foam with Rate Effects) 3.83.1m
Material Type 86 (Orthotropic-Viscoelastic) 3.86.1m
Material Type 87 (Cellular Rubber) 3.87.1m
Material Type 88 (MTS Model) 3.88.1m
Material Type 90 (Acoustic)... 3.90.1m
Material Type 90 (Acoustic)... 3.90.1m
Material Type 96 (Brittle Damage Model) 3.96.1m
Material Type 103 (Anisotropic Viscoplastic)..3.103.1m
Material Type 126 (Metallic Honeycomb)..3.126.1m
Material Type 134 (Viscoelastic Fabric)..3.134.1m
Equation-of-State Form 1 (Linear Polynomial)... 3.1.1e
Equation-of-State Form 2 (JWL) 3.2.1e
Equation-of-State Form 3 (Sack "Tuesday").. 3.3.1e
Equation-of-State Form 4 (Gruneisen) 3.4.1e
Equation-of-State Form 5 (Ratio of Polynomials).. 3.5.1e
Equation-of-State Form 6 (Linear Polynomial with Energy Leak)....................... 3.6.1e
Equation-of-State Form 7 (Ignition and Growth)... 3.7.1e
Equation-of-State Form 8 (Tabulated-Compaction).. 3.8.1e
Equation-of-State Form 9 (Tabulated)... 3.9.1e
Equation-of-State Form 10 (Propellant-Deflagration)......................................3.10.1e
Equation-of-State Form 11 (TENSOR Pore Collapse) 3.11.1e
Material Property Data Cards(Thermal)...3.3.1t
Material Type 1 - Isotropic 3.3.2t

Table of Contents

Material Type 2 - Orthotropic 3.3.3t

Material Type 3 - Isotropic Temperature Dependent 3.3.5t

Material Type 4 - Orthotropic Temperature Dependent 3.3.6t

Material Type 10 - Isotropic temperature dependent 3.3.7t

4. User Defined Integration Rules Beams with Arbitrary Cross Sections 4.1
5. User Defined Integration Rules for Shells.................... 5.1
6. Nodal Point Cards 6.1
7. Element Cards for Solid Elements 7.1
8. Element Cards for Beam Elements.................... 8.1
9. Element Cards for Shell Elements 9.1
10. Element Cards for 8-Node Solid Shell 10.1
11. Interface Definitions for Component Analysis....................11.1
12. DYNA3D/JOY Interface Definition....................12.1
13. Nodal Single Point Constraints....................13.1
14. Local Coordinate Systems....................14.1
15. Initial Vehicle Kinematics Cards 15.1
16. Sliding Boundary Plane Cards....................16.1
17. Symmetry Planes with Failure....................17.1
18. Nodal Time History Blocks....................18.1
19. Element Time History Blocks....................19.1
20. Density versus Depth Curve for Gravity Loading....................20.1
21. Brode Function Data 21.1
22. Cross Section Definition for Force Output....................22.1
23. Load Curve/Table Definition Cards....................23.1
24. Concentrated Nodal/Rigid Body Forces 24.1
25. Pressure Boundary Condition Cards....................25.1
26. Traction Boundary Cards for Beam Elements....................26.1
27. Velocity/Acceleration/Displacement Cards for Nodes/Rigid Bodies....................27.1
28. Generalized Stonewall Cards....................28.1
29. Nodal Force Groups....................29.1
30. Nodal Constraint and Spotweld Cards....................30.1
31. Initial Velocities 31.1
32. Contact Interface Definitions 32.1
33. Tie-Breaking Shell Definitions....................33.1
34. Tied Node Sets with Failure 34.1
35. Nodes Moved Via Section 11 Interface File 35.1

36. Rigid Body Merge Cards 36.1
37. Nodal Rigid Body Constraint and Welded Node Sets 37.1
38. Extra Nodes for Rigid Bodies 38.1
39. Joint Definition Cards 39.1
40. Base Acceleration in X-direction 40.1
41. Base Acceleration in Y-direction 41.1
42. Base Acceleration in Z-direction 42.1
43. Angular Velocity About X-Axis 43.1
44. Angular Velocity About Y-Axis 44.1
45. Angular Velocity About Z-Axis 45.1
46. Body Force Material Subset for Sections 39-44 .... 46.1
47. Generalized Body Force Load Input 47.1
48. Momentum Deposition Data 48.1
49. Detonation Point Data 49.1
50. Shell-Brick Interfaces 50.1
51. Discrete Springs, Dampers, and Masses 51.1
Material Type 1 for Discrete Elements (Linear Elastic) 51.3
Material Type 2 for Discrete Elements (Linear Viscous) 51.3
Material Type 3 for Discrete Elements (Isotropic Elastoplastic) 51.3
Material Type 4 for Discrete Elements (Nonlinear Elastic) 51.3
Material Type 5 for Discrete Elements (Nonlinear Viscous) 51.4
Material Type 6 for Discrete Elements (General Nonlinear) 51.4
Material Type 7 for Discrete Elements (Three Parameter Maxwell Viscoelastic) 51.5
Material Type 8 for Discrete Elements (Inelastic tension or compression only) 51.6
52. Seat Belts 52.1
Belt Material Definition 52.1
Belt Element Definition 52.3
Slipring Definition 52.4
Retractor Definition 52.6
Sensor Definition 52.11
Pretensioner Definition 52.14
Accelerometers 52.16
53. Rigid Body Inertial Properties and Constraints 53.1
54. Nonreflecting Boundaries 54.1
55. Temperature Data Option I 55.1
56. Temperature Data Option II 56.1

Table of Contents

57. 1D Slideline Definitions 57.1
58. Lumped Parameter Control Volumes 58.1
59. Geometric Contact Entities 59.1
60. Rigid Body Stoppers 60.1
61. AVS Database 61.1
62. MPGS Database 62.1
63. MOVIE Database 63.1
64. System Damping by Material 64.1
65. Input Data For User Interface Control Subroutine 65.1
66. Input Data For User Interface Friction Subroutine 66.1
67. Linear Constraint Equations 67.1
68. Cyclic Symmetry 68.1
69. Implicit Springback Solution 69.1
70. Superplastic Forming Option 70.1
71. Material Repositioning Section 71.1
72. Termination Criterion 72.1
73. ALE Smoothing Constraints 73.1
74. Tracer Particles 74.1
75. Shell Parts Tied to Solid Parts 75.1
76. Multi-Material Euler 76.1
77. Pressure Outflow Boundary Conditions 77.1
78. Reference Geometry for Airbag 78.1
79. USA Surface Boundary Condition Cards 79.1
80. MCOL Input Cards 80.1
81. Temperature Initial Condition Cards 81.1
82. Element Heat Generation Cards 82.1
83. Temperature Boundary Condition Cards 83.1
84. Flux Boundary Condition Cards 84.1
85. Convection Boundary Condition Cards 85.1
86. Radiation Boundary Condition Cards 86.1
87. Boundary Element Method for Fluid Dynamics 87.1
88. User Defined Loading and Sub-Sea Structural Loading 88.1
89. Subsystem Definitions for the SSSTAT File 89.1
90. Feedback Control for Load Curve Adjustment 90.1
91. Beam Force and Moment Release Cards 91.1
92. Rigid/Deformable Material Switching 92.1

93. Velocity Reinitialization After DR 93.1

RESTART INPUT DECK (Optional) R.1

1. Title Card R.1.1
2. Control Cards R.2.1
3. Sliding Interfaces Small Penetration Option R.3.1
4. Deleted/Activated Sliding Interfaces R.4.1
5. Deleted Contact Entities R.5.1
6. Deleted Materials R.6.1
7. Deleted Solid Element Blocks R.7.1
8. Deleted Beam Element Blocks R.8.1
9. Deleted Shell Element Blocks R.9.1
10. Deleted Thick Shell Element Blocks R.10.1
11. Changed Boundary Condition Cards R.11.1
12. Changed Rigid Body Restraints R.12.1
13. Material Initialization R.13.1
14. Load Curve Cards R.14.1
15. Damping/Dynamic Relaxation R.15.1
16. Implicit Springback Solution R.16.1
17. Changed Rigid Body Stoppers R.17.1
18. Displacement Termination R.18.1
19. Modified Nodal Velocities R.19.1
20. Modified Rigid Body Velocities R.20.1
21. Rigid/Deformable Material Switching R.21.1

REFERENCES REF.1

APPENDIX A

User Defined Materials A.1

APPENDIX B

User Defined Airbag Sensor B.1

APPENDIX C

User Defined Solution Control C.1

APPENDIX D

User Defined Interface Control D.1

APPENDIX E

User Defined Interface Friction E.1

Table of Contents

APPENDIX F

Version 912 Control Cards........F.1

Control Cards........F.1

Optional Card 11.1 for Seat Belts and Accelerometers (7I5)........F.17

Optional Card 11.2 for MADYMO3D/CAL3D Coupling........F.18

APPENDIX G

Occupant Simulation Including The Coupling to Programs CAL3D and MADYMO........G.1

Introduction........G.1

The LS-DYNA-Occupant Simulation Link........G.1

Dummy Modeling........G.4

Airbag Modeling........G.4

Knee Bolster........G.6

Common Errors........G.6

APPENDIX H

Interactive Graphics Commands........H.1

Interactive Rezoning Commands........H.1

APPENDIX I

Interactive Material Model Driver........I.1

Introduction........I.1

Input Definition........I.1

Interactive Driver Commands........I.3

APPENDIX J

Commands for Two-Dimensional Rezoning........J.1

APPENDIX K

LS-TAURUS User's Manual........K.1

LS-DYNA USER'S MANUAL
Nonlinear Dynamic Analysis of Structures

ABSTRACT

This report provides an updated user's manual for LS-DYNA , an explicit three-dimensional finite element code for analyzing the large deformation dynamic response of inelastic solids and structures. A contact-impact algorithm permits gaps and sliding along material interfaces with friction. Using a specialization of this algorithm, such interfaces can be rigidly tied to admit variable zoning without the need of transition regions. Spatial discretization is achieved by the use of 8-node solid elements, 2-node beam elements, 4-node shell elements, 8-node solid shell elements, truss elements, membrane elements, discrete elements, and rigid bodies. The equations-of-motion are integrated in time by the central difference method. LS-DYNA currently contains more than fifty material models and eleven equations-of-state to cover a wide range of material behavior.

BACKGROUND

DYNA3D [Hallquist 1979] was developed in 1976 and was successfully applied to a moderate number of problems. These early applications tended to be time consuming and, as a result, discouraged many potential users. Furthermore, the sliding interface logic lacked the capability to treat interfaces comprised of one or more triangular segments that are common in meshes of axisymmetric geometries. In an attempt to alleviate these drawbacks, a new version of DYNA3D was released in 1979 that had been re-programmed to provide near optimal speed on the CRAY-1 computers, contained an improved sliding interface treatment that permitted triangular segments, and was an order of magnitude faster than the previous treatment. The 1979 version eliminated structural and higher order solid elements and some of the material models of the first version. These latter features were eliminated due primarily to excessive computational cost and lack of use. This version also included an optional element-wise implementation of the integral difference method of Wilkins et al. [1974]. DYNA3D has been used continuously since 1979.

The 1981 version of DYNA3D [Hallquist 1981] evolved from the 1979 version. Body force loads and nine additional material models were added for modeling a much broader range of problems, including explosive-structure and soil-structure interactions. A link was established from the 3D Eulerian code JOY [Couch] to DYNA3D for studying the structural response of impacts by penetrating projectiles. JOY computes the penetration problem and specifies the motion of a common interface. A sliding-only interface option, based on the same option in DYNA2D [Hallquist 1978 1980], was also added. The finite difference option [Wilkins et al. 1974] was eliminated since it was much more expensive than the finite element method without a compensating increase in accuracy (see Goudreau 1982 for additional information).

The 1982 version of DYNA3D was reorganized to accept DYNA2D material input directly. The new organization permitted each equation-of-state and constitutive model to have unique storage requirements. The complete vectorization of the material models gave an additional 10 percent increase in execution speed. Theoretical documentation [Hallquist 1982], now somewhat dated, describes the procedure for incorporating new material models.

In the 1987 version of DYNA3D 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,

that greatly expand its range of applications. New capabilities added in the 1988 version include:

- cost effective resultant beam element,
- truss element,
- C^o triangular shell,
- BCIZ triangular shell,
- mixing of element formulations in calculations,
- composite failure modeling for solids,
- noniterative plane stress plasticity,
- contact surfaces with spotwelds,
- tiebreak sliding surfaces,
- beam surface contact,
- finite stonewalls,
- stonewall reaction forces,

- energy calculations for all elements,
- crushable foam constitutive model,
- comment cards in the input,
- one-dimensional slidelines.

The 1989 version of DYNA3D introduces many enhanced capabilities, including:

- interface segment save option,
- one way treatment of slide surfaces with voids and friction,
- cross section forces for structural elements,
- optional user specified minimum time step size for solid and shell elements using elastic or elastoplastic material models (once this minimum Δt is reached, material properties are modified to prevent further decrease in the time step),
- nodal accelerations in the time history database,
- time history specification now permits up to 2000 blocks each containing 2000 nodes, solid elements, beam elements, shell elements, or thick shell elements,
- stonewall forces in the TAURUS database (plot using global component 13 in TAURUS),
- compressible Mooney-Rivlin material model,
- closed-form update shell plasticity model,
- Frazer-Nash rubber material model,
- unique penalty specifications for each slide surface,
- external work is computed (except specified velocity and acceleration boundary conditions),
- optional time step criterion for 4-node shell elements,
- elements sorted internally to allow full vectorization of right-hand-side calculations; user no longer needs to group similar materials together for best performance.

During the past seven years at LSTC, considerable progress has been made as may be seen in the chronology of the developments which follows. In 1990 the following capabilities were delivered to users in LS-DYNA3D:

- 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 brick 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,

and 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 hystersis,
- 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.

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 brick 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,

new options added to version 936 in 1995 include:

- 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 now 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 now 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.

and in version 950, in 1996:

- 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
- Mechanical threshold stress (MTS) plasticity model for rate effects.
- Anisotropic viscoplastic material law (model 103)
- 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.
- Tied nodes with failure now also can apply to solid elements.
- 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.
- 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 [Nucholtz, 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.
- 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..

and many more enhancements not mentioned above.

In the sections that follow, some aspects of the current version of LS-DYNA are briefly discussed.

MATERIAL MODELS

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 (1992).

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 soil and crushable foam, the linear viscoelastic, and the rubber subroutines were adapted from HONDO and recoded for vectorization; 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 1 are presently available. The structural elements are the Hughes-Liu rectangular beams and shells [Hughes and Liu 1981a 1981b 1981c], implemented as described in [Hallquist et al. 1985, Hallquist and Benson 1986], as well as the Belytschko-Tsay shell [Belytschko and Tsay 1981 1983 1984], the YASE shell [Englemann et al. 1989] and the Belytschko-Schwer beam [Belytschko and Schwer 1977]. Triangular shell elements have now been implemented, based on work by Belytschko and co-workers [Belytschko and Marchertas 1974, Bazeley et al. 1965, Belytschko et al. 1984], (**Note:** collapsed quadrilateral shell elements lock due to the transverse shear). Three dimensional plane stress constitutive subroutines are implemented for the shell elements which update the stress tensor such that the stress component normal to the shell midsurface is zero. One constitutive evaluation is made for each integration point through the shell thickness. The 8-node solid element uses either one point integration or the constant stress formulation of Flanagan and Belytschko [1981] with exact volume integration. Zero energy modes in the shell and solid elements are controlled by either an hourglass viscosity or stiffness. Eight node solid-shell elements are implemented, but should be used cautiously; we are experimenting with their formulation. 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.

SLIDING INTERFACES

The three-dimensional contact-impact algorithm was orginally an extension of the NIKE2D [Hallquist 1979] two-dimensional algorithm. As currently implemented, one surface of the interface is identified as 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. 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. Twenty types of interfaces can presently be defined including:

1-sliding only for fluid/structure or gas/structure interfaces,
2-tied,
3-sliding, impact, friction,
4-single surface contact (NMS = 0),
5-discrete nodes impacting surface,
6-discrete nodes tied to surface,
7-shell edge tied to shell surface,
8-nodes spot welded to surface,
9-tiebreak interface,
10-one way treatment of sliding, impact, friction,
11-box/material limited automatic contact for shells,
12-automatic contact for shells (no additional input required),
13-automatic single surface with beams and arbitrary orientations,
14-surface to surface eroding contact,
15-single surface eroding contact,
16-node to surface eroding contact,
17-surface to surface symmetric constraint method [Taylor and Flanagan 1989],
18-node to surface constraint method [Taylor and Flanagan 1989],
19-rigid body to rigid body contact with arbitrary force/deflection curve,
20-rigid nodes to rigid body contact with arbitrary force/deflection curve,
21-single edge contact.
22-drawbead

Interface friction can be used with interface types 3, 4, 5, 8-13, and 17-20. 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 (Types 5,10, 17, and 18) with rigid materials, the rigid material should be chosen as the master surface.

INTERFACE DEFINITIONS FOR COMPONENT ANALYSIS

Interface definitions in Section 11 are used to define surfaces, nodal lines, or nodal points for which the displacement and velocity time histories are saved at some user specified frequency. This data may then be used in subsequent analyses as master surfaces of type 2 sliding interfaces of Section 31, as master lines in the tie breaking shell definitions of Section 32, or as the controlling nodes for determining the motion of single nodal points in Section 34. 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.

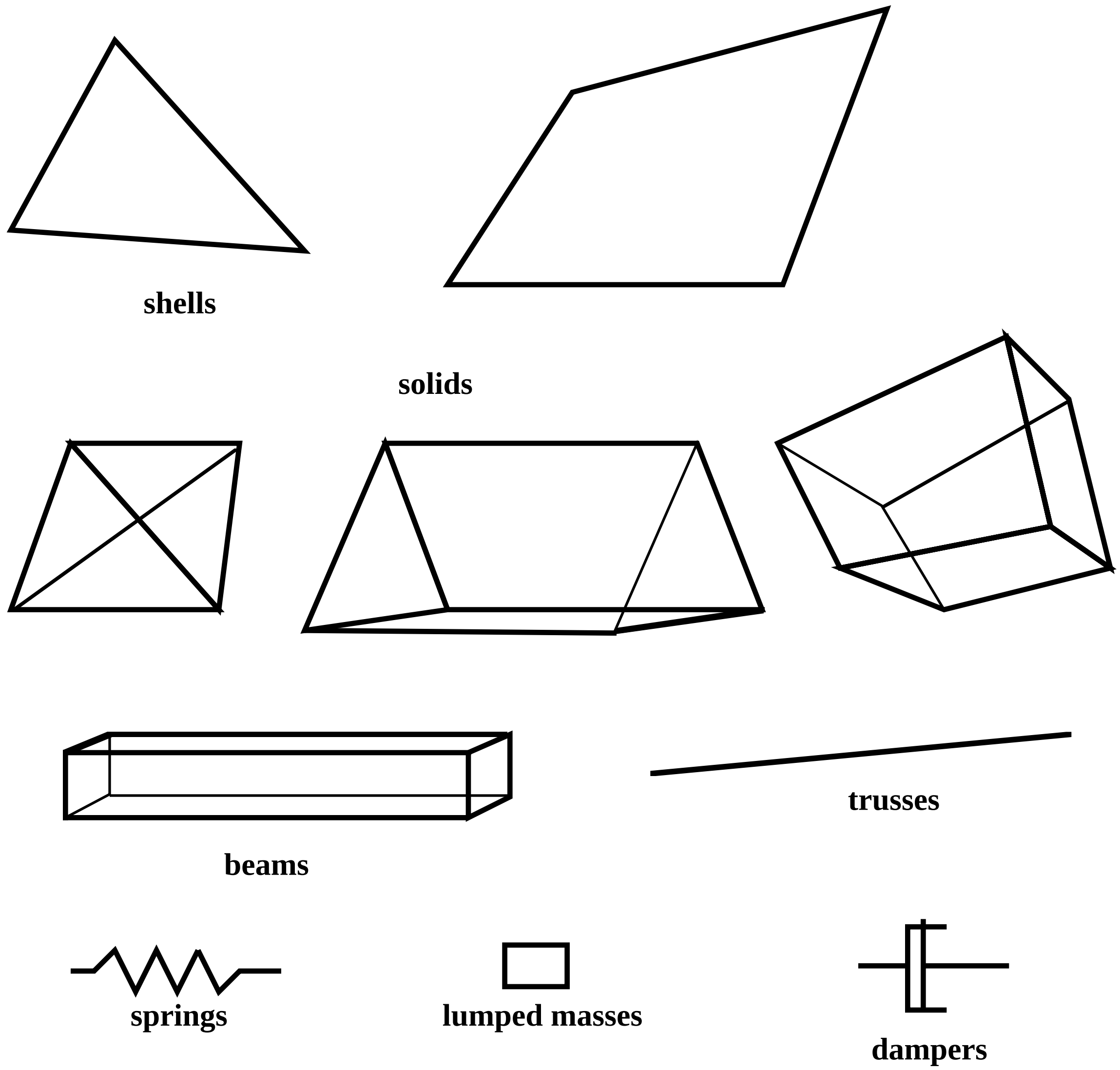


Figure 1. Elements in LS-DYNA .

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 200,000 elements.

CODE ORGANIZATION

LS-DYNA consists of one source that compiles under FORTRAN compilers on most UNIX workstations and supercomputers. The programming follows the FORTRAN 77 standard. LS-DYNA has seven segments in the main code. They are:

- input,
- restart,
- initialization,
- solution,
- interactive real time graphics,
- rezoning,
- remapping.

Data parallel versions of LS-DYNA are supported for the SGI and CRAY, and an MPP version of LS-DYNA is now be ported to a subset of the commercially available MPP machines.

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).

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 type “kill -2 psid”. LS-DYNA will first look for a file called “switch” which should contain the sense switch data. Otherwise, an SW2 is assumed and the output is sent to standard out.

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 much less sensitive to machine precision than other finite element solution methods. Consequently, on VMS and UNIX systems LS-DYNA uses 32-bit arithmetic rather than 64-bit. The benefits of this are greatly improved utilization of memory and disk. When problems have been found we have usually been able to overcome the round-off by reorganizing the algorithm where the problem occurs. More recently we have modified several critical areas to use double precision on engineering workstations. A few of the known problems include: **(32-bit implementation 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.

EXECUTION

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

DYNA3D I=**inf** O=**otf** G=**ptf** D=**dpf** F=**thf** 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**

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),
thf = binary plot file for time histories of selected data (default=D3THDT),
tpf = optional temperature file (TOPAZ3D plotfile),
rrd = running restart dump file (default=RUNRSF),
sif = stress initialization file,
jif = 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 current calculation or restart,
kill = if LS-DYNA encounters this file name it will terminate with a restart file (default=D3KIL),
vda = CAD database for geometrical surfaces,
c3d = CAL3D input file.
nwds = Number of words to be allocated. On engineering workstations a word is usually 32bits.

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.

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

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.

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. The file size may be increased by specifying the file size on the execute line using X=**scl**. The default file size hold seven times one-million octal word (262144) or 1835008 words. If the core required by LS-DYNA requires more space the **scl** should 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 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

DYNA3D I=**inf** O=**otf** G=**ptf** D=**dpf** R=**rtf** F=**thf** 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* **MEMORY=nwds**

where

rtf = up to eight character restart filename.

If the data from the last run is to be remapped onto a new mesh, then specify: Q=**remap.** The remap option is available for two and three dimensional solid elements only.

File name dropouts are permitted, for example the execution lines are acceptable.

DYNA3D I=**inf**
DYNA3D R=**rtf**

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

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

For an analysis using interface segments line one of the NAMES file in the first analysis is given by:

I=inf Z=isf1

and in the second by:

I=inf L=isf1

Files inf, rtf, and isf1 above are user defined names.

VDA DATABASE

If VDA surfaces are to be used, the file specified by **vda** must have the following form. The file is free formatted with blanks as delimiters. Note that the characters “}” and “{” must be separated from the other input by spaces or newlines. The **vda** file may contain any number of input file specifications of the form

file **afile** **bfile** {

alias definitions

}

alias definitions

followed by optional runtime parameters and a final end statement.

afile is optional, and if given must be the name of an ASCII input file formatted in accordance with the VDA Surface Interface Definitions as defined by the German automobile and automotive supply industry. **bfile** is required, and is the name of a binary VDA file. If **afile** is given, **bfile** will be created or overwritten. If **afile** is not specified, **bfile** will be read instead. The purpose of **bfile** is that it allows for much faster initialization if the same VDA surfaces are to be used in a future LS-DYNA run. The alias definitions are of the form

alias **name** { el1 el2 ... eln }

where **name** is any string of up to 12 characters, and el1,...,eln are the names of VDA elements as specified in **afile**. The list of elements can be empty, in which case all the SURF and FACE VDA elements in **afile** will be used. Care should be taken to ensure that the alias **name** is unique, not only among the other aliases, but among the VDA element names in **afile**. This collection of VDA elements can later be indicated by the alias **name**. In particular, **name** may appear in later alias definitions. As an option, the keyword **offset** may appear in the alias list which allows a new surface to be created as a normal offset (plus translation) of a VDA element in the file. The keyword **offset** my be applied to VDA elements only, not aliases. The usage of **offset** follows the form

offset elem **normal x y z**

where **normal** is the amount to offset the surface along the normal direction, and **x**,**y**,**z** are the translations to be applied. The default normal direction is given by the cross product of the local u and v directions on the VDA surface, taken in that order. **normal** can be negative. One of the primary uses of this is to allow a sheetmetal punch to be simulated given only the VDA information for the die.

Frequently, it is convenient create a new alias **name** by offsetting and translating an existing **name**. The keyword **goffset** provides this funtion:

goffset **alias_name** $\mathbf{x_c}$ $\mathbf{y_c}$ $\mathbf{z_c}$ **normal x y z** { previous **alias_name** }

where **normal**, **x**, **y**, and **z** are defined as in the offset keyword. A reference point $\mathbf{x_c}$, $\mathbf{y_c}$, and $\mathbf{z_c}$ defines a point in space which determines the normal direction to the VDA surface.

Finally, several parameters affecting the VDA surface iteration routines can be reset in the file **vda**. These parameters, and their default values, are:

gap [5.0] The maximum allowable surface gap to be filled in during the iterations. Points following the surface will effectively extend the edges of surfaces if necessary to keep them from falling through cracks in the surface smaller than this. This number should be set as small as possible while still allowing correct results. In particular, if your VDA surfaces are well formed (having no gaps) this parameter can be set to 0.

track [2.0] A point must be within this distance of contact to be continually tracked. When a point not being tracked comes close to a surface, a global search is performed to find the near surface point. While a point is being tracked, iterations are performed every cycle. These iterations are much faster, but if the point is far away it is faster to occasionally do the global search.

track2 [5.0] Every VDA surface is surrounded by a bounding box. When a global search needs to be performed but the distance from a point to this box is > **track2**, the actual global search is not performed. This will require another global search to be performed sooner that if the actual distance to the surface were known, but also allows many global searches to be skipped.

ntrack [4] The number of VDA surfaces for which each point maintains actual distance information. A global lower bound on distance is maintained for all remaining surfaces. Whenever the point moves far enough to violate this global lower bound, all VDA surfaces must have the global search performed for them. Hence this parameter should be set to the maximum number of surfaces that any point can be expected to be near at one time (the largest number of surfaces that come together at one point). Setting **ntrack** higher will require more memory but result in faster execution. If **ntrack** is too low, performance may be unacceptably slow.

toroid [.01] Any surface with opposing edges which are within distance [t] of each other is assumed to be cylindrical. Contacts occuring on one edge can pass to the adjacent edge. The default value is 0.01.

converge [.01] When surface iterations are performed to locate the near point, iteration is continued until convergence is detected to within this distance (all VDA coordinates are in mm). The default value is 0.01.

iterate [8] Maximum number of surface iterations allowed. Since points being tracked are checked every cycle, if convergence fails it will be tried again next cycle, so setting this parameter high does not necessarily help much.

On the other hand, a point converging to a crease in the VDA surface (a crease between patches with discontinuous derivative, for example) may bounce back and forth between patches up to this many times, without actually moving. Hence, this value should not be too large. The default value is 8.

el_size [t mx mn]

Controls the generation of elements where:

t =surface tolerance for mesh generation,

mx=maximum element size to generate,

mn=minimum element size to generate.

The default values are [0.25 100. 1.0]

aspect [s1 s2]

Controls the generation of elements where:

s1=maximum difference in aspect ratio between elements generated in neighboring VDA patches,

s2=maximum aspect ratio for any generated element.

The default values are [1.5 4.0]

cp_space [10] Determines the spacing around the boundaries of parts at which the size of elements is controlled. In the interior of the part, the element size is a weighted function of these control points as well as additional control points in the interior of the region. If there are too few control points around the boundary, elements generated along or near straight boundaries, but between control points, may be too small. The default value is 10.

meshonly The existance of this keyword causes LS-DYNA to generate a file containing the mesh for the VDA surfaces and then terminate.

onepatch The existance of this keyword causes LS-DYNA to generate a single element on each VDA patch.

somepatch [n] Like onepatch, but generates an element for 1 out of every [n] patches.**converge** [.01] When surface iterations are performed to locate the near point, iteration is continued until convergence is detected to within this distance (all VDA coordinates are in mm).

Here is a short example of a **vda** input file:

```
file vda1 vda1.bin {
        alias die {
                sur0001
                sur0003
                offset fce0006 1.5 0 0 120
        }
        alias holder1 { sur008 }
}
file vda2 vda2.bin {
        alias holder2 { sur003 }
}
alias holder { holder1 holder2 }
ntrack 6
gap 0.5
end
```

MESH GENERATION

LS-DYNA relies on stand-alone mesh generators for creation of input files. LS-INGRID [Stillman and Hallquist 1985] is the recommended method of generating LS-DYNA meshes since it provides complete support for all slide surface data, boundary conditions, loads, material properties and control parameters.

POST-PROCESSING

LS-TAURUS, a much enhanced version of TAURUS [Brown and Hallquist 1984], processes output from LS-DYNA . LS-TAURUS 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. TAURUS can compute a variety of strain measures, reaction forces along constrained boundaries, and momenta. TAURUS is operational on most computers. Interfaces from LS-TAURUS to other commercial post-processors are available.

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 last contains interface data for contact surfaces.

Because of the difficulty in handling one large ASCII file an alternative method for obtaining printed output is now available. Twenty 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.

EXECUTION SPEEDS

The execution speeds on the Cray-YMP for various elements in LS-DYNA are tabulated below in microseconds per element cycle:

Element Type	CPU Cost
8 node solid with 1 point integration and default hourglass control	12
as above but with Flanagan-Belytschko hourglass control	15
constant stress and Flanagan-Belytschko hourglass control, i.e., the Flanagan-Belytschko element	20
4 node Hughes-Liu shell with four thickness integration points	30
4 node Belytschko-Tsay shell with four thickness integration points	13
4 node Belytschko-Tsay shell with resultant plasticity	11
BCIZ triangular shell with four thickness integration points	22
C^o triangular shell with four thickness integration points	11
2 node Hughes-Liu beam with four integration points	28
2 node Belytschko-Schwer beam	5
2 node simple truss elements	3
8 node solid-shell with four through the thickness integration points	33

These timings are very approximate and do not account for the inclusion of sliding interfaces or complex material models. Each interface node of the sliding interfaces is roughly equivalent to one-half zone cycle in cost.

LS-DYNA User's Guide
1. Title Card
(12A6,A2,A1,A5)

Columns	Quantity	Format
1-72	Heading to appear on output	12A6
73-74	Input code version EQ.87: input follows manual published in 1987 EQ.88: input follows manual published in 1989 EQ.90: input follows manual published in 1990 EQ.91: input follows manual published in 1991 EQ.92: input follows manual published in 1992 EQ.93: input follows this manual	A2
75	Version number EQ.0: for versions 87, 88, 89, 91, 92, and 93 EQ.4: for versions 903, 904, 905 and 906 EQ. i: activates reading of Implicit Control Cards	A1
76-80	Input format "LARGE" - Large input format node and element numbers upto 99999999 may be used. "MLARG" - As "LARGE" but material (part) numbers upto 99999999 may be used.	A5

2. Control Cards

Card 1. Model Size—General

(7I10,2I5)

Columns	Quantity	Format
1-10	Number of part sets, NMMAT	I10
11-20	Number of nodal points, NUMNP	I10
21-30	Number of solid hexahedron elements, NUMELH	I10
31-40	Number of beam elements, NUMELB	I10
41-50	Number of 4-node shell elements, NUMELS	I10
51-60	Number of 8-node solid shell elements, NUMELT	I10
61-70	Number of user defined material subroutines, NUSRMT	I10
71-75	Number of parts (shells) tied to solid element parts, NALTIE	I5
76-80	Number of tracer particles for tracking fluid flow, NTRACE	I5

Card 2. Model Size—Boundary Conditions (8I5)

Columns	Quantity	Format
1-5	Number of single point constraint nodes, NODSPC	I5
6-10	Number of coordinate systems, NSPCOR	I5
11-15	Number of velocity/acceleration boundary condition cards, NUMVC	I5
16-20	Number of nonreflecting boundary segments, NNRBS In two dimensional problems, NNRBS is the number of boundary definitions where a boundary definition includes a string of consecutive nodal points. (See Section 53.)	I5
21-25	Number of sliding boundary planes, NUMRC	I5
26-30	Number of symmetry planes with failure, NUMRCF	I5
31-35	Number of nodes in DYNA3D-JOY interface, NUMSNC	I5
36-40	Number of nodes in each interface for cyclic symmetry, NNCSYM	I5

Card 3. Model Size—Loading
(11I5)

Columns	Quantity	Format
1-5	Number of load curve/table definitions, NLCUR GT.0:define load curves in Section 23. Arbitrary numbering cannot be used. LT.0:the absolute value is the number of load curves. Arbitrary numbering is assumed. Section 23 data is moved to the beginning of the part/material definitions.	I5
6-10	Number of concentrated nodal loads, NUMCL	I5
11-15	Number of segments having pressure loads applied, NUMPC	I5
16-20	Number of generalized body force loads (old name NUMGBL), IBODYL	I5
21-25	Number of traction boundary cards for beam elements, NUMBPC	I5
26-30	Number of detonation points, NDTPTS. Also, see Control Card 10, Columns 41-45.	I5
31-35	Number of solid hexahedron elements for momentum deposition NELMD	I5
36-40	Number of points in density versus depth curve, NUMDP	I5
41-45	Number of outflow boundary segments attached to ambient elements, NOFLOW	I5
46-50	Number of load curve feedback sets.	I5
51-55	Number of pressure load sets by shell part ID with masks, NUMPRM.	I5

Card 4. Model Size—Constraints and Contact
(10I5)

Columns	Quantity	Format
1-5	Number of rigid walls (stonewalls), NUMRW	I5
6-10	Number of sliding interfaces (old name NUMSI), NUMSV	I5
11-15	Number of shell-solid element interface definitions, NBLK	I5
16-20	Number of tie-breaking shell slidelines, NTBSL	I5
21-25	Number of tied node set with failure definitions, NTNWF	I5
26-30	Number of nodes, NTNPFL, tied to nodes in an interface data-base generated in a previous run, introduced by Section 11 or 31 input below. This file is specified on the execute line by specifying I=inf2, where inf2 is the file name.	I5
31-35	Number of nodal constraint cards, NUMCC	I5
36-40	Number of linear constraint equations, NOCEQS	I5
41-45	Number of 1D slideline definitions, NUMSL	I5
46-50	Number of adaptive constraints	I5
51-55	Number of ALE smoothing constraints, NALESC	I5
56-60	Number of 2D slideline definitions, NSL This input is based on the DYNA2D contact options and will work in a similar way to DYNA2D.	I5
61-65	Number of 2D *automatic* slideline definitions, NAUTO. This input is for the new 2D contact algorithm.	I5
66-70	Number of part ID’s (solid element parts) for interior contact. This option requires additional input at the beginning of section 31 to provide the list of part ID's for which internal contact is generated. The purpose of this option is to avoid negative volumes when the foam elements compact.	I5

Card 5. Model Size—Rigid Body Parameters
(8I5)

Columns	Quantity	Format
1-5	Number of nodal rigid body constraint sets, NUMRBS	I5
6-10	Number of rigid body merge cards, NRBC	I5
11-15	Number of joint definitions, NJT	I5
16-20	Number of rigid bodies for which extra nodes are defined, NXTRA	I5
21-25	Number of rigid bodies for which inertial properties are defined, NUMRBI. This is an optional override of values otherwise computed internally.	I5
26-30	Number of rigid body geometric contact entities	I5
31-35	Number of generalized joint stiffnesses, NJTS.	I5
36-40	Number of rigid body stoppers, NRBSTP.	I5

Card 6. Model Size—Discrete Elements and Seat Belts (12I5)

If any of NMMTDE, NMCORD, NMELDE or NMMASS is greater than 99,999, enter -1 in the appropriate column and define it instead in I10 format on card 6a below.

Columns	Quantity	Format
1-5	Number of material definitions, NMMTDE EQ:-1 read from card 6a	I5
6-10	Number of orientation vectors, NMCORD EQ:-1 read from card 6a	I5
11-15	Number of discrete springs and dampers, NMELDE EQ:-1 read from card 6a	I5
16-20	Number of discrete masses, NMMASS EQ:-1 read from card 6a	I5
21-25	Number of belt materials, NUMSBM	I5
26-30	Number of seat belt elements, NUMSBE	I5
31-35	Number of sliprings, NMSBSR	I5
36-40	Number of retractors, NMSBRT	I5
41-45	Number of sensors, NMSBSE	I5
46-50	Number of pretensioners, NMSBPT	I5
51-55	Number of accelerometers, NUMACC	I5
56-60	Number of nodal inertia tensors, NUMRBI	I5

Card 6a. Model Size—Discrete Elements - Extra card
(I10)

Define this card if and only if -1 appears in any of the first four columns of card 6. Define one card for each parameter given a value of -1 The order is as specified on card 6.

Columns	Quantity	Format
1-10	Value for parameter defined above.	I10

Card 7. Model Size—Output Control
(4I5)

Columns	Quantity	Format
1-5	Number of cross section definitions for force output, NUMCSD	I5
6-10	Number of nodal force groups for resultant force output, NODEFR	I5
11-15	Number of interface definitions output for subsequent component analysis, NUMIFS	I5
16-20	Maximum number of 4-node segments (optional) specified in definitions of any stonewall, NRWSEG. This option is necessary to get the stonewall force distribution.	I5

Card 8. Computation Options—Termination
(E10.0,I10,3E10.0,2I10,I5)

Columns	Quantity	Format
1-10	Termination time, ENDTIM. Default = 0.0	E10.0
11-20	Termination cycle. Default = 0, in which case check is not made.	I10
21-30	Reduction (or scale) factor DTMIN for initial time step size to determine minimum time step, TSMIN. TSMIN=DTSTART*DTMIN where DTSTART is the initial step size determine by LS-DYNA . When DTMIN is reached LS-DYNA terminates with a restart dump. Default = 0.0. Also see Card 9 below, columns 61-65.	E10.0
31-40	Percent change in total energy ratio for termination of calculation. If undefined this option is inactive.	E10.0
41-50	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 Card 9 below Columns 41-50. Default=10000%.	E10.0
51-60	NUMSTOP, number of nodal point displacement termination conditions. See Section 72.	I10
61-70	NRBEND, number of rigid body displacement termination conditions. See Section 72.	I10
71-75	NCNEND, number of contact termination conditions. See Section 72.	I10

Card 9. Computation Options—Time Step Size Control (2E10.0,I10,2E10.0,I10,2I5)

Columns	Quantity	Format
1-10	Initial time step size, DT2OLD. EQ.0.0: LS-DYNA determines initial step size	E10.0
11-20	Scale factor for computed time step (old name SCFT), TSSFAC. (Default = .90; if high explosives are used, the default is lowered to .67).	E10.0
21-30	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.	I10
31-40	Shell element minimum time step assignment, TSLIMT. When a shell controls the time step, element material properties will be modified such that the time step does not fall below the assigned step size. Applicable only to shell elements using material models 3, 18, 19, and 24. The option in 41-50 below applies to all materials and element classes and may be preferred.	E10.0
41-50	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 Card 8 above, Columns 41-50. TSSFAC is defined above in Columns 11-20. See flag for limited mass scaling below.	E10.0
51-60	Load curve number that limits maximum time step size, LCTM (optional). This load curve specifies the maximum time step versus time.	I10

Columns	Quantity	Format
61-65	Erosion flag for solid and solid shell elements when DTMIN (see Card 8 above) is reached. If this flag is not set the calculation will terminate. DTMIN must exceed zero. EQ.0: no EQ.1: yes	I5
66-70	Limit mass scaling to the first step and fix the mass vector afterwards. The time step will not be fixed but may drop during the calculation from the specified minimum. EQ.0: no EQ.1: yes	I5

Card 10. Computation Options—Loading
(8I5)

Columns	Quantity	Format
1-5	Base acceleration in x-direction, NTHPX EQ.0:no EQ.1:yes	I5
6-10	Base acceleration in y-direction, NTHPY EQ.0:no EQ.1:yes	I5
11-15	Base acceleration in z-direction, NTHPZ EQ.0:no EQ.1:yes	I5
16-20	Angular velocity about x-axis, NTHSX EQ.0:no EQ.1:yes	I5
21-25	Angular velocity about y-axis, NTHSY EQ.0:no EQ.1:yes	I5
26-30	Angular velocity about z-axis, NTHSZ EQ.0:no EQ.1:yes	I5
31-35	Number of materials for body force loads. EQ.0:input in columns 1-30 applies to all materials. EQ.n:input in columns 1-30 applies to n materials. Addtional input is expected in Section 45.	I5
36-40	Flag for user defined subroutine, FUDS. See Section 86 where additional input may be required. LT.0:yes, |FUDS| equals the number of input constants EQ.0:no EQ.1:yes, but no input is required.	I5
41-45	ISHADOW, high explosive initiation option (also, see Section 48 and Control Card 3, Columns 26-30) : EQ.0:detonation time is based on actual distance from the detonation point. It is possible with this option that the line of sight passes outside of the explosive material.	I5

Columns	Quantity	Format
	EQ.1:detonation time is based on the minimum distance from the detonation points within the explosive materials and the detonation velocities if different explosivies are used. Geometric effects are automatically taken into account. This option is useful, for example, when modelling wave shapers.	
46-50	SSA, flag for sub-sea structural analysis option (Section 86). EQ.0:no EQ.1: one point quadrature on surface segements. EQ.4: four point quadrature on surface segements.	I5

Card 11. Computation Options—Input Control
(3I5,A5,2I5)

Columns	Quantity	Format
1-5	Initial condition parameter, INITV (refer to Section 30) EQ.0: initialize velocities to zero EQ.1: initial velocities are read in EQ.2: a uniform velocity field is defined EQ.3: same as 2 but exempted nodes are defined EQ.4: box option EQ.5: generation with arbitrary node numbering (not recommended) EQ.6: rotational and translational via material/node ID EQ.7: rotational and translational via material/node ID with velocity reinitialization in Section 85	I5
6-10	Arbitrary node, element, and material numbering, NSORT, EQ.0: consecutive EQ.1: arbitrary	I5
11-15	Flag for defining material models by specifying constitutive, equation-of-state, and cross-section property definitions, IARB. EQ.0: off EQ.1: on.	I5
16-20	Nodal coordinate format, NIF: either E10.0 or E20.0. Default = E10.0	A5
21-25	Nonzero flag for input of lumped parameter control volumes, ITHCNV	I5
26-30	Brode function flag, IBRODE EQ.0: Brode parameters are not defined EQ.1: Brode parameters are defined in input	I5

Card 12. Computation Options—Beams and Shells
(E10.0,10I5,I2,I3,E10.0)

Columns	Quantity	Format
1-10	Shell element warping angle in degrees, WRPANG. If warping greater than this angle is found, a warning message is printed. See comments below. EQ.0: default set to 20 degrees	E10.0
11-15	Plane stress plasticity option (applies to materials 3, 18, 19 and 24), MITER EQ.0: default is set to 1 EQ.1: iterative plasticity with 3 secant iterations (default) EQ.2: full iterative plasticity EQ.3: radial return noniterative plasticity	I5
16-20	Automatic sorting of triangular shell element to treat degenerate quadrilateral shell elements as C_0 triangular shells, ITRIST EQ.0: default is set to 2 EQ.1: full sorting EQ.2: no sorting required	I5
21-25	Hughes-Liu shell normal update option, IRNXX EQ.-2: unique nodal fibers EQ.-1: compute normals each cycle (default) EQ.0: default set to -1 EQ.1: compute on restarts EQ.n: compute every n cycles	I5
26-30	Shell thickness change option, ISTUPD EQ.0: no change EQ.1: membrane straining causes thickness change	I5
31-35	Default shell theory required, IBELYT EQ.0: default is set to 2 EQ.1: Hughes-Liu EQ.2: Belytschko-Tsay (default, see columns 56-60 below) EQ.3: BCIZ triangular shell EQ.4: C_o 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 Belyschko-Tsay membrane EQ.10: Belytschko-Wong-Chiang EQ.11: co-rotational Hughes-Liu	I5
36-40	Number of user specified beam integration rules, NUBIR	I5

Columns	Quantity	Format
41-45	Maximum number of integration points required in the user specified rules for beam elements, MPUBR	I5
46-50	Number of user specified shell integration rules, NUSIR	I5
51-55	Maximum number of integration points required in the user specified rules for shell elements, MPUSR	I5
56-60	Warping stiffness for Belytschko-Tsay shells EQ.1:Belytschko-Wong-Chiang warping stiffness added EQ.2:Belytschko-Tsay (default)	I5
61-62	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	I2
63-65	Read in reference geometry, ARG. See Comments below. EQ.0:no reference geometry is read EQ.1:read reference geometry, active for all airbags EQ.2:read reference geometry and use in time step size computation; otherwise, LS-DYNA will base the time step size on the current geometry. With this option, the reference geometry is used to increase the time step size. Applies to airbags only. EQ.10:read reference geometry. active for foam and hyperelastic materials types 2, 5, 7, 21, 23, 27, 31, 38, 57, and 73. This option applies to 8-noded solid elements only. EQ.11:options 1 and 10 active. EQ.12:options 2 and 10 active.	I3
66-70	Invarient node numbering for shell elements EQ.1:Off (default) EQ.2:On	I5
71-80	Birth time for reference geometry. The time step size will be based on the actual geometry until the birth time is reached. Afterwards, the time step siz calculation will depend on the parameter specified in columns 61-65 above.	E10.0

The warping angle is found by computing the normal vectors at each element node based on the edges. If diagonally opposite vectors have an included angle that exceeds the specified warping angle a warning message is printed.

If ARG is set to 2, the time step size for airbag elements will be based on the reference geometry. 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 so that the step size will increase as the area of the elements increase. The default can be much more expensive but sometimes more stable.

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 recomkmended 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 a 35 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.

Card 13. Computation Options—Material Related Input
(I1,I4,E10.0,I5,2E10.0,6I5)

Columns	Quantity	Format
1-1	Hourglass formulation. EQ.0: default EQ.1: Version 936 and earlier formulations. The w-mode of the hourglass control was not orthogonal to rigid body rotations in version 936 and earlier versions when the elements were warped. This mode is now orthogonal to rigid body rotations in version 940 and later versions. However, this results is a slightly softer behavior which causes discrepancies between version 936 and 940.	I5
2-5	Default hourglass viscosity type, IHQ EQ.0: default is set to 1 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) In the shell elements, IHQ < 4 is the viscous form based on Belytschko-Tsay. If IHQ = 4 or 5, 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.	I5
6-15	Default hourglass coefficient, QH (default = .10). Values of QH that exceed .15 may cause instabilities. The recommended default applies to all options.	E10.0
16-20	Default bulk viscosity type, IBQ (Default=1) EQ.--1: standard (also type 2, 10, and 16 shell elements) EQ.+1: standard	I5
21-30	Default quadratic viscosity coefficient, Q1 (default = 1.5)	E10.0
31-40	Default linear viscosity coefficient, Q2 (default = .06)	E10.0
41-46	Flag for Rayleigh damping input (RDFLAG) by material property set EQ.0: off (default) EQ.1: on	I5
46-50	Flag for rigid/deformable material switching, IRDMS EQ.1: off (default if input as 0 or blank) EQ.2: on EQ.3: on and automatic switching of materials	I5

Columns	Quantity	Format
	If this flag is set to 2 or 3 then any deformable material in the model may be switched between rigid and deformable during the calculation. Materials that are define as type 20 in the input are permanently rigid. Additional input is required if this flag is set.	
51-55	Thermal effects option, ITEMP EQ.0: no thermal effects (default) EQ.n: temperature-time history is defined by load curve n LT.0: nodal temperatures are defined in TOPAZ3D generated disk files n=-2 and n=-9999 are special cases for steady state temperature input. See Sections 53 and 54.	I5
56-60	Superplastic analysis input option, ISUPER EQ.0: no input EQ.1: read superplastic input section	I5
61-65	Objective stress update for large timestep size EQ.0: off EQ.1: on	I5
66-70	MTCNTC, flag to read default contact parameters. In automatic contact types 4, 13, 15, and 26 only one set of friction coefficients, thickness and penalty scale factors apply. It is now possible to change these based on part/material ID if the MLARG option flag is used. The default vaules values are used unless overridden in the part/material definition. These values are read in the beginning of Section 3. EQ.0: off EQ.1: on	I5

The bulk viscosity creates an additional additive pressure term given by:

$$q = \rho l \left(Q_1 l \, \dot{\varepsilon}_{kk}^2 - Q_2 a \dot{\varepsilon}_{kk} \right) \quad \text{if } \dot{\varepsilon}_{kk} < 0$$

$$q = 0 \qquad\qquad \text{if } \dot{\varepsilon}_{kk} \ge 0$$

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.

Card 14. Computation Options—Damping/Dynamic Relaxation (I5,E10.0,2I5,4E10.0,I5,E10.0)

Columns	Quantity	Format
1-5	Load curve ID which specifies a time dependent system damping constant, LCDAMP, for mass proportional damping. See discussion below. EQ.0: no damping EQ.-1: system damping is defined for each material by load curves. The data are defined in a separate input section called “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.	I5
6-15	System damping constant, d, for mass proportional damping (this option is bypassed if the load curve number defined above is nonzero), VALDMP. If negative the absolute value is used but directional damping is assumed, and, input six scale factors on the next optional card below for each global translation and rotation. If the load curve ID above is greater than zero and directional damping is desired, set VALDMP=-1.0.	E10.0
16-20	Dynamic relaxation flag for stress initialization, IDRFLG EQ.-999: dynamic relaxation phase will not be run even if specified on load curve input. EQ.-1: dynamic relaxation activated with time history output. EQ.0: inactive EQ.1: dynamic relaxation is activated EQ.2: initialization to a prescribed geometry (see Note below).	I5
21-25	Number of iterations between convergence checks, for dynamic relaxation option (default = 250), NRCYCK	I5
26-35	Convergence tolerance for dynamic relaxation option, DRTOL. (default = 0.001)	E10.0
36-45	Dynamic relaxation factor (default = .995), DRFCTR.	E10.0
46-55	Optional termination time for dynamic relaxation, DRTERM. Termination occurs at this time or when convergence is attained. Default = infinity)	E10.0

Columns	Quantity	Format
56-65	Scale factor for computed time step during dynamic relaxation, TSSFDR. If zero, the value is set to TSSFAC defined on Card 9. After converging, the scale factor is reset to TSSFAC.	E10.0
66-70	Automatic control for dynamic relaxation option based on algorithm of Papadrakakis, IRELAL [Papadrakakis 1981] EQ.0: inactive EQ.1: active	I5
71-80	Convergence tolerance on automatic control of dynamic relaxation, EDTTL	E10.0

With system damping, the equations of motion are given by:

$$a^n = M^{-1}\left(P^n - F^n + H^n - F^n_{damp} \right)$$

where a^n is the acceleration at time n, M^{-1} is the inverse of the diagonal mass matrix, P^n is the applied force vector, F^n is the internal force vector, and F^n_{damp} is the force vector due to system damping given in terms of the damping constant D_s and nodal velocity vector, v^n, at time n as:

$$F^n_{damp} = D_s M v^n$$

The best damping constant for system damping is usually some fraction of the fundamental frequency, ω_{min} . For example, D_s could be set to:

$$D_s = 2\omega_{min}$$

if critical damping is desired. Larger values should not be used. Caution must be observed when using system damping so as not to overdamp the system or to use large damping constants which may lead to unstable solutions.

Stress initialization in LS-DYNA for small strains may be accomplished by linking to an implicit code. A displacement state is required that gives for each nodal point its ID, xyz displacement and xyz rotation. This data is read from unit 7 (m=) with the format (I8,6E15.0)

Card 14a. Computation Options—Damping/Dynamic Relaxation - Optional Extra card (6E10.0)

Define this card if and only if VALDMP is input as a negative value above. The scale factors must be positive values, but there is no defaults.

Columns	Quantity	Format
1-10	Scale factor for x-translational system damping.	E10.0
11-20	Scale factor for y-translational system damping.	E10.0
21-30	Scale factor for z- translational system damping.	E10.0
31-40	Scale factor for x-rotational system damping.	E10.0
41-50	Scale factor for y-rotational system damping.	E10.0
51-60	Scale factor for z-rotational system damping.	E10.0

Card 15. Computation Options—Contact
(2E10.0,I3,I2,8I5,E10.0)

Columns	Quantity	Format
1-10	Scale factor for sliding interface penalties, SLSFAC EQ.0: default = .10	E10.0
11-20	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.	E10.0
21-23	Bypass projection of slave nodes to master surface in types 2, 6, and 7 tied interface options: EQ.0: eliminate gaps by projection nodes, EQ.1: bypass projection. Gaps create rotational constraints which can substantially affect results.	I3
24-25	Initial penetration check in contact surfaces with indication of initial penetration in output file, ISLCHK EQ.0: the default is set to 2 EQ.1: no checking EQ.2: full check of initial penetration is performed	I2
26-30	Shell thickness considered in type 3, 5, and 10 contact options, where options 1 and 2 below activate the new contact algorithms. The thickness offsets are are always included in contact types 4, 11, 12, 13, 17, and 18, ISHLTK. See comments 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	I5
31-35	Penalty stiffness value EQ.0: the default is set to 1 EQ.1: minimum of master segment and slave node (default) EQ.2: use master segment stiffness (old way) EQ.3: use slave node value EQ.4: use slave node value, area or mass (type=5) weighted EQ.5: same as 4 but inversely proportional to the shell thickness. This may require special scaling and is not generally recommended.	I5

Card 15. Computation Options—Contact (cont.)
(2E10.0,9I5,E10.0,3x,2I1)

Columns	Quantity	Format
36-40	Shell thickness changes considered in type 4 and 13 single surface contact* EQ.0: no consideration EQ.1: shell thickness changes are included	I5
41-45	Optional automatic reorientation of contact interface segments during initialization. EQ.0: default is set to 1 EQ.1: active for automated input only EQ.2: active for manual and automated input EQ.3: inactive	I5
46-50	Storage per contact interface for user supplied interface control subroutine. 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.	I5
51-55	Storage per contact interface for user supplied interface friction subroutine. 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.	I5
55-60	Number of time steps between contact searching using three dimensional bucket searches, NSBCS, in single surface and the new surface to surface contact (Default=10).	I5
61-65	Flag for intermittent searching in old type 3 contact using the. interval specified as NSBCS in columns 55-60 above. EQ.0: off EQ.1: on	I5
66-75	Contact surface maximum penetration check multiplier, XPENE. If the small penetration checking (Section 31, Card 1, column 50) option 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. EQ.0: default is set to 4.0	E10.0
79-79	Flag for using actual shell thickness in single surface contact logic-types 4, 13, and 15. See comments below. EQ.0: logic is enabled (default), EQ.1: logic is skipped (sometimes recommended for metal forming calculations).	I1

Card 15. Computation Options—Contact (cont.)
(2E10.0,9I5,E10.0)

Columns	Quantity	Format
80-80	Time step size override for eroding contact. EQ.0: contact time size may control Δt. EQ.1: contact is not considered in Δt determination.	I1

The shell thickness change option must be active on Control Card 12 (column 30) and a nonzero flag specified in columns 26-30 above before the shell thickness changes can be included in contact types 3, 5, 10, 11, 12, 17, and 18. An additional flag must be set in columns 36-40 if thickness changes are included in types 4 and 13, i.e., the single surface contact. If the shell thickness change is active and if a nonzero flag is set in column 30 the thickness changes are automatically included in contact types, 3, 5, and 10. The new contact algorithms that include the shell thickness are relatively recent and are now fully optimized and parallelized. The searching in the new algorithms is considerably more extensive and therefore somewhat more expensive.

In the single surface contacts types 4, 13, and 15, 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.

Card 16. Computation Options—Parallel and Subcycling (4I5)

Columns	Quantity	Format
1-5	Number of CPU's for parallel execution, NCPU (default = 1)	I5
6-10	Number of right-hand sides allocated in memory: EQ.0: same as NCPU, always recommended, EQ.1: allocate only one.	I5
11-15	Subcycling flag EQ.0: off EQ.1: on	I5
16-20	Consistency flag, CONST, for parallel solution (NCPU>1) EQ.1: on EQ.2: off, for a faster solution (default)	I5
21-25	Flag for parallel force assembly if CONST=1 EQ.0: off EQ.1: on	I5

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.

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.

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.

Card 17. Computation Options—Coupling
(3E10.0,I5,2X,3I1,E10.0,6I5)

Columns	Quantity	Format
1-10	Unit conversion factor for length, UNLENG. MADYMO3D/GM-CAL3D lengths are multiplied by UNLENG to obtain LS-DYNA lengths.	E10.0
11-20	Unit conversion factor for time, UNTIME. MADYMO3D/GM-CAL3D time is multiplied by UTIME to obtain LS-DYNA time.	E10.0
21-30	Unit conversion factor for force, UNFORC. MADYMO3D/GM-CAL3D force is multiplied by UNFORC to obtain LS-DYNA force	E10.0
31-35	Material repositioning flag for MADYMO/GM-CAL3D coupling, IMOVIN. EQ.0: off EQ.1: on, section 68 input data is read. EQ.2: no repositioning of defined nodes	I5
38	Flag for flipping X-coordinate of CAL3D/MADYMO relative to the LS-DYNA model. EQ.0: off EQ.1: on	I1
39	Flag for flipping Y-coordinate of CAL3D/MADYMO relative to the LS-DYNA model. EQ.0: off EQ.1: on	I1
40	Flag for flipping Z-coordinate of CAL3D/MADYMO relative to the LS-DYNA model. EQ.0: off EQ.1: on	I1
41-50	Idle time during which CAL3D or MADYMO is computing and LS-DYNA remains inactive.	E10.0
51-55	Implicit coupling for springback calculations: EQ.0: off EQ. 1: output LS-NIKE3D (filename:NIKIN) EQ. 2: seamlessly switch to implicit springback in LS-DYNA EQ. 10: output LS-DYNA (filename:DYNAIN) EQ.100: output NASTRAN (filename:NASTIN)	I5
56-60	USA coupling option EQ.0: off EQ.1: on	I5

Columns	Quantity	Format
61-65	Number of time steps for CAL3D/MADYMO subcycling EQ.0: no subcycling EQ.n: there is one CAL/MADYMO time step for every n LS-DYNA time steps	I5
66-70	Number of ships in MCOL coupling, NMCOL	I5
71-75	Number of vehicles initialized with kinematical info, NVHINI	I5
76-80	CFD coupling option, ICFD EQ.0: off EQ.1: FAST3D CFD analysis EQ.2: Boundary Element Method	I5

Card 18. Computation Options—Output Control
(9I5,E10.0, 2I5)

Columns	Quantity	Format
1-5	Print suppression during input phase flag, NPOPT EQ.0: no suppression EQ.1: nodal coordinates, element connectivities, rigid wall definition and initial velocities are not printed	I5
6-10	Printout flag for element time step on the first cycle, IETSPF EQ.0: no printout EQ.1: the governing time step sizes for each element are printed	I5
11-15	Problem status report interval steps, IKEDIT EQ.0: default set to 100	I5
16-20	Number of time steps between restart dump files, IRDECK EQ.0: default set to 100000000	I5
21-25	Number of time steps between running restart dump files, NCBRRF. (The same file is overwritten). EQ.0: default set to 100000000	I5
26-30	Node and element suppression flag for echo file, NEECHO 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	I5
31-35	Debug option EQ.0: no printout EQ.1: progress of input phase is tracked in message file	I5
36-40	Flag to update reference node coordinates for beam element, NREFUP. This option requires that each reference node is unique to the beam. The purpose of this is for the display of beams in the post-processor in a way that show their orientation. EQ.0: off EQ.1: on	I5
41-45	Averaged accelerations from velocities in file “nodout” and the time history database file “d3thdt”, IACCOP EQ.0: no average (default) EQ.1: averaged between output intervals	I5
46-55	Output interval for interface file (Δt), OPIFS	E10.0

Columns	Quantity	Format
56-60	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 (material) definition section. This option is meant to reduce the file size 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	I5
61-65	Spring forward nodal force file. This option is to ouput resultant nodal force components of sheet metal at the end of the forming simulation into an ASCII file, "SPRINGFORWARD", for spring forward and die corrective simulations. EQ.0: off EQ.1: output element nodal force vector for deformable nodes EQ.2: output element nodal force vector for materials subset listed in Section 67.	I5
66-70	Word size of the binary 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	I5
71-75	Interval of for flushing I/O buffers EQ.0: default set to 5000	I5

Card 19. Computation Options—Output Energy
(4I5)

Columns	Quantity	Format
1-5	Hourglass energy calculation option, IEHGC. This option requires significant additional storage and increases cost by ten percent EQ.0: the default is set to 1 EQ.1: hourglass energy is not computed (default) EQ.2: hourglass energy is computed and included in the energy balance	I5
6-10	Stonewall energy dissipation option, NRWO EQ.0: the default is set to 2 EQ.1: energy dissipation is not computed EQ.2: energy dissipation is computed and included in the energy balance (default)	I5
11-15	Sliding interface energy dissipation option EQ.0: the default is set to 1 EQ.1: energy dissipation is not computed EQ.2: energy dissipation is computed and included in the energy balance	I5
16-20	Rayleigh damping energy dissipation option EQ.0: the default is set to 1 EQ.1: energy dissipation is not computed EQ.2: energy dissipation is computed and included in the energy balance	I5

Card 20. Computation Options—LS-TAURUS Database Control I (3E10.0,9I5)

Columns	Quantity	Format
1-10	Time interval between complete state dumps and interface force databases, PLTC. (This interval must be defined, otherwise the time interval will be equal to the time step size).	E10.0
11-20	Time interval between dumps of interface force database, FRCINT. If zero, the default is the same as for complete state dumps.	E10.0
21-30	Time interval between dumps of time history data, PRTC*	E10.0
31-35	Number of nodal time history blocks, NDTH	I5
33-40	Number of hexahedron elements time history blocks, NSTH	I5
41-45	Number of beam element time history blocks, NSTB	I5
46-50	Number of shell element time history blocks, NSTS	I5
51-55	Number of thick shell element time history blocks, NSTT	I5
56-60	Optional load curve specifing time interval between complete state database dumps	I5
61-65	Optional load curve specifing time interval between interface force database dumps	I5
66-70	Optional load curve specifing time interval between timehistory database dumps	I5
71-75	Flag, user output into shell resultant vector: EQ.1: off (default) EQ.2: on, user subroutine is called.	I5
76-80	NSS, number of subsystems for the subsystem statistics file, SSSTAT, See Section 87.	I5

*The time interval between dumps of time history data refers to the output frequency for the file specified by the “F=” parameter of the LS-DYNA command line. Only a subset of the nodes and elements is output as specified by the node and element print blocks, referred to by the five entries following on this card, which are read in if the appropriate keyword flags are set.

Card 21. Computation Options—LS-TAURUS Database Control II (16I5)

Columns	Quantity	Format
1-5	LS-TAURUS database during dynamic relaxation option, IDRINT EQ.0: database is not written EQ.1: LS-TAURUS database state for each convergence check EQ.n: LS-TAURUS database state written every nth convergence check.	I5
6-10	Number of additional integration point history variables written to the LS-TAURUS database for solid elements, NEIPH	I5
11-15	Number of additional integration point history variables written to the TAURUS database for shell elements for each integration point, NEIPS	I5
16-20	Number of shell integration points written to the TAURUS database, MAXINT (default = 3). See Notes below.	I5
21-25	FLAG = 1 to dump: 1. Two shell strain tensors at inner and outer integration points 2. Solid element strain tensor at centroid. for plotting by LS-TAURUS and ASCII file ELOUT.	I5
26-30	Flag for including stress tensor in the shell TAURUS database EQ.1: include (default) EQ.2: exclude	I5
31-35	Flag for including effective plastic strains in the shell database EQ.1: include (default) EQ.2: exclude	I5
36-40	Flag for including stress resultants in the shell database EQ.1: include (default) EQ.2: exclude	I5
41-45	Flag for including internal energy and thickness in the database EQ.1: include (default) EQ.2: exclude	I5
46-50	Every plot state for "d3plot" database is written to a separate file, IEVERP. 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	I5
51-55	Orthotropic and anisotropic material stress output in local	I5

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.

Card 21. Computation Options—TAURUS Database Control II (Cont.) (16I5)

Columns	Quantity	Format
56-60	Number of beam integration points for output, MAXINTB. See note below.	I5
61-65	Write to extra time history file (XTFILE) EQ.0: File not written EQ.1: File written at same interval as D3THDT file	I5
66-70	Data compression to eliminate shell rigid body data. EQ.1: off EQ.2: on	I5
71-75	Output shell hourglass energy EQ.1: off EQ.2: on	I5
76-80	Output shell element time step EQ.1: off EQ.2: on	I5

The option to control the shell output requires the version of LS-TAURUS from January 1992 or later. The variable MAXINT controls the number of integration point stresses that are output for each shell element.

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-TAURUS 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.

Beam stresses are output to the LS-TAURUS database if and only if MAXINTB 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.

The data compression of rigid body data (columns 66-70) can reduce the size of the state database in the D3PLOT files by a factor of 2 or 3 in metal forming simulations. Post-processors, other than LS-TAURUS, may need to be updated to read the compressed database.

Card 22. Computation Options—ASCII Output Control I (8E10.0)

Define Control Cards 22-24 in all cases. If the output intervals are defined as 0.0 no output is provided for the corresponding file.

Columns	Quantity	Format
1-10	Output interval for cross-section forces	E10.0
11-20	Output interval for rigid wall forces	E10.0
21-30	Output interval for nodal point data	E10.0
31-40	Output interval for element data	E10.0
41-50	Output interval for global data and subsystems statistics	E10.0
51-60	Output interval for discrete elements	E10.0
61-70	Output interval for material energies	E10.0
71-80	Output interval for nodal interface forces	E10.0

Card 23. Computation Options—ASCII Output Control II
(8E10.0)

Columns	Quantity	Format
1-10	Output interval for resultant interface forces	E10.0
11-20	Output interval for smug animator instant	E10.0
21-30	Output interval for spc reaction forces	E10.0
31-40	Output for nodal constraint resultants (spotwelds and rivets)	E10.0
41-50	Output interval for airbag statistics	E10.0
51-60	Output interval for AVS database	E10.0
61-70	Output interval for nodal force groups	E10.0
71-80	Output interval for boundary condition forces and energy on nodal points with discrete forces, pressures, or designated velocities.	E10.0

Card 24. Computation Options—ASCII Output Control III
(7E10.0)

Columns	Quantity	Format
1-10	Output interval for rigid body data	E10.0
11-20	Output interval for geometric contact entities	E10.0
21-30	Output interval for MPGS database	E10.0
31-40	Output interval for MOVIE database	E10.0
41-50	Output interval for sliding interface database	E10.0
51-60	Output interval for seat belt database	E10.0
61-70	Output interval for joint forces	E10.0
71-80	Output interval for tracer particles	E10.0

By defining the output interval a file is created for the output. Each output type is placed into a separate file. Normally these names are assigned by LS-DYNA . Using the "W=" option on startup, a root name can be specified. Extensions are then added to this root to form the output file names **(This option is available only on designated installations and is to some degree machine dependent).** The file names and corresponding unit numbers are:

	I/O UNIT #	FILE NAME
Cross-section forces	i/o unit#31	SECFORC
Rigidwall forces	i/o unit#32	RWFORC
Nodal point data	i/o unit#33	NODOUT
Element data	i/o unit#34	ELOUT
Global data	i/o unit#35	GLSTAT
Subsystems statistics	i/o unit#58	SSSTAT
Discrete elements	i/o unit#36	DEFORC
Material energies	i/o unit#37	MATSUM
Nodal interface forces	i/o unit#38	NCFORC
Resultant interface forces	i/o unit#39	RCFORC
Smug animator database	i/o unit#40	DEFGEO
Nastran/BDF file	i/o unit#49	NASBDF (see comment below)
SPC reaction forces	i/o unit#41	SPCFORC
Nodal constraint resultants (spotwelds/rivets)	i/o unit #42	SWFORC
Airbag statistics	i/o unit #43	ABSTAT
ASCII database	i/o unit #44	AVSFLT
Nodal force group	i/o unit #45	NODFOR
Boundary conditions nodal forces and energies	i/o unit #46	BNDOUT
Rigid body data	i/o unit #47	RBDOUT
Contact entities	i/o unit #48	GCEOUT
MPGS file family	i/o unit #50	MPGSnnn.xxx where nnn=001-999
MOVIE file family	i/o unit #50	MOVIEnnn.xxx where.nnn=001-999
Interface energies	i/o unit #51	SLEOUT
Seat belts	i/o unit #52	SBTOUT

	I/O UNIT #	FILE NAME
Joint forces	i/o unit #53	JNTFORC
tracer particles	i/o unit #70	TRHIST
thermal output	i/o unit #73	TPRINT (see control card 28)

The Nastran/Bulk Data File (BDF) contains all geometric information such as nodal coordinates and element connectivieis. This file is written during the input phase of LS-DYNA and can be used with the NODOUT file for animating results. An equivalent file is written for the contact surfaces call SLSBDF which contains the connectivities of the contact segments. The suffix xxx on the MPGS and MOVIE databases are descriptors such as ACL, VEL, DIS, and GEO that describe the type of data in the familied files.

Card 25. Computation Options—Arbitray Lagrangian Eulerian (3I10,5E10.0)

Columns	Quantity	Format
1-10	Default continuum treatment EQ.1: Lagrangian (default) EQ.2: Eulerian EQ.3: Arbitrary Lagrangian Eulerian EQ.4: Eulerian Ambient	I10
11-20	Number of cycles between advections.	I10
21-30	Advection method EQ.1: donor cell (first order accuracte) EQ.2: Van Leer + half index shift. EQ.3: Van Leer EQ.4: donor cell + half index shift.	I10
31-40	Smoothing weight factor - Simple average (AFAC) EQ.-1: turn smoothing off	E10.0
41-50	Smoothing weight factor - Volume weighting (BFAC)	E10.0
51-60	Smoothing weight factor - Isoparametric (CFAC)	E10.0
61-70	Smoothing weight factor - Equipotential (DFAC)	E10.0
71-80	Smoothing weight factor - Equilibrium (EFAC)	E10.0

For ALE calculations the smoothing parameters should be set. For Eulerian calculations the smoothing facter, AFAC, should be set to -1.0 which turns off the smoothing and saves computation time.

For supersonic Eulerian flows the advection formulation should be either two or three (Van Leer) since a first order solution with the much faster donor cell method may be inaccurate. The number of cycles between advections should also be set to 1. For slow speed flows this parameter may be set to a larger number to save much CPU costs.

Card 26. Computation Options—Arbitray Lagrangian Eulerian (3E10.0,I5,2E10.0,I5)

Columns	Quantity	Format
1-10	Start time for smoothing	E10.0
11-20	End time for smoothing	E10.0
21-30	ALE advection factor (Default=1.0)	E10.0
31-35	ALE analysis option, NVOIDM. See Section 75. EQ.0: single material per element EQ. n: number of single materials with possible void EQ.-n: number of multi-material Euler groups, which must not EXCEED three (3).	I5
36-45	Void factor EQ.0.0: default 1.0e-04. This is the definition of a "void". A void is obtained by multiplying the time zero density of an element by a factor call the "void factor".	E10.0
46-55	Velocity limit. The time step is scaled down if the velocities exceed this limit.	E10.0
56-60	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.	I5

Card 27. Thermal Boundary Conditions
(Input for Thermal or Coupled Structural/Thermal Analysis only-See the execution line syntax which includes the words THERMAL or COUPLE)
(I10,10I5,10x,I1)

Columns	Quantity	Format
1-10	Number of nodes with initial temperature conditions	I10
11-15	Number of elements with heat generation	I5
16-20	Number of nodes with temperature boundary conditions	I5
21-25	Number of flux boundary condition surfaces	I5
26-30	Number of convection boundary condition surfaces	I5
31-35	Number of radiation boundary condition surfaces	I5
36-40	Number of enclosure radiation surfaces	I5
41-45	Radiation calculation type EQ.1: view factors EQ.2: exchange factors (not implemented)	I5
46-50	Number of bulk nodes	I5
51-55	Number of bulk nofe segments	I5
56-60	Number of bulk fluid flow elements + number => artificial difussion is on - number => artificial difussion is off	I5
61-70	skip	10x
71	Goldak weld model 0=off 1=on	I1

Card 28. Thermal Solver and Output Controls
(Input for Thermal or Coupled Structural/Thermal Analysis only)
(I5,2E10.0,I5,2E10.0)

Columns	Quantity	Format
1-5	Solver type (isoln) EQ.1: actcol - symmetric direct solver EQ.2: dactcol - nonsymmetric direct solver EQ.3: dscg - diagonal scaled conjugate gradient iterative (default) EQ.4: iccg - incomplete choleski conjugate gradient iterative	I5
6-15	CG convergence tolerance (if isoln.ge.3) EQ.0: default set to 1000*machine roundoff suggested range is 1.e-04 to 1.e-06	E10.0
16-25	Output time interval for ascii print file TPRINT Data is written to the TPRINT file based on the thermal analysis time. EQ.0: no data is written to TPRINT file.	E10.0
26-30	Element integration quadrature (default=8): EQ.1: one point quadrature is used, EQ.8: eight point quadrature is used.	I5
31-40	Mechanical equivalent of heat (default = 1.)	E10.0
41-50	Fraction of plastic work converted to heat (default = 1.)	E10.0

Card 29. Thermal Time Step Controls
(Input for Thermal or Coupled Structural /Thermal Analysis only)
(2I5,6E10.0)

Columns	Quantity	Format
1-5	Analysis type EQ.0: steady state EQ.1: transient	I5
6-10	Time step code (0=fixed, 1=variable)	I5
11-20	Time intergration parameter EQ.0.5 - Crank Nicolson EQ.1.0 - fully implicit (default)	E10.0
21-30	Initial thermal time step	E10.0

Define the following if the variable time step option is specified above:

Columns	Quantity	Format
31-40	Minimum time step used (default explicit time step)	E10.0
41-50	Maximum time step used (default 100 x explicit time step)	E10.0
51-60	Maximum temperature change in each time step above which the time step will be decreased. EQ.0.0 - default set to 1.0	E10.0
61-70	Time step control parameter $0. < p \leq 1.$ EQ.0.0 - default set to 0.5	E10.0

Card 30. Thermal Nonlinear Problem Controls
(Input for Thermal or Coupled Structural/Thermal Analysis only)
(2I5, 5X,2E10.0)

Columns	Quantity	Format
1-5	Type of problem EQ.0: linear problem EQ.1: nonlinear problem - material properties evaluated at gauss point temperature EQ.2: nonlinear problem - material properties evaluated at element average temperature	I5
6-10	Maximum number of reformations per time step EQ.0: default set to 10	I5
11-15	Blank	5X
16-25	Convergence tolerance EQ.0: default set to 1000*machine roundoff	E10.0
26-35	Divergence control parameter steady state $0.3 \le p \le 1.$ default 1.0 transient $0.0 < p \le 1.$ default 0.5	E10.0

Card 31. Implicit Control Card 1: General Data
(Input for Implicit Analysis only, code version "i" on title card)
(I5,E10.0)

Columns	Quantity	Format
1-5	Implicit/Explicit switching flag EQ. 0: explicit analysis EQ. 1: implicit analysis EQ. 2: explicit followed by one implicit step (*"springback analysis"*)	I5
6-15	Initial time step size for implicit solution	E10.0
16-20	Element formulation switching flag EQ. 1: switch to fully integrated formulation for implicit phase of springback analysis (DEFAULT) EQ. 2: retain original element formulation	I5
21-25	Number of steps for nonlinear springback EQ. 0: DEFAULT = 1	I5
26-30	Geometric (initial stress) stiffness flag EQ. 1: include EQ. 2: ignore (DEFAULT)	I5

Card 32. Implicit Control Card 2: Nonlinear Solver
(Input for Implicit Analysis only, code version "i" on title card)
(3I5,4E10.0,3I5)

Columns	Quantity	Format
1-5	Nonlinear solution method for implicit analysis, NSOLVR 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 *(NOTE: for arclength methods see implicit control card 6 below)*	I5
6-10	Iteration limit between automatic stiffness reformations EQ.0: DEFAULT = 11	I5
11-15	Stiffness reformation limit per time step EQ.0: DEFAULT = 15	I5
16-25	Displacement convergence tolerance EQ.0: DEFAULT = 0.001	E10.0
26-35	Energy convergence tolerance EQ.0: DEFAULT = 0.01	E10.0
36-45	(blank)	E10.0
46-55	Line Search convergence tolerance EQ.0: DEFAULT = 0.90	E10.0
56-60	Displacement norm for convergence test EQ.1: increment vs. displacement over current step EQ.2: increment vs. total displacement (DEFAULT)	I5
61-65	Divergence flag EQ.1: reform stiffness if divergence detected (DEFAULT) EQ.2: ignore divergence during equilibrium iterations	I5

(continued on next page)

Columns	Quantity	Format
66-70	Initial stiffness formation flag EQ.1: reform stiffness at start of each step (DEFAULT) EQ.n: reform at start of every “n”th step	I5
71-75	Nonlinear solver print flag EQ.1: print iteration info to screen, message, d3hsp files EQ.2: print info to messag, d3hsp files only (DEFAULT)	I5

Card 33. Implicit Control Card 3: Linear Solver
(Input for Implicit Analysis only, code version "i" on title card)
(3I5)

Columns	Quantity	Format
1-5	Linear equation solver EQ.1: sparse, direct, automatic out-of-core (DEFAULT) EQ.2: sparse, direct, incore	I5
6-10	Linear solver print flag EQ.1: timing summary at end of output file (DEFAULT) EQ.2: timing, storage information to screen, output file	I5
11-15	Negative eigenvalue flag EQ.1: stop or retry step if negative eigenvalues detected EQ.2: print warning message, try to continue (DEFAULT)	I5

Card 34. Implicit Control Card 4: Auto Time Step Control
(Input for Implicit Analysis only, code version "i" on title card)
(3I5,2E10.0)

Columns	Quantity	Format
1-5	Auto time step control flag EQ.0: constant time step size EQ.1: automatically adjusted step size	I5
6-10	Optimum iteration count per time step EQ.0: DEFAULT = 11	I5
11-15	Allowable iteration window EQ.0: DEFAULT = 5	I5
16-25	Minimum time step size EQ.0: DEFAULT = 0.001 * DT	E10.0
26-35	Maximum time step size EQ.0: DEFAULT = 10 * DT	E10.0
36-45	Blank	10X
46-50	Artificial Stabilization Flag, ASFLAG EQ.1: active for all deformable shell elements (DEFAULT for *springback analysis*) EQ.2: inactive (DEFAULT for *standard analysis*)	I5
51-60	Scale factor for Artificial Stabilization LT.0.0: absolute value gives load curve for scale factor vs. time EQ.0.0: DEFAULT = 1.0	E10.0
61-70	Time when Artificial Stabilization begins EQ.0.0: DEFAULT = immediately upon entering IMPLICIT mode	E10.0
71-80	Time when Artificial Stabilization ends EQ.0.0: DEFAULT = termination time	E10.0

Card 35. Implicit Control Card 5. Dynamics
(Input for Implicit Analysis only, code version "i" on title card)

Columns	Quantity	Format
1-5	IMASS, flag for implicit solution type EQ.0: static EQ.1: dynamic (Newmark method)	I5
6-15	Newmark parameter GAMMA EQ.0: DEFAULT = 0.50	E10.0
16-25	Newmark parameter BETA EQ.0: DEFAULT = 0.25	E10.0

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.

Card 36. Implicit Control Card 6. Arc Length
(Input for Implicit Analysis only, code version “i” on title card)

The following parameters are ignored unless an arc length method is chosen on implicit control card 2 above ($6 \leq$ NSOLVR ≤ 9).

Columns	Quantity	Format
1-5	Arc length controlling node ID, ARCCTL EQ.0: generalized arc length method (DEFAULT)	I5
6-10	Arc length controlling node direction (ignored if ARCCTL=0) EQ.1: global X-translation EQ.2: global Y-translation EQ.3: global Z-translation	I5
11-20	Arc length size EQ.0.0: automatically chosen using initial step size	E10.0
21-25	Arc length method EQ.1: Crisfield (DEFAULT) EQ.2: Ramm	I5
26-30	Arc length damping option EQ.1: on, oscillations in static solution are supressed EQ.2: off (DEFAULT)	I5

Remarks:

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 method cannot be used with a dynamic analysis.

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 The arc length is similar to the step size in a standard nonlinear multi-step simulation. Smaller arc length will cause more steps to be taken during the simulation.

3. Material/Part Definitions

Control Parameters for User Defined Material
(7I5)

Define the following cards if and only if **NUSRMT** (Control Card 1, column 70) is nonzero. If so, insert NUSRMT cards here.

Columns	Quantity	Format
1-5	Material type number between 41 and 50 inclusive	I5
6-10	Length of material constants array; i.e. number of constants to be read	I5
11-15	Number of history variables required (stresses are stored by default)	I5
16-20	Address of coordinate system definition in material constants array	I5
21-25	Address of bulk modulus in material constants array	I5
26-30	Address of shear modulus in material constants array	I5
31-35	Vectorization flag (on=1)	I5

One card is required for each user defined material subroutine. The number of history variables is arbitrary and can be any number greater than or equal to 0. The coordinate system definition is optional but is probably necessary if the model involves materials that have directional properties such as composites and anisotropic plasticity models. When the coordinate system option is used then all data passed to the constitutive model is in the local system. A bulk modulus and shear modulus are required for transmitting boundaries, contact interfaces, rigid body constraints, and time step size calculations. The number of constants read in columns 6-10 includes the eight values for the coordinate system option if it is nonzero and two values for the bulk and shear modulus.

For the user defined material model 42 (planar anisotropic plasticity included in LS-DYNA) this card should appear in a (6I5) field as:

42 24 1 17 3 4

Load/Table Definitions (NLCUR<0)

If NLCUR is less than zero move Section 23 data to this location. Arbitrary load curve ID's can then be used throughout the input file.

Default Parameters for Automatic Contact (7E10.0)

Define the following card if and only if **MTCNTC** (Control Card 13, column 70) is equal to one. These parameters are used if and only if the MLARG option is on.

Columns	Quantity	Format
1-10	Default static coefficient of friction	E10.0
11-20	Default dynamic coefficient of friction	E10.0
21-30	Default default exponential decay coefficient	E10.0
31-40	Default viscous friction coefficient	E10.0
41-50	Optional default contact thickness	E10.0
51-60	Optional default thickness scale factor	E10.0
61-70	Optional default local penalty scale factor	E10.0

INPUT SECTION FOR IARB=0 (CONTROL CARD 11, COLUMN 15)

Part Definition Control Cards (IARB=0)

Repeat the following card sets for a total of NMMAT (Control Card 1, columns 1-10) sets. A set consists of the constitutive model definition, equation-of-state if required, and cross-sectional properties. The input for the case where IARB=1 is described before the material model input which starts on page 3.1.1m.

For the "MLARG" format two cards are used. The material ID is the first item followed by parameters for the automatic contact. The second card is as for the small format with columns 1 to 5 blank.

Additional Card for MLARG option (I10,7E10.0)

Columns	Quantity	Format
1-10	Material identification number (Part ID) LE: NMMAT if consecutive node, element and material IDs are used. i.e., NSORT (control card 11, column 10) equals zero LE: 99999999 if arbitrary node, element, and material IDs are used, i.e., NSORT equals 1 for arbitrary numbering	I10

The following parameters may be defined by part ID for automatic single surface contact:

Columns	Quantity	Format
11-20	Static coefficient of friction EQ:0.0: set to default	E10.0
21-30	Dynamic coefficient of friction EQ:0.0: set to default	E10.0
31-40	Exponential decay coefficient EQ:0.0: set to default	E10.0
41-50	Viscous friction coefficient EQ:0.0: set to default	E10.0
51-60	Optional contact thickness EQ: 0.0: set to default	E10.0
61-70	Optional thickness scale factor EQ:0.0: set to default	E10.0
71-80	Local penalty scale factor EQ:0.0: set to default	E10.0

Card 1 (I5,I2,I3,E10.0,2I5,E10.0,I5,2E10.0,3I5) or
(5X,I2,I3,E10.0,2I5,E10.0,I5,2E10.0,3I5) for MLARG

Columns	Quantity	Format
1-5	Part (material) identification number LE: NMMAT if consecutive node, element and material IDs are used. i.e., NSORT (control card 11, column 10) equals zero LE: 99999 if arbitrary node, element, and material IDs are used, i.e., NSORT equals 1 for arbitrary numbering	I5
6-7	Print flag for RBDOUT and MATSUM files. If undefined the default value is taken from control card 18, columns 56-60. This flag is ignored if the RBDOUT and MATSUM files are not printed. EQ.0: write part data into both MATSUM and RBDOUT EQ.1: write part data into RBDOUT file only EQ.2: write part data into MATSUM file only EQ.3: do not write part data into RBDOUT and MATSUM	I2
8-10	Material type, MT. The numbers in brackets identify the element types for which the material is available: 0-solids, 1H-Hughes-Liu beam, 1B-Belytschko beam,1I-Belytschko integrated beams, 1T-truss, 1D-discrete beam, 2-shells, and 3-thick shells. EQ.1: elastic/simple fluid [0,1H, 1B, 1I, 1T,2,3] EQ.2: orthotropic elastic [0,2,3] EQ.3: kinematic/isotropic plasticity [0,1H, 1I, 1T,2,3] EQ.4: thermo-elastic-plastic [0,1H,2,3] EQ.5: soil and crushable foam [0] EQ.6: linear viscoelastic [0,1H] EQ.7: rubber [0,2] EQ.8: high explosive burn [0,1H,1B,1T,2] EQ.9: hydrodynamic without deviatoric stresses [0, 1H,2] EQ.10: elastoplastic hydrodynamic [0] EQ.11: temperature dependent elastoplastic [0] EQ.12: isotropic elastoplastic [0,2,3] EQ.13: isotropic elastoplastic with failure [0] EQ.14: soil and crushable foam with failure [0] EQ.15: Johnson/Cook plasticity model [0,2] EQ.16: pseudo TENSOR geological model [0] EQ.17: elastoplastic with fracture [0] EQ.18: power law isotropic plasticity [0,1H,2] EQ.19: strain rate dependent plasticity [0,2,3] EQ.20: rigid [0,1H,1B,1T,2,3] EQ.21: thermal orthotropic with 12 constants [0,2,3] EQ.22: composite damage model [0,2,3] EQ.23: thermal orthotropic with 12 curves [0,2,3] EQ.24: piecewise linear isotropic plasticity [0,1H,2,3]	I3

Columns	Quantity	Format
	EQ.25: inviscid, two invariant geologic cap model [0] EQ.26: orthotropic crushable model [0] EQ.27: Mooney-Rivlin rubber [0,2] EQ.28: resultant plasticity [1B,2] EQ.29: force limited resultant formulation [1B] EQ.30: closed form update shell plasticity [2,3] EQ.31: Frazer-Nash rubber model [0] EQ.32: composite glass model [2,3] EQ.33: Barlat anisotropic plasticity model [0,2,3] EQ.34: fabric model [2] EQ.35: isotropic/kinematic hardening Green-Naghdi rate [0] EQ.36: 3-parameter Barlat plasticity [2] EQ.37: anisotropic plasticity [2,3] EQ.38: compressible foam rubber [0,2] EQ.39: anisotropic plasticity with FLD[2,3] EQ.41-50: user defined materials EQ.42: user defined planar anisotropic plasticity [2,3] EQ.51: temperature and rate dependent plasticity [2,3] EQ.52: Sandia's damage model [2,3] EQ.53: low density closed cell polyurethane foam [1] EQ.54: Composite damage with Chang matrix failure [2] EQ.55: like 54 but with Tsay-Wu criterion for matrix failure [2] EQ.57: low density urethane foam [0] EQ.59: composite failure [0,2] EQ.60: viscous glass [0] EQ.61: Maxwell/Kelvin viscoelastic [0] EQ.62: viscous foam model [0] EQ.63: isotropic crushable foam[0] EQ.64: rate sensitive powerlaw plasticity [0] EQ.65: modified Zerilli-Armstrong [0,2] EQ.66: linear stiffness/viscous 3D discrete beam [1D] EQ.67: nonlinear stiffness/nonlinear viscous 3D discrete beam [1D] EQ.68: nonlinear plastic/linear viscous 3D discrete beam [1D] EQ.69: Sid Impact Dummy damper [1D] EQ.70: hydraulic/gas damper [1D] EQ.71: cable [1D] EQ.72: concrete damage [0] EQ.75: Bilhku/Dubois foam [0] EQ.76: General viscoelastic [0,2] EQ.77: hyperelastic rubber [0] EQ.78: soil and concrete [0] EQ.79: hysteretic soil [0] EQ.80: Ramberg-Osgood soil [0] EQ.81: plastic with damage [2,3]	

Columns	Quantity	Format
	EQ.86: orthotropic viscoelastic [2] EQ.87: cellular rubber [0] EQ.88: MTS [0,2] EQ.90: acoustic media such as air or water [0] EQ.96: brittle damage model [0] EQ.100: spot weld[9B] EQ.103: anisotropic viscoplastic [0,2] EQ.126: orthotropic crushable model [0] EQ.134: viscoelastic fabric [2]	
11-20	Mass density	E10.0
21-25	Equation-of-state type. Define for material types 8, 9, 10, 11, 15, 16, 17, 65, 72, and 88 when these materials are used with solid materials. EQ.1: linear polynomial EQ.2: JWL high explosive EQ.3: Sack "Tuesday" high explosive EQ.4: Gruneisen EQ.5: ration of polynomials EQ.6: linear polynomial with energy deposition EQ.7: ignition and growth of reaction in HE EQ.8: tabulated compaction EQ.9: tabulated EQ.10: propellant deflagration EQ.11: TENSOR pore collapse EQ.14: JWLB high explosive	I5
26-30	Hourglass control type, IHQ. For shell elements the hourglass control is based on the formulation of Belytschko and Tsay, i.e., options 1-3 are identical, and options 4-5 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 The stiffness forms of the hourglass control can stiffen the response especially if deformations are large and therefore should be used with care. However, the stiffness form is often superior in reliability.	I5
31-40	Hourglass coefficient, QH (default = .10). Values of QH that exceed .15 may cause instabilities. The recommended default applies to all options.	E10.0
41-45	Bulk viscosity type, IBQ (default=1) EQ.1: standard LS-DYNA	I5

Columns	Quantity	Format
46-55	Quadratic viscosity coefficient, Q1 (default = 1.5)	E10.0
56-65	Linear viscosity coefficient, Q2, (default = .06)	E10.0
66-70	Element class for which this material model is valid EQ.0: solid hexahedron EQ.1: beam EQ.2: shell EQ.3: thick shell	I5
71-75	Material initialization for gravity loading EQ.0: all initialized EQ.1: only current material is initialized	I5
76-76	Ambient element type. (Defined for 3D solid element formulations 7, 11, and 12) EQ.1: temperature EQ.2: pressure and temperature EQ.3: pressure outflow EQ.4: pressure inflow	I1
76-76	Continuum element type. (Defined for 2D solid element formulations 13, 14, and 15) EQ.1: Lagrangian EQ.2: Eulerian, single material with voids EQ.3: ALE	I1
77-80	Element formulation if other than default. **For brick elements (types 3 and 4 have rotational degrees-of-freedom at their nodal points:** EQ.1: Constant stress (default) EQ.2: 8 point integration EQ.3: 14 point integration quadratic 8-node brick EQ.4: 5 point integration quadratic 4-node tetrahedron EQ.5: 1 point ALE EQ.6: 1 point Eulerian EQ.7: 1 point Eulerian ambient EQ.8: acoustic pressure formulation EQ.9: 1 point crushable foam element EQ.10: 1 point tetrahedron EQ.11: 1 point ALE multi-material element EQ.12: 1 point integration with single material and void.	I4

For beam and 2D shell elements:
EQ.1: Hughes-Liu (default)
EQ.2: Belytschko-Schwer
EQ.3: Truss
EQ.4: Belytschko-Schwer full integration
EQ.5: Belytschko-Schwer tubular beam
(User defined integration rule advised)
EQ.6: Discrete 3D beam (Use one point integration)
EQ.7: 2D plane strain shell (Use one point integration and x-y plane)
EQ.8: 2D axisymmetric shell (Use one point integration and y-axis of symmetry)
EQ.9: Spot weld (follow the same input as Hughes -Liu beam element. Material type 100 (only) may be used with this element)

For 3D shell elements and 2D solid elements:
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 Belyschko-Tsay membrane
EQ.10: Belytschko-Wong-Chiang
EQ.11: Corotational Hughes-Liu
EQ.12: Plane stress 2D element (x-y plane)
EQ.13: Plane strain 2D element (x-y plane)
EQ.14: Axisymmetric Petrov-Galerkin 2D solid (y-axis of symmetry)
EQ.15: Axisymmetric Galerkin 2D solid (y-axis of symmetry)
EQ.16: Fully integrated shell element (very fast)

For thick shell elements:
EQ.1: single point in plane quadrature
EQ.2: selective reduced 2×2 in plane quadrature

The Hughes-Liu elements with selective-reduced (S/R) integration are integrated with 2×2 Gaussian quadrature in the plane. These elements are quite costly (type 7 is three times greater than 1) and require much more storage. They are useful however in regions where the zero-energy modes are a problem. Option 7 is recommended over option 6 due to cost. The fully integrated membrane is slightly more expensive than one point integration but may be more stable in some special applications such as in the inflation of folded membranes.

Rayleigh Damping by Material (IARB=0)
Optional Card Defined for RDFLAG=1 On Control Card 13 (E10.0)

Columns	Quantity	Format
1-10	Rayleigh damping coefficient for stiffness proportional damping.	E10.0

Note:

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 in LS-DYNA , see Control Card 14, Columns 1-5. Transforming C with the ith eigenvector σ_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. If 10% of critical damping is sought in the ith mode using stiffness proportional damping then set:

$$\beta = \frac{.20}{\omega_i}$$

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.

Part/Material Heading (IARB=0)
Card 3 (9A8)

Columns	Quantity	Format
1-8	PSHELL (or PBAR, or PSOLID)	A8
9-33	Blank	
34-41	Property name	A8

Note:

1. This card is set up in such a way that NASBDF for SMUG post processing will be written correctly.
2. If the NASBDF file is not required, an arbitrary heading in format 9A8 can be written.

Material Parameter Cards (IARB=0)
Insert 6 Cards here (8E10.0)

Insert 6 cards here for the desired material type. The material input descriptions are found on pages Page 3.1.1m to Page 3.*nn*..1m.where *nn* refers to the material type.

Material Type i

Columns	Quantity	Format
	Card 3	8E10.0
	Card 4	8E10.0
	Card 5	8E10.0
	Card 6	8E10.0
	Card 7	8E10.0
	Card 8	8E10.0

Solid Element ALE Section Heading (IARB=0)
Define for ALE, Eulerian, or Eulerian ambient Elements Only

Columns	Quantity	Format
1-72	Section identification	12A6

Solid Element ALE Section Parameter Cards (IARB=0)
Defined for ALE, Eulerian, or Eulerian ambient Elements Only
Card 1 (4E10.0)

Columns	Quantity	Format
1-10	Smoothing weight factor - Simple average (AFAC) EQ.-1: turn smoothing off	E10.0
11-20	Smoothing weight factor - Volume weighting (BFAC)	E10.0
21-30	Smoothing weight factor - Isoparametric (CFAC)	E10.0
31-40	Smoothing weight factor - Equipotential (DFAC)	E10.0

Card 2 (3E10.0)

Columns	Quantity	Format
1-10	Start time for smoothing	E10.0
11-20	End time for smoothing	E10.0
21-30	ALE advection factor	E10.0

Solid Element Equation of State Heading (IARB=0)

Columns	Quantity	Format
1-72	Equation-of-state identification	12A6

Solid Element Equation of State Parameters (IARB=0)
Number of cards depend on the Equation of State type.

Insert cards here for the specified equation-of-state type. The input descriptions are found on pages 3.1.1e to 3.11.1e which follows page 3.60.1m.

Columns	Quantity	Format
	Card 3	E10.0
	Card 4 Define if required	E10.0
	Card 5 Define if required	E10.0

Structural Beam and Shell Section Heading Card (IARB=0)
Insert 1 Card Here(12A6)
Defined for Beam and Shell Elements Only

Columns	Quantity	Format
1-72	Cross section identification	12A6

Section Parameters for Beam Elements (IARB=0)
Cross Section Card 1 of 2 (4E10.0)
Defined for Beam and Truss Elements Only

Columns	Quantity	Format
1-10	Shear factor, default = 1.0	E10.0
11-20	Quadrature rule for beam cross section. See Figure 3.1 below. 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 EQ.-n: where |n| is the number of the user defined rule	E10.0
21-30	Cross section type for Hughes-Liu beam (BCST) EQ.0.0: rectangular EQ.1.0: tubular EQ.2.0: arbitrary (user defined integration rule)	E10.0
31-40	Location of triad for tracking the rotation of the discrete beam element. The force and moment resultants in the output databases are referenced to this triad. Skip this input for other beam elements. 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.	E10.0

Note: This card is blank for the type 2 Belytschko-Schwer beam only.

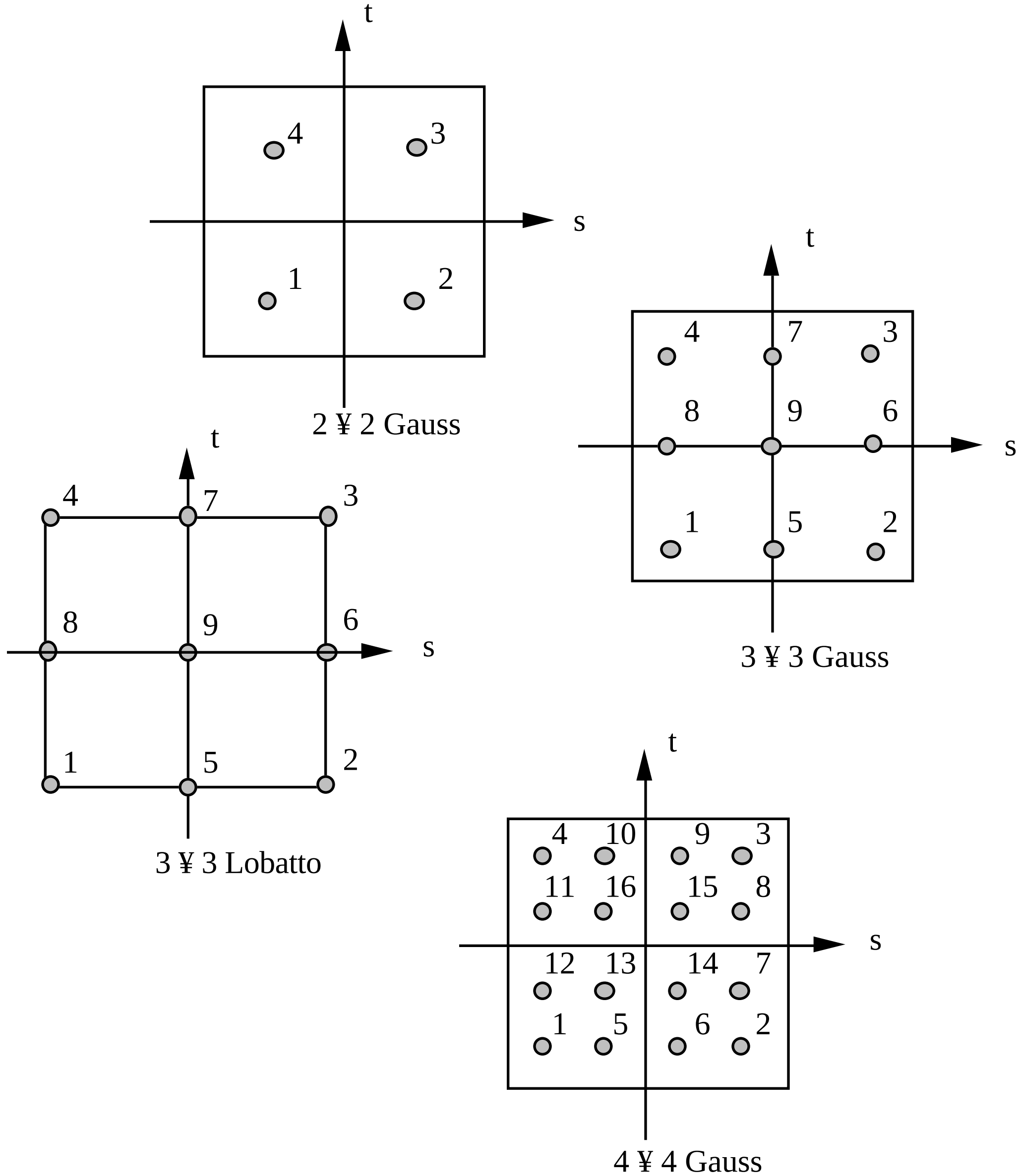


Figure 3.1. Location of the integration points in the Hughes-Liu beam element.

Beam Cross Section Card 2 of 2 (6E10.0)
Defined for the Types 1, 4, 5, 7, 8, and 9 Beam Elements

Columns	Quantity	Format
1-10	Beam thickness (BCST=0.0, 2.0) or outer diameter (BCST = 1.0) in s direction at node n_1 see Figure 3.2)	E10.0
11-20	Beam thickness (BCST=0.0, 2.0) or outer diameter (BCST = 1.0) in s direction at node n_2	E10.0
21-30	Beam thickness (BCST=0.0, 2.0) or inner diameter (BCST = 1.0) in t direction at node n_1	E10.0
31-40	Beam thickness (BCST=0.0, 2.0) or inner diameter (BCST = 1.0) in t direction at node n_2	E10.0
41-50	Location of reference surface normal to s axis (Hughes-Liu only) EQ.1.0: side at s=1 EQ.0.0: center EQ.-1.0: side at s= -1.0	E10.0
51-60	Location of reference surface normal to t axis (Hughes-Liu only) EQ.1.0: side at t=1 EQ.0.0: center EQ.-1.0: side at t=-1.0	E10.0

Beam Cross Section Card 2 of 2 (5E10.0)
Defined for Beam Types 2 and 3 only

Columns	Quantity	Format
1-10	Cross-sectional area, A	E10.0
11-20	I_{ss}	E10.0
21-30	I_{tt}	E10.0
31-40	I_{rr} (J)	E10.0
41-50	Shear area, A_s	E10.0

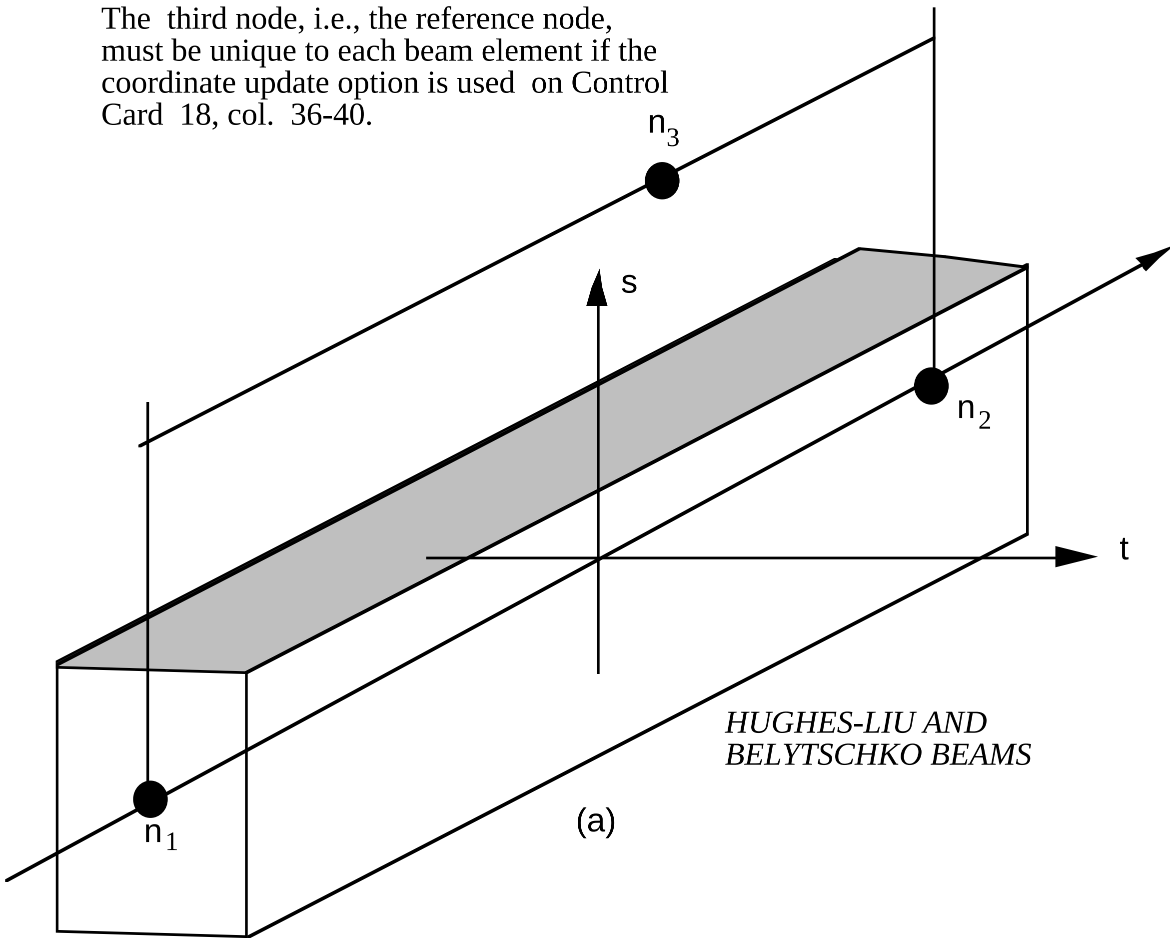
For the truss element, define the cross-sectional area, A, and leave columns 11- 80 blank.

Beam Cross Section Card 2 of 2 (8E10.0)
Define for the Discrete 3D Beam (Type 6 only)

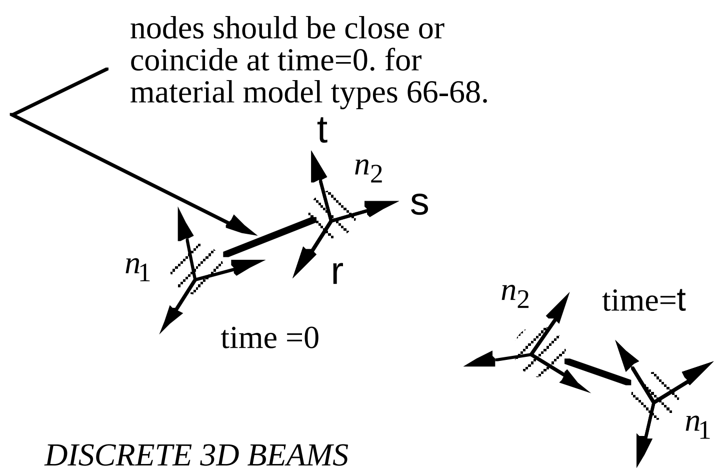
Columns	Quantity	Format
1-10	Volume of discrete beam	E10.0
11-20	I, lumped inertia of discrete beam	E10.0
21-30	Coordinate system ID for orientation, materials type ID (67-69) EQ.0: local coordinate system is aligned with global axes	E10.0
31-40	Cable area, material type ID (71)	E10.0
41-50	Offset for cable, material type ID (71)	E10.0
51-60	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	E10.0
61-70	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	E10.0
71-80	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	E10.0

Negative values for the cable offset will make the cable slack in its initial configuration. Positive values will induce a tensile force when the calculation begins. Negative offset values that exceed the length of the cable will reset internally to the cable length. Nodal masses are found by dividing the product of the volume and the density of the element by the number of nodal points defining the connectivity.

The local coordinate system rotates as the nodal point that define the beam rotate. In some cases this may lead to unexpected results if the nodes undergo significant rotation.


The third node, i.e., the reference node,
must be unique to each beam element if the
coordinate update option is used on Control
Card 18, col. 36-40.
n3
r
s
n2
t
HUGHES-LIU AND
BELYTSCHKO BEAMS
n1

(a)


nodes should be close or
coincide at time=0. for
material model types 66-68.
t
n2
s
n1
r
time =0
n2
time=t
n1
DISCRETE 3D BEAMS

(b)

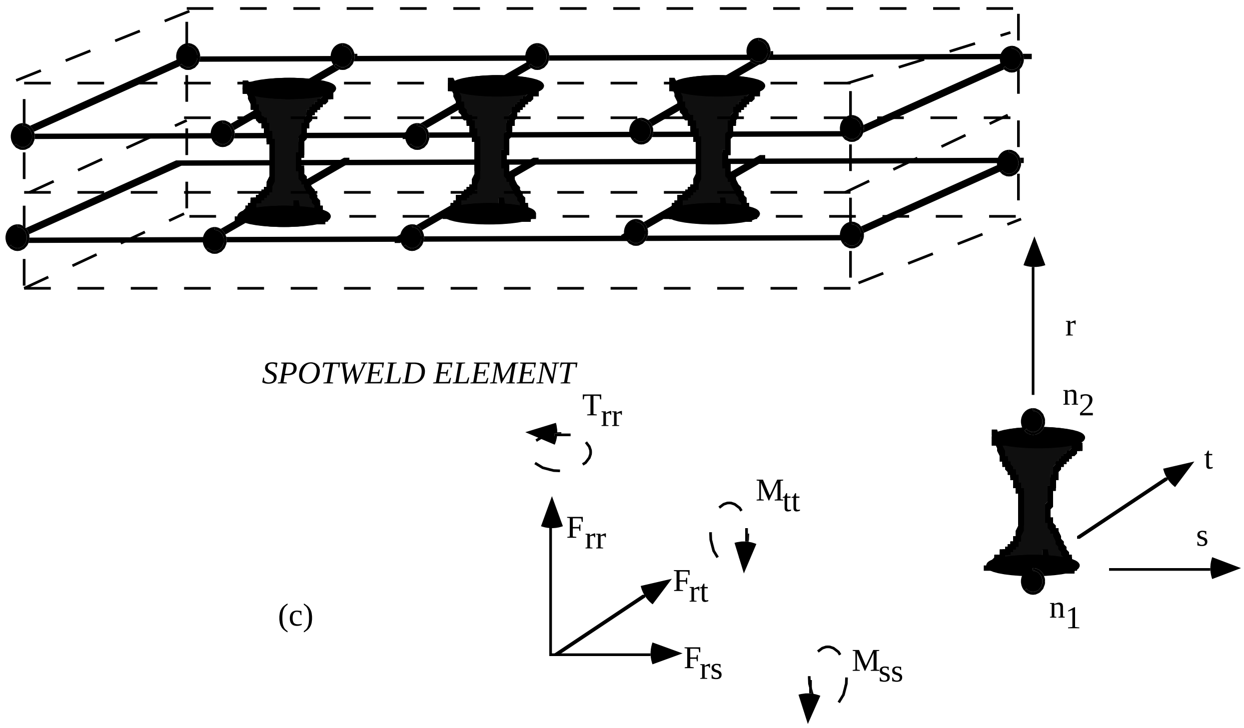
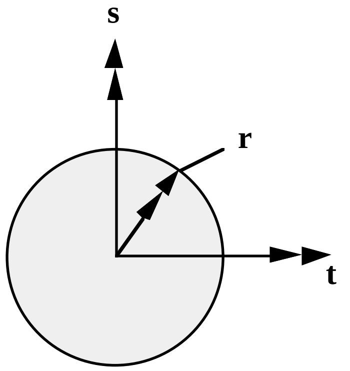
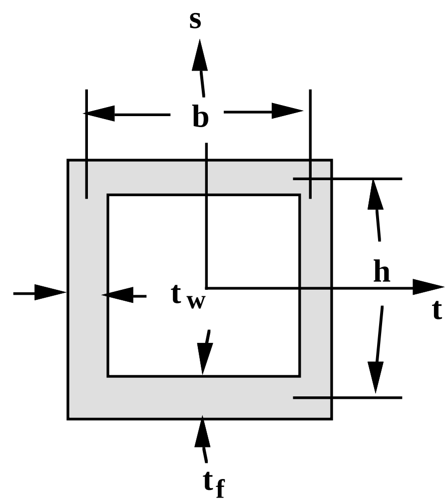


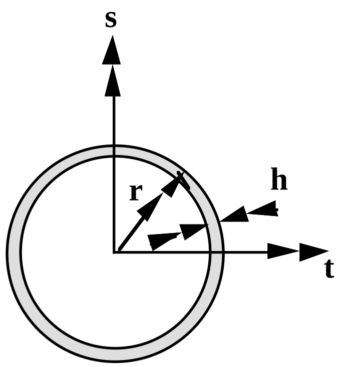
Figure 3.2. LS-DYNA beam elements: (a) 2-noded beam element, (b) discrete beam element, and (c) spot weld element. The beam spot weld elements must originate at the shell mid-surface for automatic tying with type 7 constraint contact. If the spot weld cross section is circular the orientation node is optional though necessary if the s and t directions are important. Attachments of the spot weld elements to rigid body nodes are permitted but not with type 7 constraint contact.



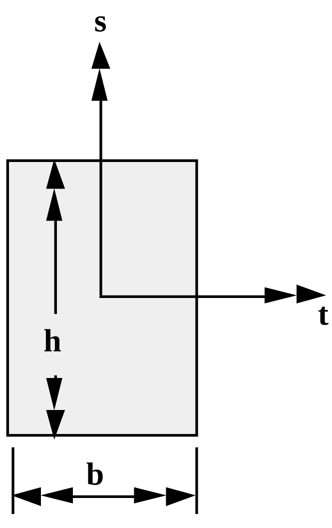
$$I_{tt} = I_{ss} = \frac{\pi r^4}{4}$$
$$J = \frac{\pi r^4}{2}$$
$$f_{tt} = f_{ss} = \frac{10}{9}$$
$$A = \pi r^2$$



$$I_{tt} \cong \left(\frac{h^2}{6}\right)(h t_w + 3 b t_f)$$
$$I_{ss} \cong \left(\frac{b^2}{6}\right)(b t_f + 3 h t_w)$$
$$J \cong \frac{2 b^2 h^2 (t_w t_f)}{(b t_w + h t_f)}$$
$$f_{tt} = \frac{A}{[2(b + t_w) t_f]}$$
$$f_{ss} = \frac{A}{[2(h + t_f) t_w]}$$
$$A = 2(b t_f + h t_w)$$



$$I_{tt} = I_{ss} \cong \pi r^3 h$$
$$J \cong 2\pi r^3 h$$
$$f_{tt} = f_{ss} = 2$$
$$A \cong 2\pi r h$$



$$I_{tt} = \frac{bh^3}{12}$$
$$I_{ss} = \frac{b^3 h}{12}$$
$$J \cong \left[\frac{1}{3} - .21\left(\frac{b}{h}\right)\left(1 - \frac{b^4}{12h^4}\right)\right] hb^3$$
$$f_{tt} = f_{ss} = \frac{6}{5}$$
$$A = bh$$

Shear Are $= \frac{A}{f} = \mu A$

Figure 3.3. Properties of beam cross section for several common cross sections.

Section Parameters for Shell Elements (IARB=0) (8E10.0)
Define for Shell and Membrane Elements
Card 1 of 2

The following cards are defined for 4 node shell, 8-node brick shells, and membrane elements.

Columns	Quantity	Format
1-10	Shear factor (default = 1.0)	E10.0
11-20	Number of through shell thickness integration points, NIP The location of the Gauss integration points are listed in Table 3.1 below. EQ.1.0: 1 point (membrane) EQ.2.0: 2 point EQ.3.0: 3 point EQ.4.0: 4 point EQ.5.0: 5 point GT.5.0: trapezoidal or user defined rule	E10.0
21-30	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	E10.0
31-40	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	E10.0
41-50	Simple average weight factor for ALE element smoothing EQ.-1: turn smoothing off.	E10.0
51-60	Volume weight factor for ALE element smoothing	E10.0
61-70	Isoparametric weight factor for ALE element smoothing	E10.0
71-80	Equipotential weight factor for ALE element smoothing	E10.0

GAUSS POINT 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

Table 3.1. 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.

Shell Cross Section Card 2 of 2 (8E10.0)

Insert a blank card here if the material definition is for a thick (8-node) shell.

Columns	Quantity	Format
1-10	Shell thickness at node n_1 (See Figure 3.4)	E10.0
11-20	Shell thickness at node n_2	E10.0
21-30	Shell thickness at node n_3	E10.0
31-40	Shell thickness at node n_4	E10.0
41-50	Location of reference surface (Hughes-Liu shell theory only) EQ.1.0: top surface EQ. 0.0: midsurface (default for Hughes-Liu and mandatory for Belytschko-Tsay) EQ.-1.0: bottom surface	E10.0
41-50	Equilibrium weight factor for ALE element smoothing	E10.0
51-60	Start time for ALE element smoothing	E10.0
61-70	End time for ALE element smoothing	E10.0
71-80	ALE advection factor	E10.0

The thickness values can be overridden on the element cards; i.e. the above values are used if and only if the thickness values are zero on the element cards.

Material Angle Cards (8E10.0)

Define the following cards if required by the constitutive model. Include as many cards as necessary. Angles are in degrees.

Columns	Quantity	Format
1-10	β_1 material angle at first integration point	E10.0
11-20	β_2 material angle at second integration point	E10.0
21-30	β_3 material angle at third integration point	E10.0
.	.	.
.	.	.
.	.	.
71-80	β_8 material angle at eighth integration point	E10.0

VDA Surface for Rigid Part (IARB=0)
Optional Material Cards for VDA Surface Definition 12 (A80)

Define the following card if required by the Type 20 material input flag. See Section 3.20m.

Card 1 (A80)

Columns	Quantity	Format
1-80	VDA surface alias name (less than 12 characters)	A80

Contact Entity Mesh for Rigid Part (IARB=0)
Optional Cards for CONTACT ENTITY Mesh Generation 12 (A80)

Define the following three cards if required by the type 20 material input flag. See Section 3.20m. This data follows the Section data. *If more than one entity is defined with the part, then define 3 cards for each entity to be generated, (Card 3, Cols. 71-80 for the type 20 material definition).*

Entity Generation Card 1 of 3
(6E10.0)

Columns	Quantity	Format
1-10	x-center, x_c	E10.0
11-20	y-center, y_c	E10.0
21-30	z-center, z_c	E10.0
31-40	x-direction for local axis X′, A_x	E10.0
41-50	y-direction for local axis X′, A_y	E10.0
51-60	z-direction for local axis X′, A_z	E10.0

Entity Generation Card 2 of 3
(3E10.0)

Columns	Quantity	Format
1-10	x-direction for local axis Y′, B_x	E10.0
11-20	y-direction for local axis Y′, B_y	E10.0
21-30	z-direction for local axis Y′, B_z	E10.0

(x_c, y_c, z_c) positions the local origin of the geometric entity in global coordinates. The entity's local X′-axis is determined by the vector (A_x, A_y, A_z) and the local Y′-axis by the vector (B_x, B_y, B_z).

Cards 1 and 2 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 $(x_c, y_c, z_c)=(0,0,0)$, $X'=(A_x, A_y, A_z)=(1,0,0)$, and $Y'=(B_x, B_y, B_z)=(0,1,0)$.

Entity Generation Card 3 of 3
(2I5,7E10.0)

Columns	Quantity	Format
1-5	Entity types available for mesh generation EQ.1: infinite plane EQ.2: sphere EQ.3: infinite cylinder EQ.4: hyperellipsoid EQ.5: torus EQ.10: finite plane EQ.11: load curve defining line	I5
6-10	In-out flag (This flag may be ignored here.) EQ.0: slave nodes exist outside of the entity EQ.1: slave nodes exist inside the entity	I5
11-20	Entity coefficient g_1	E10.0
21-30	Entity coefficient g_2	E10.0
31-40	Entity coefficient g_3	E10.0
41-50	Entity coefficient g_4	E10.0
51-60	Entity coefficient g_5	E10.0
61-70	Entity coefficient g_6	E10.0
71-80	Entity coefficient g_7	E10.0

IGTYPE = 1:	g1 = Px	g4 = Qx
	g2 = Py	g5 = Qy
	g3 = Pz	g6 = Qz
	g7 = L	

A square plane of length L on each edge is generated which represents the infinite plane.

IGTYPE = 2:	g1 = Px	g4 = r
	g2 = Py	
	g3 = Pz	

IGTYPE = 3:	g1 = Px	g4 = Qx
	g2 = Py	g5 = Qy
	g3 = Pz	g6 = Qz
	g7 = r	

A cylinder of length $\sqrt{Qx^2 + Qy^2 + Qz^2}$ and radius r is generated which represents the infinite cylinder.

IGTYPE = 4:	g1 = Px	g4 = a
	g2 = Py	g5 = b
	g3 = Pz	g6 = c
	g7 = n (order of the ellipsoid, default=2)	

IGTYPE = 5:	g1 = Radius of torus
	g2 = r
	g3 = Number of elements about the minor circumference (default=10)
	g4 = Number of elements about the major circumference (default=20).

IGTYPE = 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)

IGTYPE = 9:	g1 = Shell thickness (option to override true thickness)
	g2 = Scale factor for true thickness (optional)
	g3 = Load curve ID defining thickness versus time. (optional)

IGTYPE =10:	g1 = Length of edge along X′ axis

g2 = Length of edge along Y′ axis

IGTYPE =11: g1 =Load curve ID defining axisymmetric surface profile about Z′-axis
g2 = Number of elements about the circumference (default=10)
g3 = Number of elements along the axis (default=20).

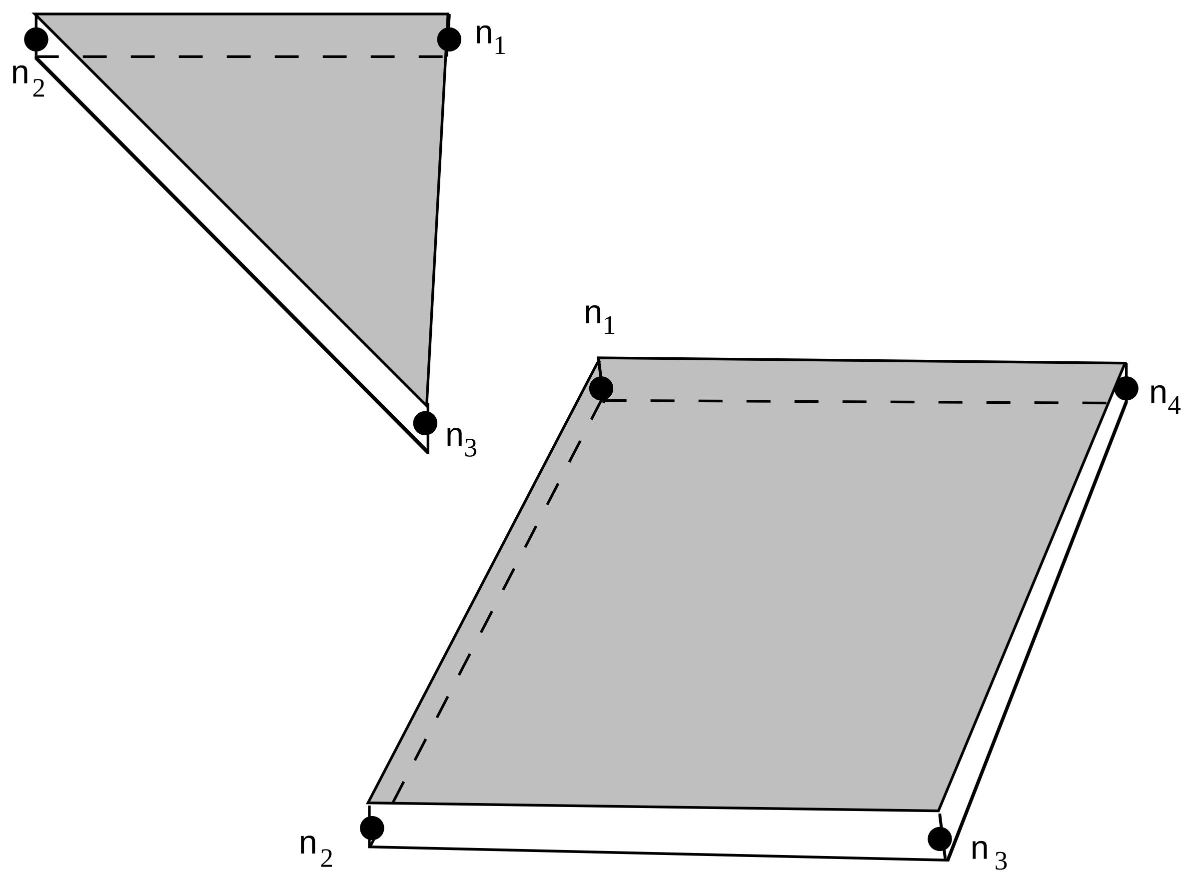


Figure 3.4. LS-DYNA shell elements. Counterclockwise node numbering determines the top surface.

Define the following three cards if required by the type 20 material input flag. See Section 3.20m. This data follows the Section data. *If more than one entity is defined with the part, then define 3 cards for each entity to be generated, (Card 3, Cols. 71-80 for the type 20 material definition).*

Optional Erosion Model For Solid Elements

Several failure criteria are available that are independent of the material models. Each one is applied independently, and once any one of them is satisfied, the element is deleted from the calculation. Individual criteria may be eliminated from the failure model by assigning the associated constants the value of the user-specified exclusion number. *This failure criterion only works with solid elements in two and three dimensions which have one point integration.*

If a user intends to alter the failure criteria in a full-deck restart, the user should specify large values for the failure criteria in the initial input so that the strains and Tuler-Butcher integral will be written to the restart files.

The criteria for failure are:

1. $P \geq P_{\min}$ where P is the pressure (positive in compression), and $P_{\min}$ is the pressure at failure.

2. $\sigma_1 \geq \sigma_{\max}$, where σ_1 is the maximum principal stress, and $\sigma_{\max}$.is the principal stress at failure.

3. $\sqrt{\frac{3}{2}\sigma'_{ij}\sigma'_{ij}} \geq \bar{\sigma}_{\max}$, where σ'_{ij} are the deviatoric stress components, and $\bar{\sigma}_{\max}$ is the equivalent stress at failure.

4. $\varepsilon_1 \geq \varepsilon_{\max}$, where ε_1 is the maximum principal strain, and $\varepsilon_{\max}$ is the principal strain at failure.

5. $\gamma_1 \geq \gamma_{\max}$, where γ_1 is the shear strain, and $\gamma_{\max}$ is the shear strain at failure.

6. The Tuler-Butcher [1968] criterion,

$$\int_0^t [\max(0, \sigma_1 - \sigma_0)]^2 \, dt \geq K_f,$$

where σ_1 is the maximum principal stress, σ_0 is a specified threshold stress, $\sigma_1 \geq \sigma_0 \geq 0$, and K_f is the stress impulse for failure. Stress values below the threshold value are too low to cause fracture even for very long duration loadings.

Failure Card 1 of 5
(E10.0)

Columns	Quantity	Format
1-10	*X*, the exclusion number.	E10.0

When any of the failure constants are set to the exclusion number, the associated failure criteria calculations are bypassed (which reduces the cost of the failure model). For example, to prevent a material from going into tension, the user should specify an unusual value for the exclusion number, e.g., 1234., set $P_{\min}$ to 0.0 and all the remaining constants to 1234. The default value is 0.0, which eliminates all criteria from consideration that have their constants set to 0.0 or left blank in the input file.

Failure Card 2 of 5
(E10.0)

Columns	Quantity	Format
1-10	$P_{\min}$, the pressure at failure	E10.0

Failure Card 3 of 5
(2E10.0)

Columns	Quantity	Format
1-10	$\sigma_{\max}$, the principal stress at failure	E10.0
11-20	$\bar{\sigma}_{\max}$, the equivalent stress at failure	E10.0

Failure Card 4 of 5
(2E10.0)

Columns	Quantity	Format
1-10	ε_{max},.the principal strain at failure	E10.0
11-20	γ_{max}, the shear strain at failure	E10.0

Failure Card 5 of 5
(2E10.0)

Columns	Quantity	Format
1-10	σ_0, specified threshold stress	E10.0
11-20	K_f , the stress impulse for failure	E10.0

INPUT SECTION FOR IARB=1 (CONTROL CARD 11, COLUMN 15)

Part Definition Control Cards (IARB=1)
(3I5)

Columns	Quantity	Format
1-5	Number of constitutive model definitions	I5
6-10	Number of equation-of-state definitions	I5
11-15	Number of cross-section property definitions	I5

Define the number of constitutive model definitions specified above. Each set consists of the following card plus cards 2-8 above corresponding to the material model type.

For the "MLARG" format two cards are used. The constitutive model ID is the first item followed by parameters for the automatic contact. The second card is as for the small format with columns 1 to 5 blank.

Additional Card for MLARG option (I10,7E10.0)

Columns	Quantity	Format
1-10	Arbitrary number, ID, identifying the constitutive model LE: NMMAT if consecutive node, element and material ISs are used. i.e., NSORT (control card 11, column 10) equals zero LE: 99999999 if arbitrary node, element, and material IDs are used, i.e., NSORT equals 1 for arbitrary numbering	I10

The following parameters may be defined by part ID for automatic single surface contact:

Columns	Quantity	Format
11-20	Static coefficient of friction EQ:0.0: set to default	E10.0
21-30	Dynamic coefficient of friction EQ:0.0: set to default	E10.0
31-40	Exponential decay coefficient EQ:0.0: set to default	E10.0

Columns	Quantity	Format
41-50	Viscous friction coefficient EQ:0.0: set to default	E10.0
51-60	Optional contact thickness EQ: 0.0: set to default	E10.0
61-70	Optional thickness scale factor EQ:0.0: set to default	E10.0
71-80	Local penalty scale factor EQ:0.0: set to default	E10.0

Card 2 (2I5,E10.0,I10,E10.0,I5,2E10.0) or (I10),(5X,I5,E10.0,I10,E10.0,I5,2E10.0) for MLARG

Columns	Quantity	Format
1-5	Arbitrary number, ID, identifying the constitutive model	I5
6-10	Material type, MT. The numbers in brackets identify the element types for which the material is available: 0-solids, 1H-Hughes-Liu beam, 1B-Belytschko beam,1I-Belytschko integrated beams, 1T-truss, 1D-discrete beam, 2-shells, and 3-thick shells. EQ. 1: elastic/simple fluid [0,1H, 1B, 1I, 1T ,2,3] EQ. 2: orthotropic elastic [0,2,3] EQ. 3: kinematic/isotropic plasticity [0,1H, 1I, 1T,2,3] EQ. 4: thermo-elastic-plastic [0,1H,2,3] EQ. 5: soil and crushable foam [0] EQ. 6: linear viscoelastic [0,1H] EQ. 7: rubber [0,2] EQ. 8: high explosive burn [0] EQ. 9: null material [0,1H,1B,1T,2] EQ.10: elastoplastic hydrodynamic [0] EQ.11: temperature dependent elastoplastic [0] EQ.12: isotropic elastoplastic [0,2,3] EQ.13: isotropic elastoplastic with failure [0] EQ.14: soil and crushable foam with failure [0] EQ.15: Johnson/Cook plasticity model [0,2] EQ.16: pseudo TENSOR geological model [0] EQ.17: elastoplastic with fracture [0] EQ.18: power law isotropic plasticity [0,1H,2] EQ.19: strain rate dependent plasticity [0,2,3] EQ.20: rigid [0,1H,1B,1T,2,3] EQ.21: thermal orthotropic with 12 constants [0,2,3] EQ.22: composite damage model [0,2,3] EQ.23: thermal orthotropic with 12 curves [0,2,3] EQ.24: piecewise linear isotropic plasticity [0,1H,2,3] EQ.25: inviscid, two invariant geologic cap model [0] EQ.26: orthotropic crushable model [0]	I5

Columns	Quantity	Format
	EQ.27: Mooney-Rivlin rubber [0,2] EQ.28: resultant plasticity [1B,2] EQ.29: force limited resultant formulation [1B] EQ.30: closed form update shell plasticity [2,3] EQ.31: Frazer-Nash rubber model [0] EQ.32: composite glass model [2,3] EQ.33: Barlat anisotropic plasticity model [0,2,3] EQ.34: fabric model [2] EQ.35: isotropic/kinematic hardening Green-Naghdi rate [0] EQ.36: 3-parameter Barlat plasticity [2] EQ.37: anisotropic plasticity [2,3] EQ.38: compressible foam rubber [0,2] EQ.39: anisotropic plasticity with FLD [2,3] EQ.41-50: user defined materials EQ.51: temperature and rate dependent plasticity [2,3] EQ.52: Sandia's damage model [2,3] EQ.53: Low Density Closed Cell Polyurethane Foam [1] EQ.54: Composite damage with Chang matrix failure [2] EQ.55: Like 54 but with Tsay-Wu criterion for matrix failure [2]. EQ.57: Low density urethane foam [0] EQ.60: viscous glass [0] EQ.61: Maxwell/Kelvin viscoelastic [0] EQ.62: viscous foam model [0] EQ.63: isotropic crushable foam[0] EQ.64: rate sensitive powerlaw plasticity [0] EQ.65: modified Zerilli-Armstrong [0,2] EQ.66: linear stiffness/viscous 3D discrete beam [1D] EQ.67: nonlinear stiffness/nonlinear viscous 3D discrete beam [1D] EQ.68: nonlinear plastic/linear viscous 3D discrete beam [1D] EQ.69: Side Impact Dummy damper [1D] EQ.70: hydraulic/gas damper [1D] EQ.71: cable [1D] EQ.72: concrete damage [0] EQ.75: Bilhku/Dubois foam [0] EQ.76: General viscoelastic [0] EQ.77: hyperelastic rubber [0] EQ.78: soil and concrete [0] EQ.79: hysteretic soil [0] EQ.80: Ramberg-Osgood soil [0] EQ.81: plastic with damage [2,3] EQ.86: orthotropic viscoelastic [2] EQ.87: cellular rubber [0] EQ.90: acoustic media such as air or water [0] EQ.96: brittle damage model [0] EQ.100: spot weld [9B] EQ.103: anisotropic viscoplastic [0,2] EQ.126: orthotropic crushable model [0] EQ.134: viscoelastic fabric [2]	

Columns	Quantity	Format
11-20	Mass density	E10.0
21-30	Hourglass viscosity type, IHQ (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 EQ. 4: Flanagan-Belytschko stiffness form EQ. 5: Flanagan-Belytschko stiffness from with exact volume integration The stiffness forms of the hourglass control can stiffen the response especially if deformations are large and therefore should be used with care. However, the stiffness form is often superior in reliability.	I10
31-40	Hourglass coefficient, QH (default = .10). Values of QH t hat exceed .15 may cause instabilities. The recommended default applies to all options.	E10.0
41-45	Bulk viscosity type, IBQ (default=1) EQ. 1: standard LS-DYNA	I5
46-55	Quadratic viscosity coefficient, Q1 (default = 1.5)	E10.0
56-65	Linear viscosity coefficient, Q2, (default = .06)	E10.0

Rayleigh Damping by Material (IARB=1)
Optional Card Defined for RDFLAG=1 (Control Card 13) (E10.0)

Columns	Quantity	Format
1-10	Rayleigh damping coefficient for stiffness proportional damping.	E10.0

See the discussion on Rayleigh damping in the IARB=0 Section above.

Part/Material Heading (IARB=1)
Insert 1 Card here (9A8)

Columns	Quantity	Format
1-8	PSHELL (or PBAR, or PSOLID)	A8
9-32	Blank	
33-41	Property name	A8

Notes:

1. This card is set up in such a way that NASBDF for SMUG post processing will be written correctly.
2. If the NASBDF file is not required, an arbitrary heading in format 9A8 can be written.

Material Parameter Cards (IARB=1)
Insert 6 Cards here (8E10.0)

Insert 6 cards here for the specified material type. The input descriptions are found on Pages 3.1.1m to 3.60.1m.

Material Type i

Columns	Quantity	Format
	Card 3	E10.0
	Card 4	E10.0
	Card 5	E10.0
	Card 6	E10.0
	Card 7	E10.0
	Card 8	E10.0

Equation of State Definitions (IARB=1)

Define the number of equation-of-state model definitions specified on the Control Card above. Each set consists of this card followed by an identification card and the input corresponding to the specified equation-of-state type.

Columns	Quantity	Format
1-5	Arbitrary number, ID, identifying Equation-of-State	I5
6-10	Equation-of-state type. Define for material types 8, 9, 10, 11, 15, 16, 17, and 18 when these materials are used with solid materials. EQ. 1: linear polynomial EQ. 2: JWL high explosive EQ. 3: Sack "Tuesday" high explosive EQ. 4: Gruneisen EQ. 5: ratio of polynomials EQ. 6: linear polynomial with energy deposition EQ. 7: ignition and growth of reaction in HE EQ. 8: tabulated compaction EQ. 9: tabulated EQ.10: propellant deflagration EQ.11: TENSOR pore collapse EQ.14: JWLB high explosive	I5

Solid Element Equation of State Heading (IARB=1)

Columns	Quantity	Format
1-72	Equation-of-state description	12A6

Solid Element Equation of State Parameters (IARB=1)
Number of cards depend on the Equation of State type.

Insert cards here for the specified equation-of-state type. The input descriptions are found on pages 3.1.1e to 3.11.1e which follows page 3.60.1m.

Columns	Quantity	Format
	Card 3	E10.0
	Card 4 Define if required	E10.0
	Card 5 Define if required	E10.0

Cross Section Definitions (IARB=1)

The section cards are to be grouped in section sets. The section sets are read after all constitutive model definition are defined. Please note that solid elements types 5, 6, and 7 need cross section definitions.

Columns	Quantity	Format
1-5	Arbitrary number, ID, identifying the cross section	I5
6-10	Element type EQ.0: solid EQ.1: beam EQ.2: shell EQ.3: thick shell	I5

Section Heading Card (IARB=1)

Columns	Quantity	Format
1-72	Cross section description	12A6

Solid Element ALE Section Card (IARB=1)1 of 2 (4E10.0)
Defined for ALE, Eulerian, or Eulerian ambient solid elements Only

Columns	Quantity	Format
1-10	Smoothing weight factor - Simple average (AFAC) EQ.-1: turn smoothing off	E10.0
11-20	Smoothing weight factor - Volume weighting (BFAC)	E10.0
21-30	Smoothing weight factor - Isoparametric (CFAC)	E10.0
31-40	Smoothing weight factor - Equipotential (DFAC)	E10.0

Solid Elelement Section Card 2 of 2 (3E10.0)
Defined for ALE, Eulerian, or Eulerian ambient solid elements Only

Columns	Quantity	Format
1-10	Start time for smoothing	E10.0
11-20	End time for smoothing	E10.0
21-30	ALE advection factor	E10.0

Beam Element Section Card (IARB=1) 1 of 2 (4E10.0)
Define for Beam and Truss Elements

Columns	Quantity	Format
1-10	Shear factor, default = 1.0	E10.0
11-20	Quadrature rule for beam cross section 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 EQ.-n: where |n| is the number of the user defined rule	E10.0
21-30	Cross section type for Hughes-Liu beam (BCST) EQ.0.0: rectangular EQ.1.0: tubular EQ.2.0: arbitrary (user defined integration rule)	E10.0
31-40	Location of triad for tracking the rotation of the discrete beam element. The force and moment resultants in the output databases are referenced to this triad. Skip this input for other beam elements. 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.	E10.0

Beam Element Section Card 2 of 2 (6E10.0)
Defined for the Types 1, 4, 5, 7, 8, and 9 Beam Elements

Columns	Quantity	Format
1-10	Beam thickness (BCST=0.0, 2.0) or outer diameter (BCST = 1.0) in s direction at node n_1 see Figure 3.2)	E10.0
11-20	Beam thickness (BCST=0.0, 2.0) or outer diameter (BCST = 1.0) in s direction at node n_2	E10.0
21-30	Beam thickness (BCST=0.0, 2.0) or inner diameter (BCST = 1.0) in t direction at node n_1	E10.0
31-40	Beam thickness (BCST=0.0, 2.0) or inner diameter (BCST = 1.0) in t direction at node n_2	E10.0
41-50	Location of reference surface normal to s axis (Hughes-Liu only) EQ.1.0: side at s=1 EQ.0.0: center EQ.-1.0: side at s=-1.0	E10.0
51-60	Location of reference surface normal to t axis (Hughes-Liu only) EQ.1.0: side at t=1 EQ.0.0: center EQ.-1.0: side at t=-1.0	E10.0

Beam Element Section Card 2 of 2 (6E10.0)
Define for the Belytschko Beam or Truss Element

Columns	Quantity	Format
1-10	Cross-sectional area, A	E10.0
11-20	I_{ss}	E10.0
21-30	I_{tt}	E10.0
31-40	I_{rr} (J)	E10.0
41-50	Shear area, A_s	E10.0

For the truss element, define the cross-sectional area, A, and leave columns 11- 80 blank.

Beam Element Section Card 2 of 2 (6E10.0)
Define for the Discrete 3D Beam

Columns	Quantity	Format
1-10	Volume of discrete beam	E10.0
11-20	I, lumped inertia of discrete beam	E10.0
21-30	Coordinate system ID for orientation, materials type ID (67-69) EQ.0: Local coordinate system is aligned with global axes.	E10.0
31-40	Cable area, material type ID (71)	E10.0
41-50	Offset for cable, material type ID (71)	E10.0
51-60	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	E10.0
61-70	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	E10.0
71-80	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	E10.0

Negative values for the cable offset will make the cable slack in its initial configuration. Positive values will induce a tensile force when the calculation begins.

Negative offset values that exceed the length of the cable will reset internally to the cable length. Nodal masses are calculated from the volume and density of the element.

The local coordinate system rotates as the nodal point that define the beam rotate. In some cases this may lead to unexpected results if the nodes undergo significant rotation.

Shell Element Section Card (IARB=1) 1 of 2 (4E10.0,I5)
Define for Shell and Membrane Elements

Columns	Quantity	Format
1-10	Shear factor (default = 1.0)	E10.0
11-20	Number of through shell thickness integration points, NIP EQ.1.0: 1 point (membrane) EQ.2.0: 2 point EQ.3.0: 3 point EQ.4.0: 4 point EQ.5.0: 5 point GT.5.0: trapezoidal or user defined rule	E10.0
21-30	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	E10.0
31-40	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	E10.0
	The following additional parameter is defined if and only if IARB=1 and this cross section is used for a layered composite model.	
41-45	Flag for layered composite material mode, ICOMP EQ.1: a material angle in degrees) is defined for each through thickness integration point starting with Section Card 5.	I5

Shell Element Section Card 2 of 2 (5E10.0)
Define for Shell and Membrane Elements

Insert a blank card here if the material definition is for a thick (8-node) shell.

Columns	Quantity	Format
1-10	Shell thickness at node n_1 (See Figure 3.4)	E10.0
11-20	Shell thickness at node n_2	E10.0
21-30	Shell thickness at node n_3	E10.0
31-40	Shell thickness at node n_4	E10.0
41-50	Location of reference surface (Hughes-Liu shell theory only) EQ.1.0: top surface EQ.0.0: midsurface (default for Hughes-Liu and mandatory for Belytschko-Tsay) EQ.-1.0: bottom surface	E10.0

The thickness values can be overridden on the element cards, i.e., the above values are used if and only if the thickness values are zero on the element cards.

The location of the reference surface can be crucial whenever beams, shells, and solids are interconnected. Often it is desirable to have the beams and shells share a common reference surface.

Material Angle Cards (IARB=1) (ICOMP=1)

Define the following cards if required by the constitutive model. Include as many cards as necessary. Angles are in degrees.

Columns	Quantity	Format
1-10	β_1 material angle at first integration point	E10.0
11-20	β_2 material angle at second integration point	E10.0
21-30	β_3 material angle at third integration point	E10.0
.	.	.
.	.	.
.	.	.
71-80	β_8 material angle at eighth integration point	E10.0

Define as many cards as necessary until NIP points are defined.

Part Set Definitions (IARB=1)

Define the number of cards sets specified in columns 1-5 of the first Control Card. (Typically, 2 cards are input for each set; but VDA requires 1 additional card and geometric contact entities requires 3 addtional cards.)

Card 1 (8I5,2E10.0) or (2I10,6I5,2E10.0) for MLARG

Columns	Quantity	Format
1-5	Part ID	I5
6-10	Constitutive model ID	I5
11-15	Equation-of-state ID EQ.0: shells, beams, and most brick materials types. NE.0: for brick elements if material type requires an equation of state.	I5
16-20	Cross-section properties ID EQ.0: for brick types 1-4 and 8-10.	I5
21-25	Element formulation if other than default. **For brick elements:** EQ.1: constant stress (default) EQ.2: 8 point integration EQ.3: 14 point integration quadratic 8-node brick EQ.4: 5 point integration quadratic 4-node tetrahedron EQ.5: 1 point ALE EQ.6: 1 point Eulerian EQ.7: 1 point Eulerian ambient EQ.8: acoustic pressure formulation EQ.9: 1 point crushable foam element EQ.10: 1 point tetrahedron EQ.11: 1 point ALE multi-material element EQ.12: 1 point integration with single material and void. **For beam elements:** EQ.1: Hughes-Liu (default) EQ.2: Belytschko-Schwer EQ.3: Truss EQ.4: Belytschko-Schwer full integration EQ.5: Belytschko-Schwer tubular beam (user defined integration rule advised.) EQ.6: Discrete 3D beam (Use one point integration) EQ.7: 2D plane strain shell (Use one point integration and x-y plane)	I5

Columns	Quantity	Format
	EQ.8: 2D axisymmetric shell (Use one point integration and y-axis of symmetry)	
	For 3D shell elements and 2D solid elements: EQ.1: Hughes-Liu EQ.2: Belytschko-Tsay EQ.3: BCIZ triangular shell EQ.4: C_o 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 Belyschko-Tsay membrane EQ.10: Belytschko-Wong-Chiang EQ.11: Corotational Hughes-Liu EQ.12: Plane stress 2D element (x-y plane) EQ.13: Plane strain 2D element (x-y plane) EQ.14: Axisymmetric Petrov-Galerkin 2D solid (y-axis of symmetry) EQ.15: Axisymmetric Galerkin 2D solid (y-axis of symmetry) EQ.16: fully integrated 4 noded shell (very fast)	
	For thick shell elements: EQ.1: single point in plane quadrature EQ.2: selective reduced 2×2 in plane quadrature	
26-30	Ambient element type. EQ.1: temperature EQ.2: pressure and temperature EQ.3: pressure outflow EQ.4: pressure inflow	I5
31-70	Blank	
71-80	FCPARM, flag to redefine contact parmeters EQ.0: no, those defined with the constitutive data are used. EQ.2: yes	I5

Card 2 (2A8)

Columns	Quantity	Format
1-8	PSHELL (or PBAR, or PSOLID)	A8
9-32	Blank	
33-40	Property name	A8

Notes:

1. This card is set up in such a way that NASBDF for SMUG post processing will be written correctly.

2. If the NASBDF file is not required, an arbitrary heading in format 9A8 can be written.

Optional Card 3 (7E10.0)
Define if and only if FCPARM=1

The following parameters may be defined by part ID for automatic single surface contact.

Columns	Quantity	Format
1-10	Static coefficient of friction EQ:0.0: set to default	E10.0
11-20	Dynamic coefficient of friction EQ:0.0: set to default	E10.0
21-30	Exponential decay coefficient EQ:0.0: set to default	E10.0
31-40	Viscous friction coefficient EQ:0.0: set to default	E10.0
41-50	Optional contact thickness EQ: 0.0: set to default	E10.0
51-60	Optional thickness scale factor EQ:0.0: set to default	E10.0
61-70	Local penalty scale factor EQ:0.0: set to default	E10.0

VDA Surface for Rigid Part (IARB=1)
(A80)

Define the following optional card if required by the Type 20 material input flag. See Section 3.20m.

Optional Card (A80)

Columns	Quantity	Format
1-80	VDA surface alias name (less than 12 characters)	A80

Contact Entity Mesh for Rigid Part (IARB=1)
(A80)

Define the following three cards if required by the Type 20 material input flag. See Section 3.20m. *If more than one entity is defined with the part, then define 3 cards for each entity to be generated, (Card 3, Cols. 71-80 for the type 20 material definition).*

Optional Entity Generation, Card 1 of 3
(6E10.0)

Columns	Quantity	Format
1-10	x-center, x_c	E10.0
11-20	y-center, y_c	E10.0
21-30	z-center, z_c	E10.0
31-40	x-direction for local axis X′, A_x	E10.0
41-50	y-direction for local axis X′, A_y	E10.0
51-60	z-direction for local axis X′, A_z	E10.0

Optional Entity Generation, Card 2 of 3
(3E10.0)

Columns	Quantity	Format
1-10	x-direction for local axis Y′, B_x	E10.0
11-20	y-direction for local axis Y′, B_y	E10.0
21-30	z-direction for local axis Y′, B_z	E10.0

(x_c, y_c, z_c) positions the local origin of the geometric entity in global coordinates. The entity's local X′-axis is determined by the vector (A_x, A_y, A_z) and the local Y′-axis by the vector (B_x, B_y, B_z).

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), X′=(A_x, A_y, A_z)=(1,0,0), and Y′=(B_x, B_y, B_z)=(0,1,0).

Optional Entity Generation, Card 3 of 3
(2I5,7E10.0)

Columns	Quantity	Format
1-5	Entity types available for mesh generation EQ.1: infinite plane EQ.2: sphere EQ.3: infinite cylinder EQ.4: hyperellipsoid EQ.5: torus EQ.10: finite plane EQ.11: load curve defining line	I5
6-10	In-out flag (this flag may be ignored here) EQ.0: slave nodes exist outside of the entity EQ.1: slave nodes exist inside the entity	I5
11-20	Entity coefficient g_1	E10.0
21-30	Entity coefficient g_2	E10.0
31-40	Entity coefficient g_3	E10.0
41-50	Entity coefficient g_4	E10.0
51-60	Entity coefficient g_5	E10.0
61-70	Entity coefficient g_6	E10.0
71-80	Entity coefficient g_7	E10.0

IGTYPE = 1: g1 = Px g4 = Qx
g2 = Py g5 = Qy
g3 = Pz g6 = Qz
g7 = L

A square plane of length L on each edge is generated which represents the infinite plane.

IGTYPE = 2: g1 = Px g4 = r
g2 = Py
g3 = Pz

IGTYPE = 3: g1 = Px g4 = Qx
g2 = Py g5 = Qy
g3 = Pz g6 = Qz
g7 = r

A cylinder of length $\sqrt{Qx^2 + Qy^2 + Qz^2}$ and radius r is generated which represents the infinite cylinder.

IGTYPE = 4: g1 = Px g4 = a
g2 = Py g5 = b
g3 = Pz g6 = c
g7 = n (order of the ellipsoid, default=2)

IGTYPE = 5: g1 = Radius of torus
g2 = r
g3 = Number of elements about the minor circumference (default=10)
g4 = Number of elements about the major circumference (default=20).

IGTYPE = 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)

IGTYPE = 9: g1 = Shell thickness (option to override true thickness)
g2 = Scale factor for true thickness (optional)
g3 = Load curve ID defining thickness versus time. (optional)

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

IGTYPE =11: g1 =Load curve ID defining axisymmetric surface profile about Z′-axis
g2 = Number of elements about the circumference (default=10)
g3 = Number of elements along the axis (default=20).

Constitutive Models
Cards 3,4,5,....,8 (6E10.0)

Material Type 1 (Elastic/Fluid)

Columns		Quantity	Format
1-10	Card 3	Young's modulus (Bulk modulus if Flag=1. in columns 11-20 and set Poisson's ratio to zero. This applies to solid elements only.)	E10.0
		The following may be defined for the Belytschko beam:	
11-20	Card 3	Axial damping factor for Belytschko's beam	E10.0
21-30	Card 3	Bending damping factor for Belytschko's beam	E10.0
		The following may be defined for the solid elements for fluids:	
11-20	Card 3	Flag=1.0 to eliminate deviatoric stress for fluid like behavior	E10.0
21-30	Card 3	Optional tensor viscosity coefficient if Flag=1.0.	E10.0
31-40	Card 3	Cavitation pressure (default=1.0e+20) if Flag=1.0.	E10.0
1-10	Card 4	Poisson's ratio	
	Card 5	Blank	
	Card 6	Blank	
	Card 7	Blank	
	Card 8	Blank	

When the flag is set to 1.0 in columns 11-20 on Card 3 fluid like behavior can be obtained where the bulk modulus, K, and pressure rate, $\dot{p}$, are given by:

$$K = \frac{E}{3(1-2\nu)}$$

$$\dot{p} = K\varepsilon_{ii}$$

and the shear modulus is set to zero. Define the bulk modulus since Poisson's ratio is ignored for Flag=1.0

Material Type 2 (Orthotropic Elastic)

Columns		Quantity	Format
1-10	Card 3	E_a (see Figure 3.5)	E10.0
11-20		E_b	E10.0
21-30		E_c	E10.0
1-10	Card 4	ν_{ba}	E10.0
11-20		ν_{ca}	E10.0
21-30		ν_{cb}	E10.0
1-10	Card 5	G_{ab}	E10.0
11-20		G_{bc}	E10.0
21-30		G_{ca}	E10.0

OPTIONAL definition of Cards 3-5 for brick elements only:

1-10	Card 3	C11 (1 corresponds to the *a* material direction)	E10.0
11-20		C12	E10.0
21-30		C22	E10.0
31-40		C13	E10.0
41-50		C23	E10.0
51-60		C33	E10.0
61-70		C14	E10.0
71-80		C24	E10.0
1-10	Card 4	C34	E10.0
11-20		C44	E10.0
21-30		C15	E10.0
31-40		C25	E10.0
41-50		C35	E10.0

Columns		Quantity	Format
51-60		C45	E10.0
61-70		C55	E10.0
71-80		C16	E10.0
1-10	Card 5	C26	E10.0
11-20		C36	E10.0
21-30		C46	E10.0
31-40		C56	E10.0
41-50		C66	E10.0
1-10	Card 6	Material axes option, AOPT EQ.0.0: locally orthotropic with material axes determined by element nodes as shown in Figure 3.5. Cards 7 and 8 are blank with this option. EQ.1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center. Card 8 below is blank. EQ.2.0: globally orthotropic with material axes determined by vectors defined on Cards 7 and 8 EQ.3.0: This option determines locally orthotropic material axes by offsetting the material axes by an angle (Card 8) from a line in the plane of the shell determined by taking the cross product of the vector defined on Card 7 with the shell normal vector. In solid elements the normal vector is normal to the plane of the midsurface between the inner surface and outer surface defined by the first four nodes and the last four nodes of the connectivity of the element, respectively. EQ.4.0: locally orthotropic in cylindrical coordinate system with material axes determined by the vector defined on Card 7 and the originating point, P, on Card 8.	E10.0
11-20		Use reference geometry to initial stresses. This option. applies to solid elements only. EQ.0.0: off EQ.1.0: on	E10.0

Columns		Quantity	Format
1-10	Card 7	x_p, define for AOPT = 1.0	E10.0
11-20		y_p, define for AOPT = 1.0	E10.0
21-30		z_p, define for AOPT = 1.0	E10.0
1-10	Card 7	a_1, define for AOPT = 2.0	E10.0
11-20		a_2, define for AOPT = 2.0	E10.0
21-30		a_3, define for AOPT = 2.0	E10.0
1-10	Card 7	v_1, define for AOPT = 3.0 & 4.0	E10.0
11-20		v_2, define for AOPT = 3.0 & 4.0	E10.0
21-30		v_3, define for AOPT = 3.0 & 4.0	E10.0
1-10	Card 8	d_1, define for AOPT = 2.0	E10.0
11-20		d_2, define for AOPT = 2.0	E10.0
21-30		d_3, define for AOPT = 2.0	E10.0
1-10	Card 8	Material angle beta, may be overridden on the element card (degrees)	E10.0
1-10	Card 8	P_1, define for AOPT = 4.0	E10.0
11-20		P_2, define for AOPT = 4.0	E10.0
21-30		P_3, define for AOPT = 4.0	E10.0

Material Type 2 (Orthotropic Elastic)

The material law that relates stresses to strains is defined as:

$$\underset{\sim}{C} = \underset{\sim}{T}^T \, \underset{\sim}{C}_L \underset{\sim}{T},$$

where $\underset{\sim}{T}$ is a transformation matrix, and $\underset{\sim}{C}_L$ is the constitutive matrix defined in terms of the material constants of the orthogonal material axes, a, b, and c. The inverse of $\underset{\sim}{C}_L$ is defined as

$$\underset{\sim}{C}_L^{-1} = \begin{bmatrix} \frac{1}{E_a} & -\frac{\nu_{ba}}{E_b} & -\frac{\nu_{ca}}{E_c} & 0 & 0 & 0 \\ -\frac{\nu_{ab}}{E_a} & \frac{1}{E_b} & -\frac{\nu_{cb}}{E_c} & 0 & 0 & 0 \\ -\frac{\nu_{ac}}{E_a} & -\frac{\nu_{bc}}{E_b} & \frac{1}{E_c} & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1}{G_{ab}} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1}{G_{bc}} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1}{G_{ca}} \end{bmatrix}$$

Note that $\frac{\nu_{ab}}{E_a} = \frac{\nu_{ba}}{E_b}, \frac{\nu_{ca}}{E_c} = \frac{\nu_{ac}}{E_a}, \frac{\nu_{cb}}{E_c} = \frac{\nu_{bc}}{E_b}.$

For brick elements the upper triangular part of the symmetric matrix $\underset{\sim}{C}_L$ can be defined columnwise to obtain fully anisotropic behavior.

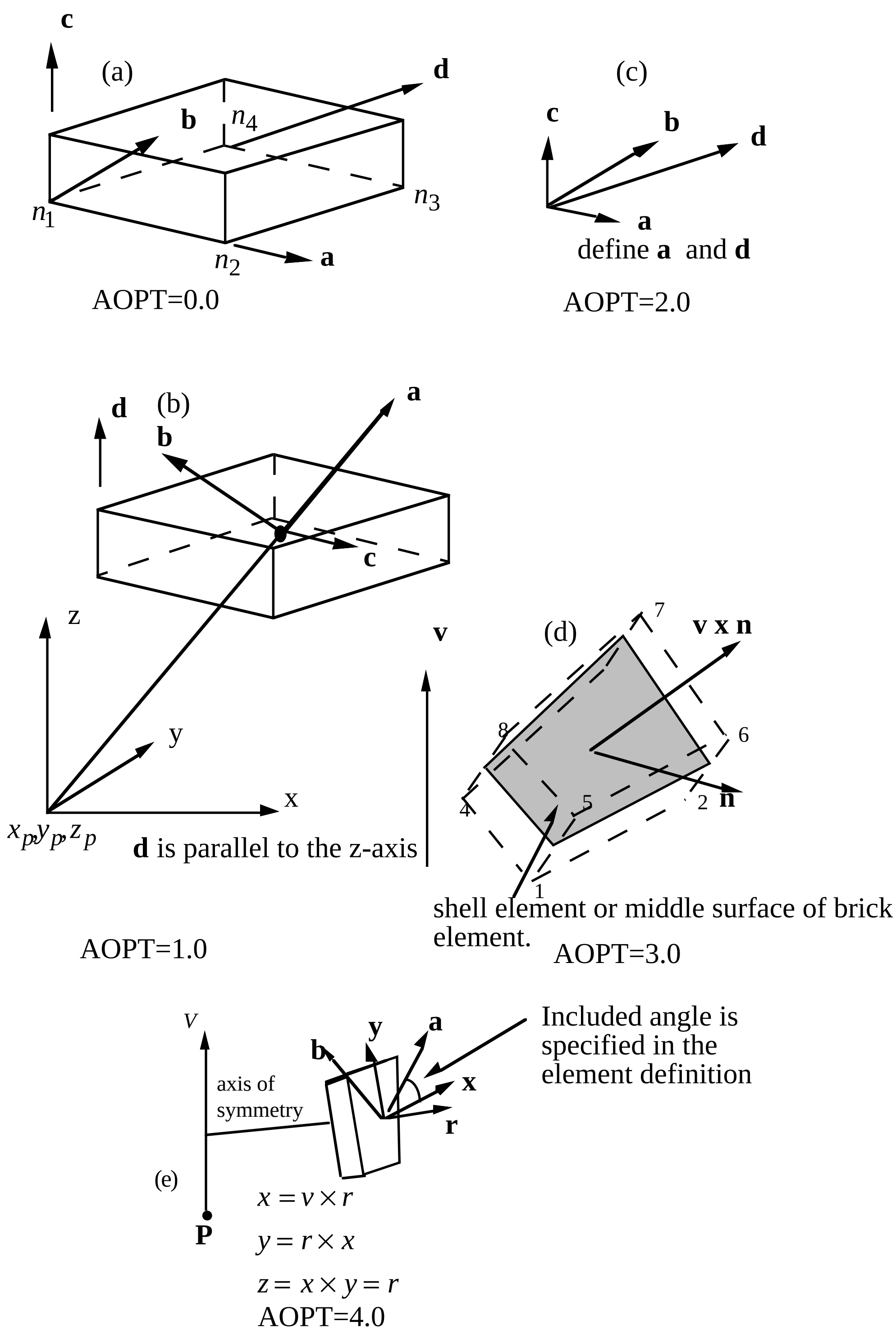


Figure 3.5. Options for determining principal material axes: (a) AOPT = 0.0, (b) AOPT = 1.0, (c) AOPT = 2.0. Note that $\underset{\sim}{c} = \underset{\sim}{a} \times \underset{\sim}{d}$ and that $\underset{\sim}{b} = \underset{\sim}{c} \times \underset{\sim}{a}$, (d) AOPT = 3.0, and (e) AOPT=4.0 for brick elements.

Material Type 3 (Kinematic/Isotropic Elastic-Plastic)

Columns	Quantity		Format
1-10	Card 3	Young's modulus	E10.0
11-20		Strain rate parameter, C	E10.0
21-30		Strain rate parameter, p	E10.0
31-40		Formulation for rate effects EQ.0.0: Scale yield stress (default) EQ.1.0: Viscoplastic formulation	E10.0

point in space and the global location 1-10

Card 4	Poisson's ratio	E10.0	
1-10	Card 5	Yield stress	E10.0
1-10	Card 6	Hardening modulus, E_t	E10.0
1-10	Card 7	Hardening parameter, β' $0 < \beta' < 1$	E10.0
1-10	Card 8	Failure strain for eroding elements.	

Strain rate is accounted for using the Cowper and Symonds model which scales the yield stress with the factor

$$1 + \left(\frac{\dot{\varepsilon}}{C} \right)^{1/p}$$

where $\dot{\varepsilon}$ is the strain rate. A fully viscoplastic formulation is optional which incorporates the Cowper and Symonds formulation within the yield surface. An additional cost is incurred but the improvement is results can be dramatic. To ignore strain rate effects set both C and p to zero.

Kinematic, isotropic, or a combination of kinematic and isotropic hardening may be specified by varying β' between 0 and 1. For β' equal to 0 and 1, respectively, kinematic and isotropic hardening are obtained as shown in Figure 3.6. For isotropic hardening, β' = 1, Material Model 12 requires less storage and is more efficient. Whenever possible,

Material Type 3 (Kinematic/Isotropic Elastic-Plastic)

Material 12 is recommended for solid elements but is less accurate for shell elements and its use may not be advisable.

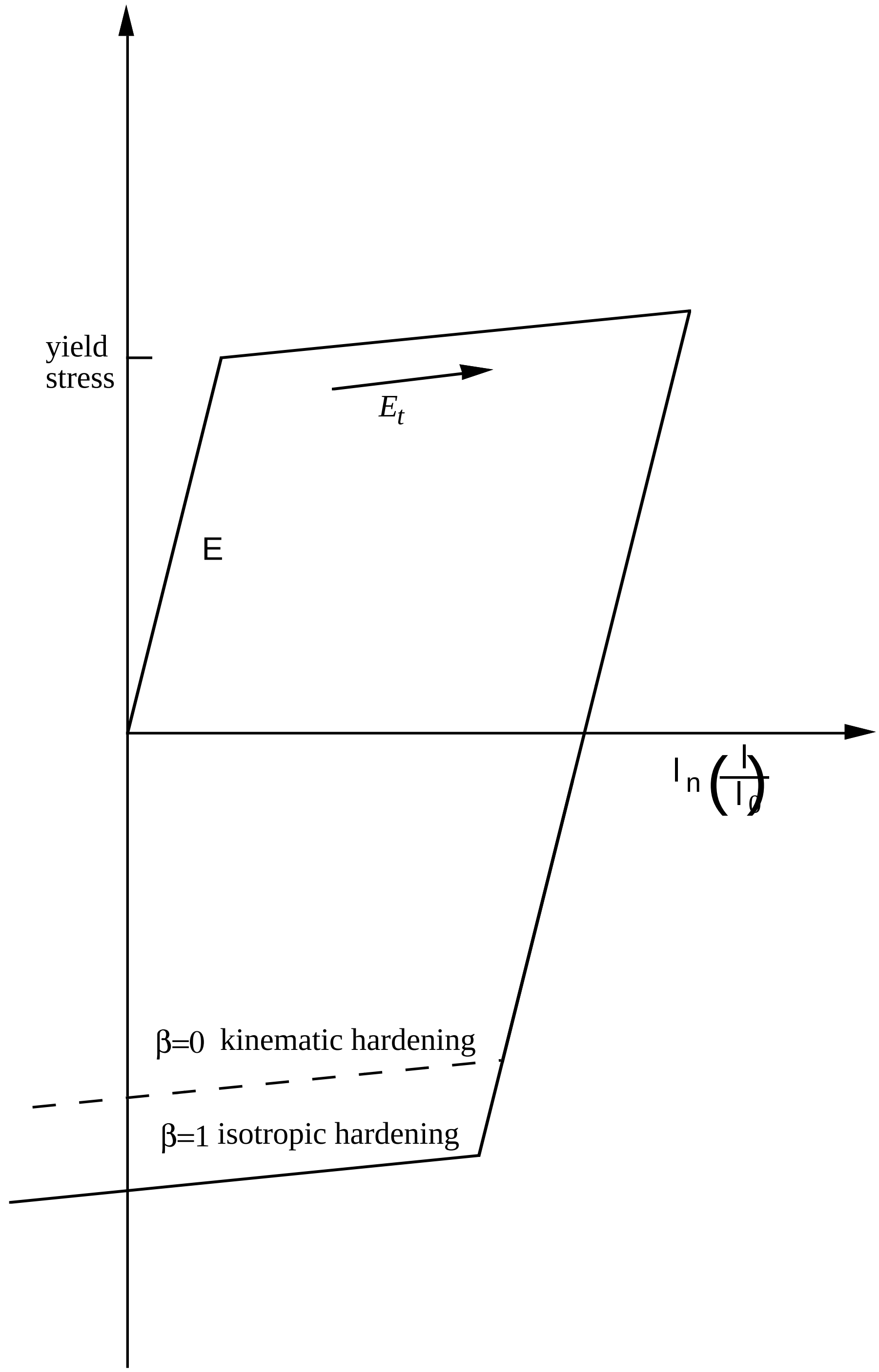


Figure 3.6. Elastic-plastic behavior with kinematic and isotropic hardening where l_0 and l are undeformed and deformed lengths of uniaxial tension specimen.

Material Type 4 (Thermo-Elastic-Plastic)

Columns		Quantity	Format
1-10	Card 3	T_1, temperature	E10.0
11-20		T_2	E10.0
.		.	.
.		.	.
.		.	.
71-80		T_8	E10.0
1-10	Card 4	E_1, Young's modulus at T_1	E10.0
11-20		E_2	E10.0
.		.	.
.		.	.
.		.	.
71-80		E_8	E10.0
1-10	Card 5	ν_1, Poisson's ratio at T_1	E10.0
11-20		ν_2	E10.0
.		.	.
.		.	.
.		.	.
71-80		ν_8	E10.0
1-10	Card 6	α_1, coefficient of thermal expansion at T_1	E10.0
11-20		α_2,	E10.0
.		.	.
.		.	.
.		.	.
71-80		α_8	E10.0

Material Type 4 (Thermo-Elastic-Plastic)

Columns		Quantity	Format
1-10	Card 7	σ_{y1}, yield stress at T_1	E10.0
11-20		σ_{y2},	E10.0
.		.	.
.		.	.
.		.	.
71-80		σ_{y8},	E10.0
1-10	Card 9	E_1^p plastic hardening modulus at T_1	E10.0
11-20		E_2^p	E10.0
.		.	.
.		.	.
.		.	.
71-80		E_8^p	E10.0

At least two temperatures and their corresponding material properties must be defined. The analysis will be terminated if a material temperature falls outside the range defined in the input. If a thermoelastic material is considered, leave Cards 7 and 8 blank. The coefficient of thermal expansion is defined with respect to the reference temperature at the beginning of the calculation for that material.

Material Type 5 (Soil and Crushable/Non-crushable Foam)

Columns		Quantity	Format
1-10	Card 3	Shear modulus	E10.0
11-20		Bulk unloading modulus	E10.0
21-30		Yield function constant a_0	E10.0
31-40		Yield function constant a_1	E10.0
41-50		Yield function constant a_2	E10.0
51-60		Pressure cutoff for tensile fracture	E10.0
61-70		Volumetric crushing option EQ.0.0: on EQ.1.0: loading and unloading paths are the same	E10.0
71-80		Use reference geometry to initialize pressure. EQ.0.0: off EQ.1.0: on	E10.0
1-10	Card 4	Volumetric strain (see Figure 3.7)	E10.0
11-20		Pressure	E10.0
21-30		Volumetric strain	E10.0
31-40		Pressure	E10.0
1-10	Card 5	Volumetric strain	E10.0
11-20		Pressure	E10.0
21-30		Volumetric strain	E10.0
31-40		Pressure	E10.0
.		.	.
.		.	.
.		.	.
1-10	Card 8	Volumetric strain	E10.0
11-20		Pressure	E10.0
21-30		Volumetric strain	E10.0
31-40		Pressure	E10.0

Pressure is positive in compression. Volumetric strain is given by the natural log of the relative volume and is negative in compression. The tabulated data should be given in order of increasing compression. If the pressure drops below the cutoff value specified, it is reset to that value.

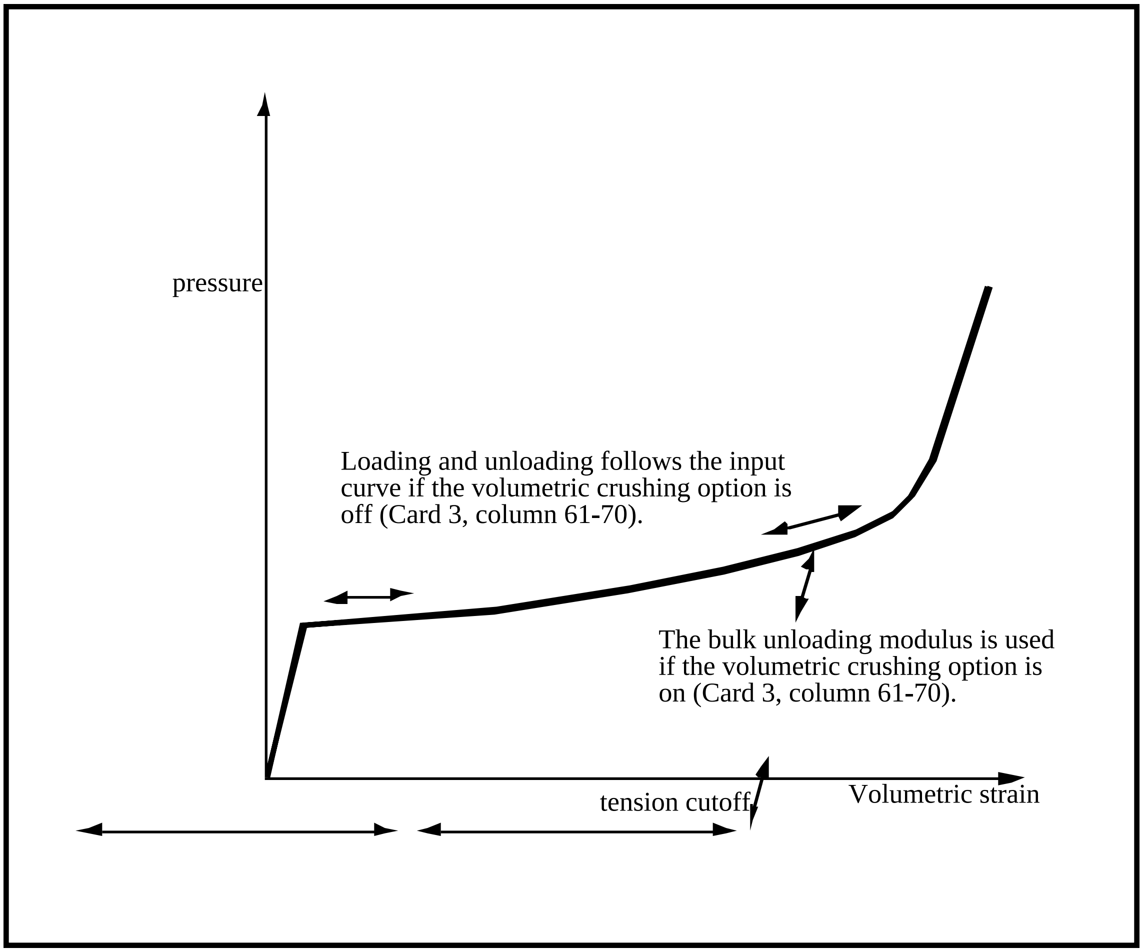


Figure 3.7. Pressure versus volumetric strain curve for soil and crushable foam model. The volumetric strain is given by the natural logarithm of the relative volume, V.

The deviatoric perfectly plastic yield function, ϕ, is described in terms of the second invariant J_2,

$$J_2 = \frac{1}{2} s_{ij} \; s_{ij}$$

pressure, p, and constants a_0, a_1, and a_2 as:

$$\phi = J_2 - \left[a_0 + a_1 p + a_2 p^2\right].$$

On the yield surface $J_2 = \frac{1}{3}\sigma_y^2$ where σ_y is the uniaxial yield stress, i.e.,

$$\sigma_y = \left[3\left(a_0 + a_1 p + a_2 p^2\right)\right]^{1/2}$$

On this surface, there is no strain hardening. For no pressure hardening, $a_1 = a_2 = 0$, and $(3a_0)^{1/2}$ defines the yield strength.

Material Type 6 (Viscoelastic)

Columns	Quantity		Format
1-10	Card 3	Bulk modulus (elastic)	E10.0
1-10	Card 4	Short-time shear modulus, G_0	E10.0
1-10	Card 5	Long-time shear modulus, G_∞	E10.0
1-10	Card 6	Decay constant, β	E10.0
	Card 7	Blank	
	Card 8	Blank	

The shear relaxation behavior is described by:

$$G(t) = G_\infty + (G_0 - G_\infty)\, e^{-\beta t}$$

A Jaumann rate formulation is used

$$\overset{\nabla}{\sigma}{}'_{ij} = 2\int_0^t G(t-\tau) D'_{ij}(\tau) dt$$

where the prime denotes the deviatoric part of the stress rate, $\overset{\nabla}{\sigma}_{ij}$, and the strain rate D_{ij} .

Material Type 7 (Blatz - Ko Rubber)

Columns		Quantity	Format
1-10	Card 3	Shear modulus, μ.	E10.0
11-20		Optional Poisson's ratio (see comments below).	E10.0
21-30		Use reference geometry to initial stresses. This option. applies to solid elements only. EQ.0.0: off EQ.1.0: on	E10.0
	Card 4	Blank	
.		.	.
.		.	.
.		.	.
	Card 8	Blank	

The second Piola-Kirchhoff stress is computed as

$$S_{ij} = \mu \left(\frac{1}{V} C_{ij} - V^{-1/(1-2\nu)} \delta_{ij} \right)$$

where V is the relative volume, C_{ij} is the right Cauchy-Green strain tensor, and ν is Poisson's ratio, which is set to .463 internally. This stress measure is transformed to the Cauchy stress, σ_{ij}, according to the relationship

$$\sigma_{ij} = V^{-1} F_{ik} F_{jl} S_{lk}$$

where F_{ij} is the deformation gradient tensor.

Material Type 8 (High Explosive Burn)

This material type models the detonation of a high explosive. In addition an equation of state must be defined (either type 2 or 3).

Columns		Quantity	Format
1-10	Card 3	D, detonation velocity	E10.0
11-20		P_{CJ}, Chapman-Jouget pressure	E10.0
21-30		Beta burn flag, BETA EQ.0.0: beta +progammed burn EQ.1.0: beta burn only EQ.2.0: programmed burn only	E10.0
31-40		K, bulk modulus (BETA=2.0 only)	E10.0
41-50		G, shear modulus (BETA=2.0 only)	E10.0
51-60		σ_y, yield stress (BETA=2.0 only)	E10.0
	Card 4	Blank	
.		.	.
.		.	.
.		.	.
	Card 8	Blank	

Burn fractions, F, which multiply the equations of states for high explosives, control the release of chemical energy for simulating detonations. At any time, the pressure in a high explosive element is given by:

$$p = Fp_{eos}(V, E)$$

where p_{eos}, is the pressure from the equation of state (either types 2 or 3), V is the relative volume, and E is the internal energy density per unit initial volume.

In the initialization phase, a lighting time t_l is computed for each element by dividing the distance from the detonation point to the center of the element by the detonation velocity D. If multiple detonation points are defined, the closest detonation point determines t_l The burn fraction F is taken as the maximum

$$F = \max(F_1, F_2)$$

where

$$F_1 = \begin{cases} \dfrac{2\left(t - t_l\right) D A_{e_{\max}}}{3 v_e} & \text{if } t > t_l \\ 0 & \text{if } t \le t_l \end{cases}$$

$$F_2 = \beta = \frac{1 - V}{1 - V_{CJ}}$$

where V_{CJ} is the Chapman-Jouguet relative volume and t is current time. If F exceeds 1, it is reset to 1. This calculation of the burn fraction usually requires several time steps for F to reach unity, thereby spreading the burn front over several elements. After reaching unity, F is held constant. This burn fraction calculation is based on work by Wilkins [1964] and is also discussed by Giroux [1973].

If the beta burn option is used, BETA=1.0, any volumetric compression will cause detonation and

$$F = F_2$$

and F_1 is not computed.

If programmed burn is used, BETA=2.0, the explosive model will behave as an elastic perfectly plastic material if the bulk modulus, shear modulus, and yield stress are defined. Therefore, with this option the explosive material can compress without causing detonation.

Material Type 9 (Null Hydrodynamics)

Columns		Quantity	Format
1-10	Card 3	Pressure cutoff (≤ 0.0)	E10.0
11-20		Viscosity coefficient, μ	E10.0
21-30		Relative volume for erosion in tension Typically, use values greater than unity. If zero, erosion in tension is inactive.	E10.0
31-40		Relative volume for erosion in compression. Typically, use values less than unity. If zero, erosion in compression is inactive.	E10.0
1-10	Card 4	Young's modulus (used for null beams and shells only)	E10.0
11-20		Poisson's ratio (used for null beams and shells only)	E10.0
	Card 5	Blank	
.		.	.
.		.	.
.		.	.
	Card 8	Blank	

The null material must be used with an equation-of-state. Pressure cutoff is negative in tension. A viscous stress of the form

$$\sigma_{ij} = \mu \dot{\varepsilon}'_{ij}$$

is computed for nonzero μ where $\dot{\varepsilon}'_{ij}$ is the deviatoric strain rate.

Sometimes it is advantageous to model contact surfaces via shell elements which are not part of the structure, but are necessary to define areas of contact within nodal rigid bodies or between nodal rigid bodies.

Beams and shells that use this material type are completely bypassed in the element processing; however, the mass of the null shell elements is computed and added to the nodal points which define the connectivity, but the mass of null beams is ignored. The Young's modulus and Poisson's ratio are used only for setting the contact interface stiffnesses, and it is recommended that reasonable values be input.

Material Type 10 (Isotropic-Elastic-Plastic-Hydrodynamic)

Columns		Quantity	Format
1-10	Card 3	Shear modulus	E10.0
11-20		Yield strength, σ_0	E10.0
21-30		Plastic hardening modulus, E_h	E10.0
31-40		Pressure cutoff (< 0.0) EQ.0.0: a cutoff of $-\infty$ is assumed	
41-50		Linear pressure hardening coefficient, a_1	E10.0
51-60		Quadratic pressure hardening coefficient, a_2	E10.0
61-70		Spall type, ISPALL EQ.0.0: default set to "1.0" EQ.1.0: pressure limit model, $p \geqq p_{cut}$ EQ.2.0: if $\sigma_{max} \geq -p_{cut}$ element spalls in tension, p < 0, is never allowed EQ.3.0: $p < -p_{cut}$ element spalls in tension, p < 0, is never allowed	E10.0
1-10	Card 4	Failure strain for erosion	E10.0
.		.	.
.		.	.
.		.	.
1-10	Card 5	ε_1, effective plastic strain	E10.0
.		ε_2	.
.		ε_3	.
.		ε_4	.
41-50		ε_5	.
.		.	.
71-80		ε_8	E10.0
1-10	Card 6	ε_9	E10.0
.		.	.
.		.	.
71-80		ε_{16}	E10.0

Material Type 10 (Isotropic-Elastic-Plastic-Hydrodynamic)

Columns		Quantity	Format
1-10	Card 7	σ_1, effective stress	E10.0
.		.	.
.		.	.
.		.	.
71-80		σ_8	E10.0
1-10	Card 8	σ_9	E10.0
.		.	.
.		.	.
.		.	.
71-80		σ_{16}	E10.0

Whenever Cards 5-8 are blank, the yield stress and plastic hardening modulus are taken from Card 3. In this case, the bilinear stress-strain curve shown in Figure 3.6. is obtained with $\beta = 1$. The yield strength is calculated as

$$\sigma_y = \sigma_0 + E_h \bar{\varepsilon}^p + (a_1 + pa_2)\max[p,0]$$

The quantity E_h is the plastic hardening modulus defined in terms of Young's modulus, E, and the tangent modulus, E_t , as follows

$$E_h = \frac{E_t\ E}{E - E_t}$$

and p is the pressure taken as positive in compression.

If Cards 5-8 are used, a curve like that shown in Figure 3.8 may be defined. Effective stress is defined in terms of the deviatoric stress tensor, s_{ij}, as:

$$\bar{\sigma} = \left(\frac{3}{2} s_{ij}\ s_{ij}\right)^{1/2} \tag{1}$$

and effective plastic strain by:

$$\bar{\varepsilon}^p = \int_0^t \left(\frac{2}{3} D_{ij}^p\, D_{ij}^p\right)^{1/2} dt, \tag{2}$$

where t denotes time and D_{ij}^p is the plastic component of the rate of deformation tensor. In this case the plastic hardening modulus on Card 3 is ignored and the yield stress is given as

$$\sigma_y = f\left(\bar{\varepsilon}^p\right),$$

where the value for $f\left(\bar{\varepsilon}^p\right)$ is found by interpolation from the data curve. In this latter case pressure hardening is unavailable.

A choice of three spall models is offered to represent material splitting, cracking, and failure under tensile loads. The pressure limit model, ISPALL=1, limits the hydrostatic tension to the specified value, p_{cut}. If pressures more tensile than this limit are calculated, the pressure is reset to p_{cut}. This option is not strictly a spall model, since the deviatoric stresses are unaffected by the pressure reaching the tensile cutoff, and the pressure cutoff value, p_{cut}, remains unchanged throughout the analysis. The maximum principal stress spall model, ISPALL=2, detects spall if the maximum principal stress σ_{max} exceeds the limiting value $-p_{cut}$. Note that the negative sign is required because p_{cut} is measured positive in compression, while σ_{max} is positive in tension. Once spall is detected with this model, the deviatoric stresses are reset to zero, and no hydrostatic tension ($p<0$) is permitted. If tensile pressures are calculated, they are reset to 0 in the spalled material. Thus, the spalled material behaves as a rubble or incohesive material. The hydrostatic tension spall model, ISPALL=3, detects spall if the pressure becomes more tensile than the specified limit, p_{cut}. Once spall is detected the deviatoric stresses are reset to zero, and nonzero values of pressure are required to be compressive (positive). If hydrostatic tension ($p<0$) is subsequently calculated, the pressure is reset to 0 for that element.

This model is applicable to a wide range of materials, including those with pressure-dependent yield behavior. The use of 16 points in the yield stress versus effective plastic strain curve allows complex post-yield hardening behavior to be accurately represented. In addition, the incorporation of an equation of state permits accurate modeling of a variety of different materials. The spall model options permit incorporation of material failure, fracture, and disintegration effects under tensile loads.

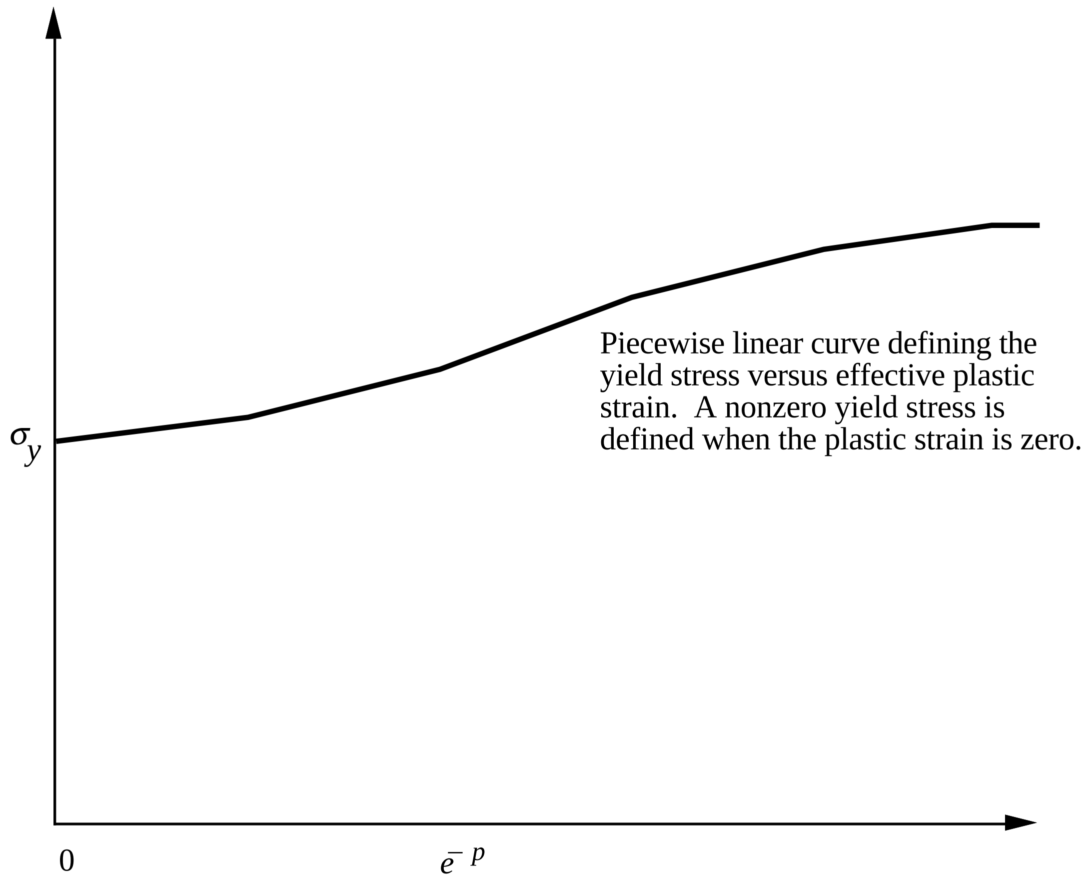


Figure 3.8. Effective stress versus effective plastic strain curve.

Material Type 11 (Temperature and Rate Dependent, Elastoplastic, Hydrodynamic)

Columns		Quantity	Format
1-10	Card 3	G_0	E10.0
11-20		σ_0	E10.0
21-30		β	E10.0
31-40		n	E10.0
41-50		γ_i	E10.0
1-10	Card 4	σ_m	E10.0
11-20		b	E10.0
21-30		b′	E10.0
31-40		h	E10.0
41-50		f	E10.0
51-60		U_k*, activation energy for rate dependent model	E10.0
61-70		C_1*, exponent prefactor in rate dependent model	E10.0
71-80		C_2*, coefficient of drag term in rate dependent model	E10.0
1-10	Card 5	A (if = 0.0, R′ must be defined)	E10.0
11-20		T_{mo}	E10.0
21-30		γ_o	E10.0
31-40		a	E10.0
41-50		p_{min} (default=-1.e+30)	
E10.0			
51-60		Y_P*, Peierls stress for rate dependent model	E10.0
61-70		Y_A*, Athermal yield stress for rate dependent model	E10.0
71-80		Y_{max}*, work hardening maximum for rate model	E10.0

*These optional constants activate the Steinberg-Lund [1989] rate dependent model.

Columns		Quantity	Format
1-10	Card 6	Spall type EQ.0.0: default set to "2.0" EQ.1.0: $p \geqq p_{cut}$ EQ.2.0: if $\sigma_{max} \geq -p_{cut}$ element spalls in tension, $p < 0$, is never allowed EQ.3.0: $p < -p_{cut}$ element spalls in tension, $p < 0$, is never allowed	E10.0
11-20		R' (if $R' \neq 0.0$, A is not defined)	E10.0
41-50		FLAG = 1.0 for μ coefficients being supplied for cold compression energy fit.	E10.0
51-60		Optional μ or η minimum value	E10.0
61-70		Optional μ or η maximum value	E10.0
1-16	Card 7	EC_0	E16.0
17-32		EC_1	E16.0
33-48		EC_2	E16.0
49-64		EC_3	E16.0
65-80		EC_4	E16.0
1-16	Card 8	EC_5	E16.0
17-32		EC_6	E16.0
33-48		EC_7	E16.0
49-64		EC_8	E16.0
65-80		EC_9	E16.0

Users who have an interest in this model are encouraged to study the paper by Steinberg and Guinan which provides the theoretical basis. Another useful reference is the KOVEC user's manual.

In terms of the foregoing input parameters, we define the shear modulus, G, before the material melts as:

$$G = G_0 \left[1 + bpV^{1/3} - h\left(\frac{E_i - E_c}{3R'} - 300 \right) \right] e^{-fE_i / (E_m - E_i)}$$

where p is the pressure, V is the relative volume, E_c is the cold compression energy:

$$\bar{\varepsilon}^{p} = \int_0^t \left(\frac{2}{3} D_{ij}^{p} D_{ij}^{p} \right)^{1/2} dt,$$

$$x = 1 - V,$$

and E_m is the melting energy:

$$E_m (x) = E_c (x) + 3R' T_m (x)$$

which is in terms of the melting temperature $T_m(x)$:

$$T_m (x) = \frac{T_{mo} \exp (2ax)}{V^{2\left(\gamma_o - a - \frac{1}{3}\right)}}$$

and the melting temperature at $\rho = \rho_o$, T_{mo}.

In the above equation R' is defined by

$$R' = \frac{R\rho}{A}$$

where R is the gas constant and A is the atomic weight. If R' is not defined, LS-DYNA computes it with R in the cm-gram-microsecond system of units.

The yield strength σ_y is given by:

$$\sigma_y = \sigma_0' \left[1 + b' p V^{1/3} - h \left(\frac{E_i - E_c}{3R'} - 300 \right) \right] e^{-fE_i / E_m - E_i}$$

if E_m exceeds E_i. Here, σ_0' is given by:

$$\sigma_y = \sigma_0' \left[1 + \beta \left(\gamma_i + \varepsilon^{-p} \right) \right]^n$$

where γ_i is the initial plastic strain. Whenever σ_0' exceeds σ_m, σ_0' is set equal to σ_m. After the materials melts, σ_y and G are set to one half their initial value.

If rate effects are included the yield stress is instead given by:

$$\sigma_y = \left\{ Y_T \left(\dot{\varepsilon}_p, T \right) + Y_A f \left(\varepsilon_p \right) \right\} \frac{G(p,T)}{G_0}$$

There are two imposed limits. The first is on the atermal yield stress:

$$Y_A f\left(\varepsilon_p\right) = Y_A \left[1 + \beta \left(\gamma_i + \varepsilon^p \right) \right]^n \leq Y_{max}$$

and the second is on the thermal part:

$$Y_T \le Y_P$$

If the coefficients EC0,...,EC9 are not defined above, LS-DYNA will fit the cold compression energy to a ten term polynomial expansion:

$$E_c = \sum_{i=0}^{9} EC_i \eta^i$$

where EC_i is the ith coefficient and $\eta = \frac{\rho}{\rho_o} - 1$. The least squares method is used to perform the fit.

A choice of three spall models is offered to represent material splitting, cracking, and failure under tensile loads. The pressure limit model, ISPALL=1, limits the hydrostatic tension to the specified value, p_{cut}. If pressures more tensile than this limit are calculated, the pressure is reset to p_{cut}. This option is not strictly a spall model, since the deviatoric stresses are unaffected by the pressure reaching the tensile cutoff, and the pressure cutoff value, p_{cut}, remains unchanged throughout the analysis. The maximum principal stress spall model, ISPALL=2, detects spall if the maximum principal stress σ_{max} exceeds the limiting value $-p_{cut}$. Note that the negative sign is required because p_{cut} is measured positive in compression, while σ_{max} is positive in tension. Once spall is detected with this model, the deviatoric stresses are reset to zero, and no hydrostatic tension ($p<0$) is permitted. If tensile pressures are calculated, they are reset to 0 in the spalled material. Thus, the spalled material behaves as a rubble or incohesive material. The hydrostatic tension spall model, ISPALL=3, detects spall if the pressure becomes more tensile than the specified limit, p_{cut}. Once spall is detected the deviatoric stresses are reset to zero, and nonzero values of pressure are required to be compressive (positive). If hydrostatic tension ($p<0$) is subsequently calculated, the pressure is reset to 0 for that element.

This model is applicable to a wide range of materials, including those with pressure-dependent yield behavior. The use of 16 points in the yield stress versus effective plastic strain curve allows complex post-yield hardening behavior to be accurately represented. In addition, the incorporation of an equation of state permits accurate modeling of a variety of different materials. The spall model options permit incorporation of material failure, fracture, and disintegration effects under tensile loads.

Material Type 12 (Isotropic-Elastic-Plastic)

Columns	Quantity		Format
1-10	Card 3	Shear modulus	E10.0
11-20		Yield stress (see Figure 3.6)	E10.0
21-30		Hardening modulus	E10.0
1-10	Card 4	K, bulk modulus	E10.0
	Card 5	Blank	
		.	
		.	
		.	
	Card 8	Blank	

Here the pressure is integrated in time

$$\dot{p} = -K \frac{\dot{V}}{V}$$

where V is the relative volume. This model is recommended for brick elements but not for shell elements since it is not too accurate.

Material Type 13 (Elastic-Plastic with Failure Model)

Columns		Quantity	Format
1-10	Card 3	Shear modulus	E10.0
11-20		Yield stress (see Figure 3.6)	E10.0
21-30		Hardening modulus	E10.0
31-40		Failure strain	E10.0
41-50		Failure pressure(≤ 0.0)	E10.0
1-10	Card 4	Bulk modulus	E10.0
1-10	Card 5	Element removal option EQ.0.0: failed element eroded after failure NE.0.0: element is kept, no removal except by Δt below	E10.0
1-10	Card 6	Delta t for element removal REM: element erosion option EQ.0.0: Δt is not considered. Default NE.0.0: element eroded if element time step size falls below Δt	E10.0
		.	
		.	
		.	
	Card 8	Blank	

When the effective plastic strain reaches the failure strain or when the pressure reaches the failure pressure, the element loses its ability to carry tension and the deviatoric stresses are set to zero, i.e., the material behaves like a fluid. If Delta t for element removal is defined the element removal option is ignored.

Material Type 14 (Soil and Crushable Foam with Failure Model)

The input for this model is the same as for Material Type 5; however when the pressure reaches the failure pressure the element loses its ability to carry tension.

Material Type 15 (Johnson/Cook Strain and Temperature Sensitive Plasticity)

Columns		Quantity	Format
1-10	Card 3	G	E10.0
11-20		A	E10.0
21-30		B	E10.0
31-40		n	E10.0
41-50		C	E10.0
51-60		m	E10.0
61-70		Melt temperature, T_m	E10.0
71-80		Room temperature, T_r	E10.0
1-10	Card 4	ε_0	E10.0
11-20		Specific heat	E10.0
21-30		p_{min} or failure stress, σ_p	E10.0
31-40		Spall Type EQ.0.0: default set to “2.0” EQ.1.0: $p \geq p_{min}$ EQ.2.0: if $\sigma_{max} \geq \sigma_p$ element spalls and tension, $p < 0$, is never allowed EQ.3.0: if $p < p_{min}$ element spalls and tension, $p < 0$, is never allowed	E10.0
41-50		Plastic strain iteration flag EQ.1.0: accurate iterative solution for plastic strain. Much more expensive than default.	E10.0
1-10	Card 5	D_1, failure parameter	E10.0
11-20		D_2	E10.0
21-30		D_3	E10.0
31-40		D_4	E10.0
41-50		D_5	E10.0

Material Type 15 (Johnson/Cook Plasticity)

Columns		Quantity	Format
.	Card 6	Blank	E10.0
	.	.	
	.	.	
1-10	Card 8	E, Young's Modulus (Shell elements only)	E10.0
11-20		ν, Poisson's ratio (Shell elements only)	E10.0
21-30		tf, time step size for automatic element deletion (Shell elements only)	E10.0
31-40		Formulation for rate effects EQ.0.0: Scale yield stress (default) EQ.1.0: Viscoplastic formulation	E10.0

Johnson and Cook express the flow stress as

$$\sigma_y = \left(A + B\,\bar{\varepsilon}^{p^n}\right)\left(1 + c\ \ln \dot{\varepsilon}^*\right)\left(1 - T^{*m}\right)$$

where

A, B, C, n, and m = input constants

$\bar{\varepsilon}^p$ = effective plastic strain

$\dot{\varepsilon}^* = \dfrac{\dot{\bar{\varepsilon}}^p}{\dot{\varepsilon}_0}$ effective plastic strain rate for $\dot{\varepsilon}_0 = 1\,s^{-1}$

T^* = homologous temperature = $\dfrac{T - T_{room}}{T_{melt} - T_{room}}$

Constants for a variety of materials are provided in [Johnson and Cook 1983]. A fully viscoplastic formulation is optional (Card 8, Columns 31-40) which incorporates the rate equations within the yield surface. An additional cost is incurred but the improvement is results can be dramatic.

Due to the nonlinearity in the dependence of flow stress on plastic strain, an accurate value of the flow stress requires iteration for the increment in plastic strain. However, by using a Taylor series expansion with linearization about the current time, we can solve for σ_y with sufficient accuracy to avoid iteration.

The strain at fracture is given by

$$\varepsilon^f = \left[D_1 + D_2 \exp D_3 \sigma^*\right]\left[1 + D_4 \ln \dot{\varepsilon}^*\right]\left[1 + D_5 T^*\right]$$

where σ^* is the ratio of pressure divided by effective stress

$$\sigma^* = \frac{p}{\sigma_{eff}}$$

Fracture occurs when the damage parameter

$$D = \sum \frac{\Delta \bar{\varepsilon}^p}{\varepsilon^f}$$

reaches the value of 1.

A choice of three spall models is offered to represent material splitting, cracking, and failure under tensile loads. The pressure limit model limits the minimum hydrostatic pressure to the specified value, $p \geq p_{min}$. If pressures more tensile than this limit are calculated, the pressure is reset to p_{min}. This option is not strictly a spall model since the deviatoric stresses are unaffected by the pressure reaching the tensile cutoff and the pressure cutoff value p_{min} remains unchanged throughout the analysis. The maximum principal stress spall model detects spall if the maximum principal stress, σ_{max}, exceeds the limiting value σ_p. Once spall is detected with this model, the deviatoric stresses are reset to zero and no hydrostatic tension is permitted. If tensile pressures are calculated, they are reset to 0 in the spalled material. Thus, the spalled material behaves as rubble. The hydrostatic tension spall model detects spall if the pressure becomes more tensile than the specified limit, p_{min}. Once spall is detected, the deviatoric stresses are set to zero and the pressure is required to be compressive. If hydrostatic tension is calculated then the pressure is reset to 0 for that element.

In addition to the above failure criterion, this material model also supports a shell element deletion criterion based on the maximum stable time step size for the element, Δt_{max}. Generally, Δt_{max} goes down as the element becomes more distorted. To assure stability of time integration, the global LS-DYNA time step is the minimum of the Δt_{max} values calculated for all elements in the model. Using this option allows the selective deletion of elements whose time step Δt_{max} has fallen below the specified minimum time step, Δt_{crit}. Elements which are severely distorted often indicate that material has failed and supports little load, but these same elements may have very small time steps and therefore control the cost of the analysis. This option allows these highly distorted elements to be deleted from the calculation, and, therefore, the analysis can proceed at a larger time step, and, thus, at a reduced cost. Deleted elements do not carry any load, and are deleted from all applicable slide surface definitions. Clearly, this option must be judiciously used to obtain accurate results at a minimum cost.

Material type 15 is applicable to the high rate deformation of many materials including most metals. Unlike the Steinberg-Guinan model, the Johnson-Cook model remains valid down to lower strain rates and even into the quasistatic regime. Typical applications include explosive metal forming, ballistic penetration, and impact.

Material Type 16 (Pseudo TENSOR Concrete/Geological Model)

Material Type 16 was developed to provide concrete and geological material modeling capabilities.

Columns		Quantity	Format
1-10	Card 3	υ (constant Poisson's ratio model) or -G (constant shear modulus model)	E10.0
11-20		Maximum principal stress failure, σ_{cut}	E10.0
21-30		Cohesion (a_0)	E10.0
31-40		Pressure hardening coefficient (a_1)	E10.0
41-50		Pressure hardening coefficient (a_2)	E10.0
51-60		Damage scaling factor b_1	E10.0
61-70		Cohesion for failed material (a_{0f})	E10.0
71-80		Pressure hardening coefficient for failed material (a_{lf})	E10.0
1-10	Card 4	Percent reinforcement, f_r, $\left(0 \le f_r \le 100\%\right)$	E10.0
11-20		Elastic modulus for reinforcement	E10.0
21-30		Poisson's ratio for reinforcement	E10.0
31-40		Initial yield stress	E10.0
41-50		Tangent modulus	E10.0
51-60		N1, curve ID giving rate sensitivity for principal material	E10.0
61-70		N2, curve ID giving rate sensitivity for reinforcement	E10.0
1-10	Card 5	ε_1 effective plastic strain or pressure, p_1, or damage, λ_1	E10.0
.		.	.
.		.	.
.		.	.
71-80		ε_8 or p_8 or λ_8	E10.0
1-10	Card 6	ε_9 or p or λ_9	E10.0
.		.	.
.		.	.

Columns		Quantity	Format
.		.	.
71-80		ε_{16} or p_{16} or λ_{16}	E10.0
1-10	Card 7	η_1, scale factor or yield stress, σ_1	E10.0
.		.	.
.		.	.
.		.	.
71-80		η_8 or σ_8	E10.0
1-10	Card 8	η_9 or σ_9	E10.0
.		.	.
.		.	.
.		.	.
71-80		η_{16} or σ_{16}	E10.0

This model can be used in two major modes - a simple tabular pressure-dependent yield surface, and a potentially complex model featuring two yield versus pressure functions with the means of migrating from one curve to the other. For both modes, load curve N1 is taken to be a strain rate multiplier for the yield strength. Note that this model must be used with equation-of-state type 8 or 9.

Response Mode I. Tabulated Yield Stress Versus Pressure

This model is well suited for implementing standard geologic models like the Mohr-Coulomb yield surface with a Tresca limit, as shown in Figure 3.9. Examples of converting conventional triaxial compression data to this type of model are found in (Desai and Siriwardane, 1984). Note that under conventional triaxial compression conditions, the LS-DYNA input corresponds to an ordinate of $\sigma_1 - \sigma_3$ rather than the more widely used $\frac{\sigma_1 - \sigma_3}{2}$, where σ_1 is the maximum principal stress and σ_3 is the minimum principal stress.

This material combined with equation-of-state type 9 (saturated)has been used very successfully to model ground shocks and soil-structure interactions at pressures up to 100kbars (approximately 1.5 x 106 psi).

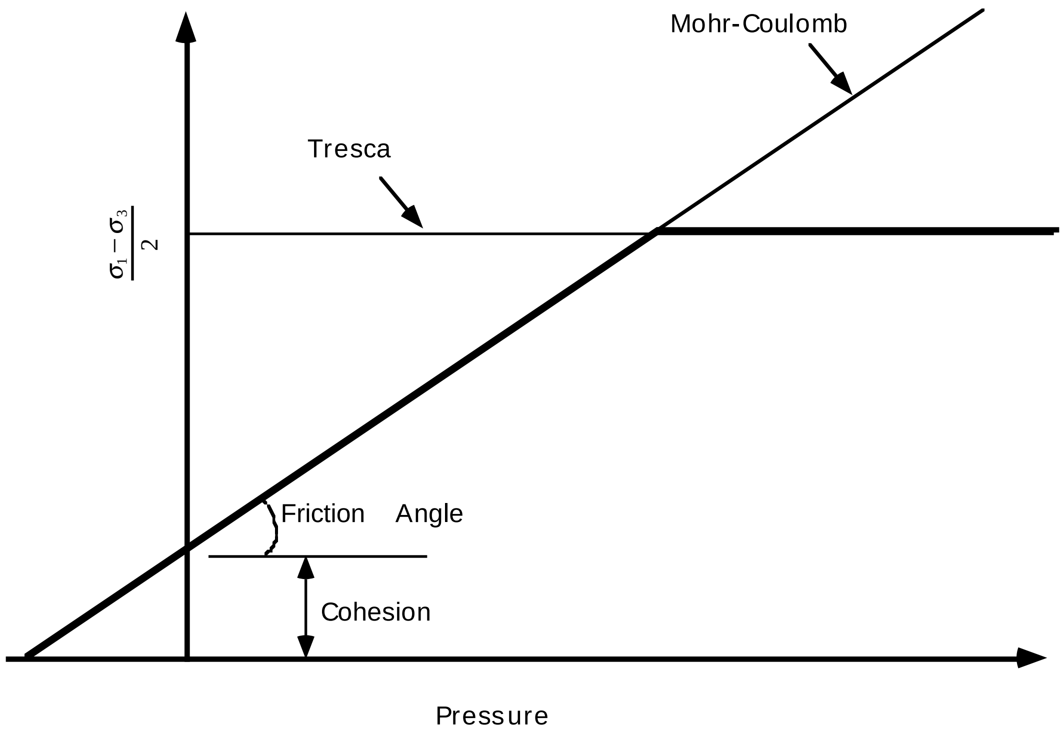


Figure 3.9. Mohr-Coulomb surface with a Tresca limit.

To invoke Mode I of this model, set a_0, a_1, a_2, b_1, a_{0f}, and a_{1f} to zero. The tabulated values of pressure should then be specified on cards 5 and 6, and the corresponding values of yield stress should be specified on cards 7 and 8. The parameters relating to reinforcement properties, initial yield stress, and tangent modulus are not used in this response mode, and should be set to zero.

Simple tensile failure

Note that a_{1f} is reset internally to 1/3 even though it is input as zero; this defines a failed material curve of slope 3*p*, where *p* denotes pressure (positive in compression). In this case the yield strength is taken from the tabulated yield vs. pressure curve until the maximum principal stress(σ_1) in the element exceeds the tensile cut-off (σ_{cut}). For every time step that $\sigma_1 > \sigma_{cut}$ the yield strength is scaled back by a fraction of the distance between the two curves until after 20 time steps the yield strength is defined by the failed curve. The only way to inhibit this feature is to set σ_{cut} arbitrarily large.

Response Mode II. Two Curve Model with Damage and Failure

This approach uses two yield versus pressure curves of the form

$$\sigma_y = a_0 + \frac{p}{a_1 + a_2 p}$$

The upper curve is best described as the maximum yield strength curve and the lower curve is the failed material curve. There are a variety of ways of moving between the two curves and each is discussed below.

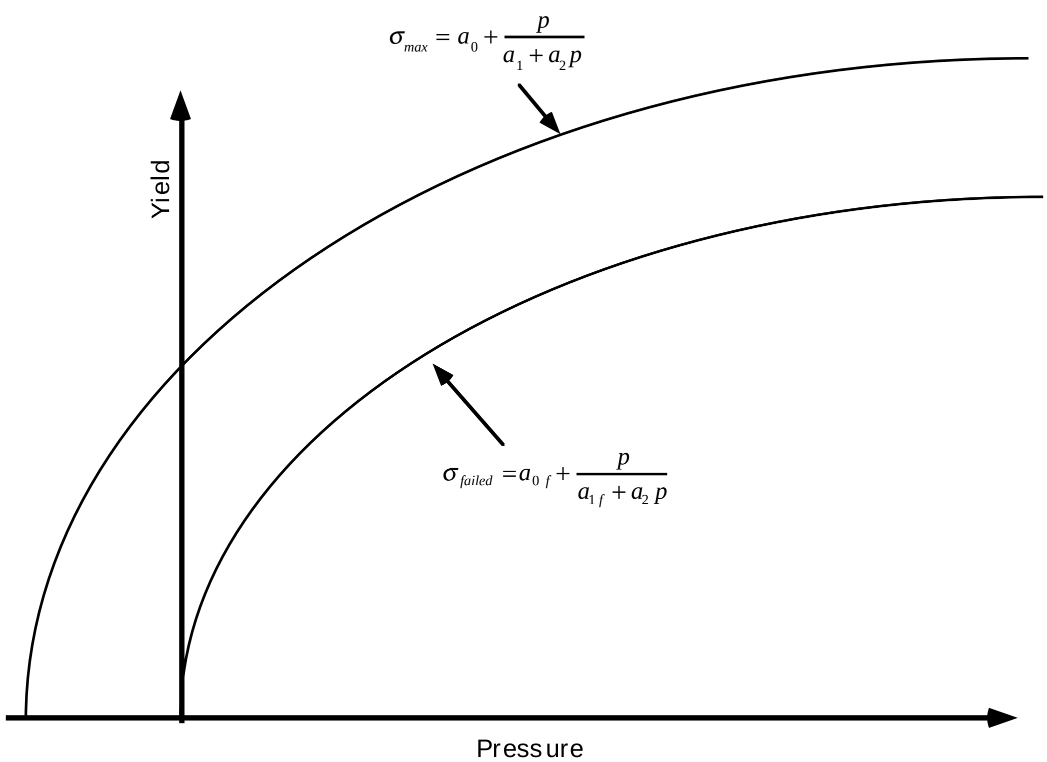


Figure 3.10. Two-curve concrete model with damage and failure.

MODE II. A: Simple tensile failure

Define a_0, a_1, a_2, a_{0f} and a_{1f}, set b_1 to zero, and leave cards 5 through 8 blank. In this case the yield strength is taken from the maximum yield curve until the maximum principal stress (σ_1) in the element exceeds the tensile cut-off (σ_{cut}). For every time step that $\sigma_1 > \sigma_{cut}$ the yield strength is scaled back by a fraction of the distance between the two curves until after 20 time steps the yield strength is defined by the failure curve.

Mode II.B: Tensile failure plus plastic strain scaling

Define a_0, a_1, a_2, a_{0f} and a_{1f}, set b_1 to zero, and user cards 5 through 8 to define a scale factor, η, versus effective plastic strain. LS-DYNA evaluates η at the current effective plastic strain and then calculated the yield stress as

$$\sigma_{yield} = \sigma_{failed} + \eta\left(\sigma_{max} - \sigma_{failed}\right)$$

where σ_{max} and σ_{failed} are found as shown in Figure 3.10. This yield strength is then subject to scaling for tensile failure as described above. This type of model allows the description of a strain hardening or softening material such as concrete.

Model II.C: Tensile failure plus damage scaling

The change in yield stress as a function of plastic strain arises from the physical mechanisms such as internal cracking, and the extent of this cracking is affected by the hydrostatic pressure when the cracking occurs. This mechanism gives rise to the "confinement" effect on concrete behavior. To account for this phenomenon a "damage" function was defined and incorporated. This damage function is given the form:

$$\lambda = \int_0^{\varepsilon^p} \left(1 + \frac{p}{\sigma_{cut}}\right)^{-b_1} d\varepsilon^p$$

Define a_0, a_1, a_2, a_{0f} and a_{1f}, and b_1. Cards 5 though 8 now give η as a function of λ and scale the yield stress as

$$\sigma_{yield} = \sigma_{failed} + \eta\left(\sigma_{max} - \sigma_{failed}\right)$$

and then apply any tensile failure criteria.

Mode II Concrete Model Options

Material Type 16 Mode II provides for the automatic internal generation of a simple "generic" model from concrete if $a0$ is negative then σ_{cut} is assumed to be the unconfined concrete compressive strength, $f_c^{'}$, and $-a_0$ is assumed to be a conversion faction from LS-DYNA pressure units to psi. In this case the parameter values generated internally are

$$\sigma_{cut} = 1.7\left(\frac{f_c^{'2}}{-a_0}\right)^{\frac{1}{3}}$$

$$a_0 = \frac{f_c^{'}}{4}$$

$$a_1 = \frac{1}{3}$$

$$a_2 = \frac{1}{3 f_c^{'}}$$

$$a_{0f} = 0$$

$$a_{1f} = 0.385$$

Note that these $a_{0\mathrm{f}}$ and $a_{1\mathrm{f}}$ defaults will be overridden by non zero entries on Card 3. If plastic strain or damage scaling is desired, Cards 5 through 8 and $b1$ should be specified in the input. When a_0 is input as a negative quantity, the equation-of-state can be given as 0 and a trilinear EOS Type 8 model will be automatically generated from the unconfined compressive strength and Poisson's ratio. The EOS 8 model is a simple pressure versus volumetric strain model with no internal energy terms, and should give reasonable results for pressures up to 5kbar (approximately 75,000 psi).

Mixture model

A reinforcement fraction, f_r, can be defined along with properties of the reinforcement material. The bulk modulus, shear modulus, and yield strength are then calculated from a simple mixture rule, i.e., for the bulk modulus the rule gives:

$$K = (1 - f_r)K_m + f_r K_r$$

where K_m and K_r are the bulk modulii for the geologic material and the reinforcement material, respectively. This feature should be used with caution. It gives an isotropic effect in the material instead of the true anisotropic material behavior. A reasonable approach would be to use the mixture elements only where the reinforcing exists and plain elements elsewhere. When the mixture model is being used, the strain rate multiplier for the principal material is taken from load curve N1 and the multiplier for the reinforcement is taken from load curve N2.

A Suggestion

The LLNL DYNA3D manual from 1991 [Whirley and Hallquist] suggests using the damage function (Mode 11.C.) in Material Type 16 with the following set of parameters:

$$a_0 = \frac{f_c^{'}}{4}$$
$$a_1 = \frac{1}{3}$$
$$a_2 = \frac{1}{3 f_c^{'}}$$
$$a_{0f} = \frac{f_c^{'}}{10}$$
$$a_{1f} = 1.5$$
$$b_1 = 1.25$$

and a damage table of:

Card 5:	0.0	8.62E-06	2.15E-05	3.14E-05	3.95E-04
	5.17E-04	6.38E-04	7.98E-04		
Card 6:	9.67E-04	1.41E-03	1.97E-03	2.59E-03	3.27E-03
	4.00E-03	4.79E-03	0.909		
Card 7:	0.309	0.543	0.840	0.975	1.000
	0.790	0.630	0.469		
Card 8:	0.383	0.247	0.173	0.136	0.114
	0.086	0.056	0.0		

This set of parameters should give results consistent with Dilger, Koch, and Kowalczyk, [1984] for plane concrete. It has been successfully used for reinforced structures where the reinforcing bars were modeled explicitly with embedded beam and shell elements. The model does not incorporate the major failure mechanism - separation of the concrete and reinforcement leading to catastrophic loss of confinement pressure. However, experience indicates that this physical behavior will occur when this model shows about 4% strain.

Material Type 17 (Isotropic Elastic-Plastic Oriented Crack Model)

Columns	Quantity		Format
1-10	Card 3	Young's modulus	E10.0
1-10	Card 4	Poisson's ratio	E10.0
1-10	Card 5	Yield stress	E10.0
1-10	Card 6	Hardening modulus	E10.0
1-10	Card 7	Fracture strength	E10.0
1-10	Card 8	Pressure cutoff ($\leq$ 0.)	E10.0

When the maximum principal stress exceeds the fracture stress, the element fails on a plane perpendicular to the direction of the maximum principal stress. In tension, the element will not carry any stresses on the fracture plane, but, in compression, it will carry both normal and shear stresses. If the fracture stress is exceeded in another direction, the element fails isotropically: The element loses its ability to carry tension, the deviatoric stresses are set to zero, and the material behaves as a fluid.

Material Type 18 (Power Law Isotropic Plasticity)

Columns		Quantity	Format
1-10	Card 3	Young's modulus	E10.0
11-20		Poisson's ratio	E10.0
21-30		k, strength coefficient	E10.0
31-40		n, hardening exponent	E10.0
41-50		C, strain rate parameter	E10.0
51-60		p, strain rate parameter	E10.0
61-70		SIGY, optional input for determining strain to yield. LT.0.02: $\varepsilon_{yp} = SIGY$ GE.0.02: See below.	E10.0
71-80		Formulation for rate effects EQ.0.0: Scale yield stress (default) EQ.1.0: Viscoplastic formulation	E10.0
	Card 4	Blank	
	.		
	.		
	.		
	Card 8	Blank	

Elastoplastic behavior with isotropic hardening is provided by this model. The yield stress, σ_y, is a function of plastic strain and obeys the equation:

$$\sigma_y = k\, \varepsilon^n = k\left(\varepsilon_{yp} + \bar{\varepsilon}^{p}\right)^n$$

where ε_{yp} is the elastic strain to yield and $\bar{\varepsilon}^{p}$is the effective plastic strain (logrithmic). If SIGY is set to zero, the strain to yield if found by solving for the intersection of the linearly elastic loading equation with the strain hardening equation:

$$\sigma = E\,\varepsilon$$

$$\sigma = k\,\varepsilon^n$$

which gives the elastic strain at yield as:

$$\varepsilon_{yp} = \left(\frac{E}{k}\right)^{\left[\frac{1}{n-1}\right]}$$

If SIGY yield is nonzero and greater than 0.02 then:

$$\varepsilon_{yp} = \left(\frac{\sigma_y}{k}\right)^{\left[\frac{1}{n}\right]}$$

Strain rate is accounted for using the Cowper and Symonds model which scales the yield stress with the factor

$$1 + \left(\frac{\dot{\varepsilon}}{C}\right)^{1/P}$$

where $\dot{\varepsilon}$ is the strain rate. A fully viscoplastic formulation is optional which incorporates the Cowper and Symonds formulation within the yield surface. An additional cost is incurred but the improvement is results can be dramatic.

Material Type 19 (Strain Rate Dependent Isotropic Plasticity)

Columns		Quantity	Format
1-10	Card 3	Young's modulus	E10.0
11-20		Poisson's ratio	E10.0
21-30		Load curve number defining σ_0 as a function of strain rate.	E10.0
31-40		Hardening modulus, E_t	E10.0
41-50		Load curve number (optional) defining Young's modulus as a function of strain rate.	E10.0
51-60		Load curve number (optional) defining the tangent modulus as a function of strain rate.	E10.0
61-70		Load curve number (optional) defining the von Mises stress at failure as a function of strain rate.	E10.0
71-80		Time step size for automatic element deletion (shells only).	E10.0
1-10	Card 4	Redefinition of failure curve: EQ.1.0: Effective plastic strain, EQ.2.0: Maximum principal stress.	E10.0
11-20		Formulation for rate effects: EQ.0.0: Scale yield stress (default) EQ.1.0: Viscoplastic formulation	E10.0
	Card 5	Blank	
	.		
	.		
	.		
	Card 8	Blank	

In this model, a load curve is used to describe the yield strength σ_0 as a function of effective strain rate $\dot{\bar{\varepsilon}}$ where

$$\dot{\bar{\varepsilon}} = \left(\frac{2}{3} \dot{\varepsilon}'_{ij} \, \dot{\varepsilon}'_{ij} \right)^{1/2}$$

and the prime denotes the deviatoric component. The yield stress is defined as

$$\sigma_y = \sigma_0 \left(\dot{\bar{\varepsilon}}\right) + E_p \; \bar{\varepsilon}^p$$

where $\bar{\varepsilon}^p$ is the effective plastic strain and E_h is given in terms of Young's modulus and the tangent modulus by

$$E_p = \frac{E \; E_t}{E - E_t} \; .$$

Both Young's modulus and the tangent modulus may optionally be made functions of strain rate by specifying a load curve ID giving their values as a function of strain rate. If these load curve ID's are input as 0, then the constant values specified in the input are used.

Note that all load curves used to define quantities as a function of strain rate must have the same number of points at the same strain rate values. This requirement is used to allow vectorized interpolation to enhance the execution speed of this constitutive model.

This model also contains a simple mechanism for modeling material failure. This option is activated by specifying a load curve ID defining the effective stress at failure as a function of strain rate. For solid elements, once the effective stress exceeds the failure stress the element is deemed to have failed and is removed from the solution. For shell elements the entire shell element is deemed to have failed if all integration points through the thickness have an effective stress that exceeds the failure stress. After failure the shell element is removed from the solution.

In addition to the above failure criterion, this material model also supports a shell element deletion criterion based on the maximum stable time step size for the element, Δt_{max}. Generally, Δt_{max} goes down as the element becomes more distorted. To assure stability of time integration, the global LS-DYNA time step is the minimum of the Δt_{max} values calculated for all elements in the model. Using this option allows the selective deletion of elements whose time step Δt_{max} has fallen below the specified minimum time step, Δt_{crit}. Elements which are severely distorted often indicate that material has failed and supports little load, but these same elements may have very small time steps and therefore control the cost of the analysis. This option allows these highly distorted elements to be deleted from the calculation, and, therefore, the analysis can proceed at a larger time step, and, thus, at a reduced cost. Deleted elements do not carry any load, and are deleted from all applicable slide surface definitions. Clearly, this option must be judiciously used to obtain accurate results at a minimum cost.

A fully viscoplastic formulation is optional which incorporates the rate formulation within the yield surface. An additional cost is incurred but the improvement is results can be dramatic.

Material Type 20 (Rigid)

Columns		Quantity	Format
1-10	Card 3	Young's modulus	E10.0
51-60		MADYMO3D [TNO, 1990] **(not CAL3D)** coupling flag, n EQ.0: use normal LS-DYNA rigid body updates GT.0: the rigid body is coupled to MADYMO ellipsoid number n. LT.0: the rigid body is coupled to MADYMO plane number |n|.	E10.0
61-70		MADYMO3D/CAL3D coupling option EQ.-1: attach VDA surface defined after the cross-section input and automatically generate a mesh for viewing the surface in LS-TAURUS. EQ.0: the undeformed geometry input to LS-DYNA corresponds to the local system for MADYMO/CAL3D. Mesh is input. EQ.1: the undeformed geometry input to LS-DYNA corresponds to the global system for MADYMO/CAL3D. EQ.2: generate a mesh for the ellipsoids and planes internally in LS-DYNA . EQ.3: generate MADYMO seatbelts. EQ.4: input a contact entity and generate a mesh. This data follows the Section cards.	E10.0
71-80		MADYMO/CAL3D Coupling flag/number of entities: EQ.0: use normal LS-DYNA rigid body updates EQ.n: this rigid body corresponds to MADYMO/ CAL3D system number n. Rigid body updates are performed by MADYMO/CAL3D. If the coupling option in columns 61-70 above equals 4 then n is the number of entities for mesh generation.	E10.0
1-10	Card 4	Poisson's ratio	E10.0
1-10	Card 5	Center of mass constraint option, CMO EQ.+1.0: constraints as defined below EQ.-1.0: SPC constraint	E10.0

Material Type 20 (Rigid)

Columns		Quantity	Format
1-10	Card 6	Local coordinate system number, CMO=-1.0 or translational constraint type defined below, CMO=1.0 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	E10.0
1-10	Card 7	SPC constraint number (between 0 and 111111. Sec. 13) CMO=-1.0 or rotational constraint boundary code (CMO=1.0) 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	E10.0
1-10	Card 8	Local coordinate system (see Section 14) number for output.	E10.0
		*****Alternative method for specifying local system below.*****	
1-60	Card 8	Define two vectors **a** and **v**, fixed in the rigid body which are used for output and the user defined airbag sensor subroutines. The output parameters are in the directions **a**, **b**, and **c** where the latter are given by the cross products $\mathbf{c}=\mathbf{a}\times\mathbf{v}$ and $\mathbf{b}=\mathbf{c}\times\mathbf{a}$. This input is optional.	6E10.0

The rigid material type 20 provides a convenient way of turning one or more parts comprised of beams, shells, or solid elements into a rigid body. Approximating a deformable body as rigid is a preferred modeling technique in many real world applications. For example, in sheet metal forming problems the tooling can properly and accurately be treated as rigid. In the design of restraint systems the occupant can, for the purposes of early design studies, also be treated as rigid. Elements which are rigid are bypassed in the element processing and no storage is allocated for storing history variables; consequently, the rigid material type is very cost efficient.

Two unique rigid part ID's may not share common nodes unless they are merged together using the rigid body merge option. A rigid body may be made up of disjoint finite element meshes, however. LS-DYNA assumes this is the case since this is a common practice in setting up tooling meshes in forming problems.

All elements which reference a given part ID corresponding to the rigid material should be contiguous, but this is not a requirement. If two disjoint groups of elements on opposite sides of a model are modeled as rigid, separate part ID's should be created for each of the contiguous element groups if each group is to move independently. This requirement arises from the fact that LS-DYNA internally computes the six rigid body degrees-of-freedom for each rigid body (rigid material or set of merged materials), and if disjoint groups of rigid elements use the same part ID, the disjoint groups will move together as one rigid body.

Inertial properties for rigid materials may be defined in either of two ways. By default, the inertial properties are calculated from the geometry of the constitutent elements of the rigid material and the density specified for the part ID. Alternatively, the inertial properties and initial velocities for a rigid body may be directly defined, and this overrides data calculated from the material property definition and nodal initial velocity definitions.

Young's modulus, E, and Poisson's ratio, υ, are used for determining sliding interface parameters if the rigid body interacts in a contact definition. Realistic values for these constants should be defined since unrealistic values may contribute to numerical problem in contact.

Material Type 21 (Thermal Orthotropic Elastic)

Columns		Quantity	Format
1-10	Card 3	E_a, (see Figure 3.5)	E10.0
11-20		E_b	E10.0
21-31		E_c	E10.0
1-10	Card 4	υ_{ba}	E10.0
11-20		υ_{ca}	E10.0
21-30		υ_{cb}	E10.0
31-40		α_a, (coefficient of thermal expansion)	E10.0
41-50		α_b,	E10.0
51-60		α_c,	E10.0
1-10	Card 5	G_{ab}	E10.0
11-20		G_{bc}	E10.0
21-30		G_{ca}	E10.0
1-10	Card 6	Materials axes option AOPT EQ.0.0: locally orthotropic with material axes determined by element nodes n_1, n_2, and n_4 as shown in Figure 3.5. Card 7 and 8 below are blank with this option. EQ.1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center. Card 8 below is blank. EQ.2.0: globally orthotropic with material axes determined by vectors defined on cards 7 and 8. EQ.3.0: applicable to shell elements only. This option determines locally orthotropic material axes by offsetting the material axes by an angle (Card 8) from a line in the plane of the shell determined by taking the cross product of the vector defined on card 7 with the shell normal vector.	E10.0
11-20		Use reference geometry to initial stresses. EQ.0.0: off EQ.1.0: on	E10.0

Columns		Quantity	Format
1-10	Card 7	x_p, define for AOPT=1.0	E10.0
11-20		y_p, define for AOPT=1.0	E10.0
21-30		z_p, define for AOPT=1.0	E10.0
1-10	Card 7	a_1, define for AOPT=2.0	E10.0
11-20		a_2, define for AOPT=2.0	E10.0
21-30		a_3, define for AOPT=2.0	E10.0
1-10	Card 7	v_1, define for AOPT=3.0	E10.0
11-20		v_2, define for AOPT=3.0	E10.0
21-30		v_3, define for AOPT=3.0	E10.0
1-10	Card 8	d_1, define for AOPT=2.0	E10.0
11-20		d_2, define for AOPT=2.0	E10.0
21-30		d_3, define for AOPT=2.0	E10.0
1-10	Card 8	Material angle beta (may be overridden on the element card) (degrees)	E10.0

Material Type 22 (Composite Damage Model)

Columns		Quantity	Format
1-10	Card 3	E_a, longitudinal direction	E10.0
11-20		E_b, transverse direction	E10.0
21-30		E_c , normal direction	E10.0
31-40		K_f, bulk modulus of failed material *(solid element only)*	E10.0
41-50		S_n, normal tensile strength *(solid element only)*	E10.0
51-60		S_{yz}, transverse shear strength *(solid element only)*	E10.0
1-10	Card 4	υ_{ba}	E10.0
11-20		υ_{ca}	E10.0
21-30		υ_{cb}	E10.0
1-10	Card 5	G_{ab}	E10.0
11-20		G_{bc}	E10.0
21-30		G_{ca}	E10.0
1-10	Card 6	Material axes option, AOPT EQ.0.0: locally orthotropic with material axes determined by element nodes n_1, n_2, and n_4 as shown in Figure 3.5. Cards 7 and 8 below are blank with this option. EQ.1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center. Card 8 below is blank. EQ.2.0: globally orthotropic with material axes determined by vectors defined on Cards 7 and 8. EQ.3.0: applicable to shell elements only. This option determines locally orthotropic material axes by offsetting the material axes by an angle (Card 12) from a line in the plane of the shell determined by taking the cross product of the vector defined on Card 7 with the shell normal vector.	E10.0
11-20	Material axes change flag for brick elements EQ.1.0: default EQ.2.0: switch material axes a and b EQ.3.0: switch material axes a and c		E10.0

Material Type 22 (Composite Damage Model)

Columns		Quantity	Format
1-10	Card 7	x_p, define for AOPT=1.0	E10.0
11-20		y_p, define for AOPT=1.0	E10.0
21-30		z_p, define for AOPT=1.0	E10.0
31-40		S_{zx}, transverse shear strength *(solid element only)*	E10.0
1-10	Card 7	a_1, define for AOPT=2.0	E10.0
11-20		a_2, define for AOPT=2.0	E10.0
21-30		a_3, define for AOPT=2.0	E10.0
31-40		S_{zx}, transverse shear strength *(solid element only)*	E10.0
1-10	Card 7	v_1, define for AOPT=3.0	E10.0
11-20		v_2, define for AOPT=3.0	E10.0
21-30		v_3, define for AOPT=3.0	E10.0
31-40		S_{zx}, transverse shear strength *(solid element only)*	E10.0
1-10	Card 8	d_1, define for AOPT=2.0	E10.0
11-20		d_2, define for AOPT=2.0	E10.0
21-30		d_3, define for AOPT=2.0	E10.0
31-40		S_C, shear strength, ab plane	E10.0
41-50		x_t, longitudinal tensile strength, a-axis	E10.0
51-60		y_t, transverse tensile strength, b-axis	E10.0
61-70		y_c, transverse compressive strength	E10.0
71-80		Nonlinear shear stress parameter	E10.0

Define a material angle for each of the through-the-thickness integration points. For shell elements this data must follow Card 11.

The number of additional integration point variables for shells written to the LS-TAURUS database is input on Control Card 21 in Columns 11-15 as variable NEIPS. For Model 22 these additional vaiables are tabulated below (*ip* = shell integration point):

History Variable	Description	Value	LS-TAURUS Component
$ef(i)$	*tensile fiber mode*		*81*
$cm(i)$	*tensile matrix mode*	*1 - elastic*	*82*
$ed(i)$	*compressive matrix mode*	*0 - failed*	*83*

These variables can be plotted in LS-TAURUS as element components 81, 82, ..., 80+ NEIPS. The following components are stored as element component 7 instead of the effective plastic strain.:

Description	Integration point
$\frac{1}{nip}\sum_{i=1}^{nip} ef(i)$	*1*
$\frac{1}{nip}\sum_{i=1}^{nip} cm(i)$	*2*
$\frac{1}{nip}\sum_{i=1}^{nip} ed(i)$	*3*

Examples:

a) Fringe of tensile fiber mode for integration point 3:

LS-TAURUS commands: *intg 3 frin 81*

b) Sum of failure indicator of tensile matrix mode:

LS-TAURUS commands: *intg 2 frin 7*

Material Type 23 (Thermal Orthotropic Elastic with 12 Curves)

Columns		Quantity	Format
1-10	Number of points in material constant versus temperature curves NUMPTS (1< NUMPTS<49)		E10.0
1-10	Card 4	Blank	E10.0
1-10	Card 5	Blank	E10.0
1-10	Card 6	Material axes option, AOPT EQ.0.0: locally orthotropic with material axes determined by element nodes n_1, n_2, and n_4 as shown in Figure 3.5. Cards 7 and 8 below are blank with this option. EQ.1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center. Card 8 below is blank. EQ.2.0: globally orthotropic with material axes determined by vectors defined on Cards 7 and 8. EQ.3.0: applicable to shell elements only. This option determines locally orthotropic material axes by offsetting the material axes by an angle (Card 8) from a line in the plane of the shell determined by taking the cross product of the vector defined on Card 7 with the shell normal vector.	E10.0
11-20		Use reference geometry to initial stresses. EQ.0.0: off EQ.1.0: on	E10.0
1-10	Card 7	x_p, define for AOPT=1.0	E10.0
11-20		y_p, define for AOPT=1.0	E10.0
21-30		z_p, define for AOPT=1.0	E10.0
1-10	Card 7	a_1, define for AOPT=2.0	E10.0
11-20		a_2, define for AOPT=2.0	E10.0
21-30		a_3, define for AOPT=2.0	E10.0
1-10	Card 7	v_1, define for AOPT=3.0	E10.0

Columns		Quantity	Format
11-20		v_2, define for AOPT=3.0	E10.0
21-30		v_3, define for AOPT=3.0	E10.0
1-10	Card 8	d_1, define for AOPT=2.0	E10.0
11-20		d_2, define for AOPT=2.0	E10.0
21-31		d_3, define for AOPT=2.0	E10.0

Define the following card sets for each of the 12 orthotropic constants followed by the list of corresponding temperatures using the format (8E10.0). For shell elements, this data must follow cards 10 and 11.

Cards 9, 10,...(bricks)
Cards 12, 13,...(shells)

Columns	Quantity	Format
1-10	E_a at temperature T_1	E10.0
11-20	E_a at temperature T_2	E10.0
.	.	.
.	.	.
.	.	.
71-80	E_a at temperature T_8	E10.0

Continue on additional cards until NUMPTS points have been defined. Definitions for variables, E_b, E_c, υ_{ba}, υ_{ca}, υ_{cb}, α_a, α_b, α_c, G_{ab}, G_{bc}, G_{ca}, and T (the list of temperatures) follow.

For shell elements only the material angles for each through the thickness integration point. must be defined following card 11.

Material Type 24 (Piecewise Linear Isotropic Plasticity)

Columns		Quantity	Format
1-10	Card 3	Young's modulus	E10.0
11-20		Strain rate parameter, C	E10.0
21-30		Strain rate parameter, p	E10.0
31-40		Formulation for rate effects: EQ.0.0: Scale yield stress (default) EQ.1.0: Viscoplastic formulation	E10.0
1-10	Card 4	Poisson's ratio	E10.0
1-10	Card 5	Yield stress	E10.0
11-20		Load curve ID or Table ID. The load curve ID defines effective stress versus effective plastic strain. Cards 7 and 8 are ignored with this option. The table ID, see Figure 3.9, defines for each strain rate value a load curve ID giving the stress versus effectiveplastic strain for that rate. The stress versus effective plastic strain curve for the lowest value of strain rate is used if the strain rate falls below the minimum value. Likewise, the stress versus effective plastic strain curve for the highest value of strain rate is used if the strain rate exceeds the maximum value. The strain rate parameters on card 3, the curve ID on card 6, and cards 7 and 8 are ignored if a Table ID is defined.	E10.0
1-10	Card 6	Tangent modulus, ignored if the stress-strain curve is defined below	E10.0
11-20		Failure flag: LT.0.0: User defined failure subroutine is called to determine failure EQ.0.0: Failure is not considered. This option is recommended if failure is not of interest since many caluculations will be saved. GT.0.0: Plastic strain to failure. When the plastic strain reaches this value, the element is deleted from the calculation.	E10.0
21-30		Time step size for automatic element deletion	E10.0
31-40		Load curve number to scale yield stress to account for strain rate effects.	E10.0

Material Type 24 (Piecewise Linear Isotropic Plasticity)

Columns		Quantity	Format
1-80	Card 7	Effective plastic strain values (define up to 8 points)	E10.0
1-80	Card 8	Corresponding yield stress values	E10.0

The stress strain behavior may be treated by a bilinear stress strain curve by defining the tangent modulus. Alternately, a stress versus effective plastic strain curve (Card 5, Columns 11-20) similar to that shown in Figure 3.8 can be used. If eight point are insufficient, a load curve may be used with an arbitrary number of points. The cost is roughly the same for either approach. The most general approach is to used the table definition, (Card 5, Columns 11-20) discussed below.

Three options to account for strain rate effects are possible.

I. Strain rate may be accounted for using the Cowper and Symonds model which scales the yield stress with the factor

$$1+\left(\frac{\dot{\varepsilon}}{C}\right)^{1/p}$$

where $\dot{\varepsilon}$ is the strain rate.

II. For complete generality a load curve (Card 5) to scale the yield stress may be input instead. In this curve the scale factor versus strain rate is defined.

III. If different stress versus strain curves can be provided for various strain rates, the option using the reference to a table (Card 5, Columns 11-20) can be used. Then the table input in Section 22 (Load Curve/Table Definitions) has to be used. See Figure 3.9.

A fully viscoplastic formulation is optional which incorporates the different options above within the yield surface. An additional cost is incurred over the simple scaling but the improvement is results can be dramatic.

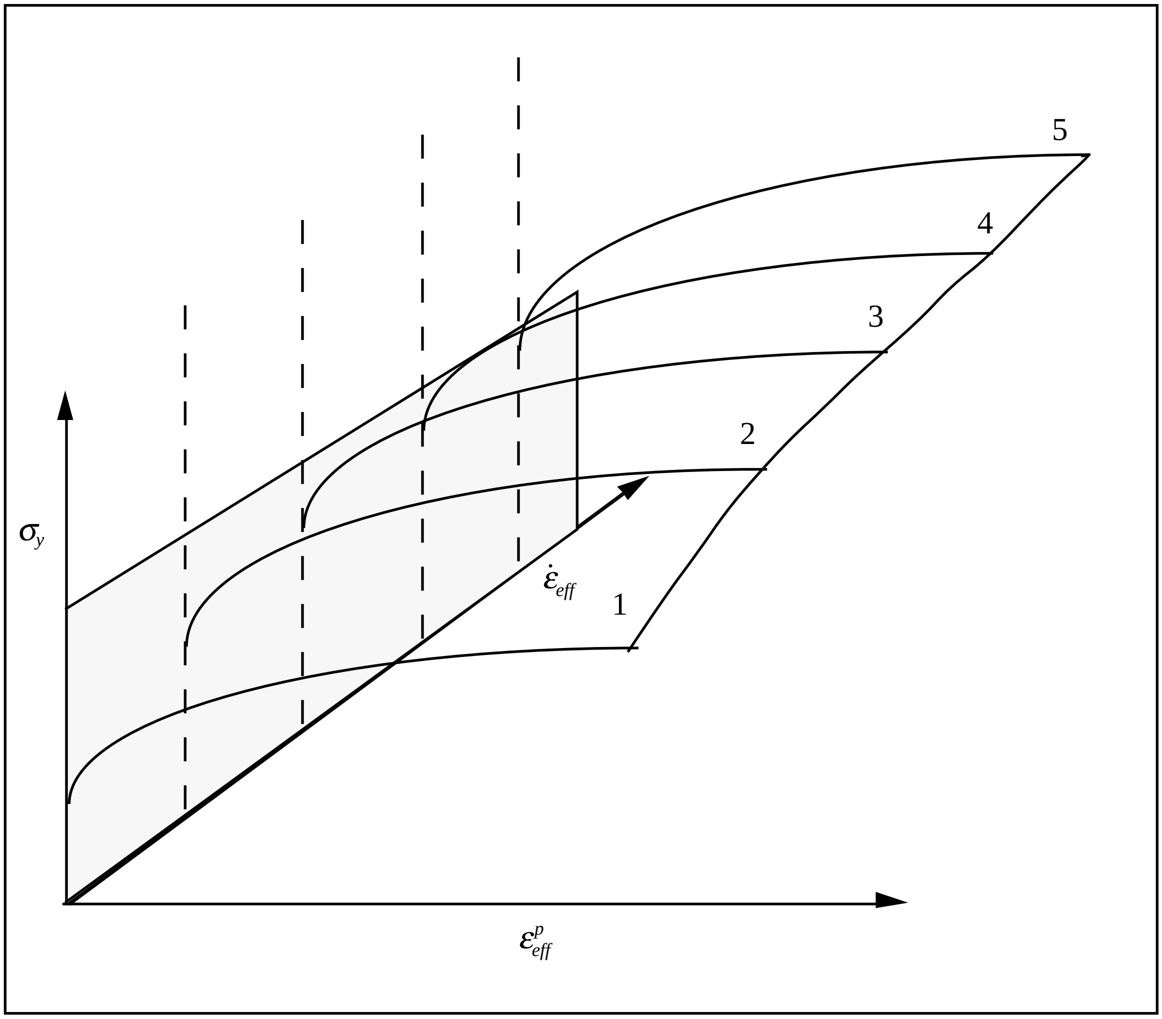


Figure 3.9. Rate effects may be accounted for by defining a table of curves. If a table ID is specified a curve ID is given for each strain rate, see Section 22. Intermediate values are found by interpolating between curves. Effective plastic strain versus yield stress is expected.

Material Type 25 (Kinematic Hardening Cap Model)

Columns		Quantity	Format
1-10	Card 3	Initial bulk modulus, K	E10.0
11-20		Initial shear modulus, G	E10.0
1-10	Card 4	Failure envelope parameter, α	E10.0
11-20		Failure envelope linear coefficient, θ	E10.0
21-30		Failure envelope exponential coefficient, γ	E10.0
31-40		Failure envelope exponent, β	E10.0
41-50		Cap, surface axis ratio, R	E10.0
1-10	Card 5	Hardening law exponent, D	E10.0
11-20		Hardening law coefficient, W	E10.0
21-30		Hardening law exponent, X_0	E10.0
31-40		Kinematic hardening coefficient, $\bar{c}$	E10.0
41-50		Kinematic hardening parameter, N	E10.0
1-10	Card 6	Plot database flag, IPLOT EQ.1.0: hardening variable, κ EQ.2.0: cap - J_1 axis intercept, X (κ) EQ.3.0: volumetric plastic strain, ε_V^p EQ.4.0: first stress invariant, J_1 EQ.5.0: second stress invariant, $\sqrt{J_{2D}}$ EQ.6.0: not used EQ.7.0: not used EQ.8.0: response mode number EQ.9.0: number of iterations	E10.0
1-10	Card 7	Formulation flag, ITYPE EQ.1.0: soil or concrete (Cap surface may contract) EQ.2.0: rock (Cap doesn't contract)	E10.0
11-20		Vectorization flag, IVEC EQ.0.0: vectorized (fixed numb of iterations) EQ.1.0: fully iterative	E10.0
1-10	Card 8	Tension cutoff, T < 0 (positive in compression)	E10.0

Material Type 25 (Kinematic Hardening Cap Model)

The implementation of an extended two invariant cap model, suggested by Stojko [1990], is based on the formulations of Simo, et. al. [1988, 1990] and Sandler and Rubin [1979]. In this model, the two invariant cap theory is extended to include nonlinear kinematic hardening as suggested by Isenberg, Vaughn, and Sandler [1978]. A brief discussion of the extended cap model and its parameters is given below.

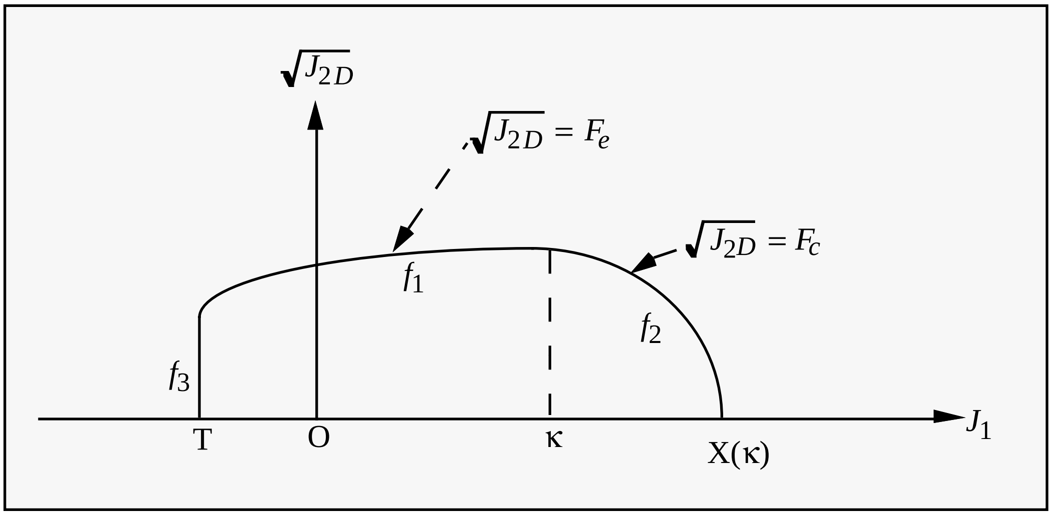


Figure 3.10. The yield surface of the two-invariant cap model in pressure $\sqrt{J_{2D}} - J_1$ space. Surface f_1 is the failure envelope, f_2 is the cap surface, and f_3 is the tension cutoff.

The cap model is formulated in terms of the invariants of the stress tensor. The square root of the second invariant of the deviatoric stress tensor, $\sqrt{J_{2D}}$ is found from the deviatoric stresses **s** as

$$\sqrt{J_{2D}} \equiv \sqrt{\frac{1}{2} s_{ij} s_{ij}}$$

and is the objective scalar measure of the distortional or shearing stress. The first invariant of the stress, J_1, is the trace of the stress tensor.

The cap model consists of three surfaces in $\sqrt{J_{2D}} - J_1$ space, as shown in Figure 3.10. First, there is a failure envelope surface, denoted f_1 in the figure. The functional form of f_1 is

$$f_1 = \sqrt{J_{2D}} - \min\left(F_e(J_1), T_{mises}\right) ,$$

where F_e is given by

$$F_e(J_1) \equiv \alpha - \gamma \exp(-\beta J_1) + \theta J_1$$

and $T_{mises} \equiv \left| X(\kappa_n) - L(\kappa_n) \right|$. This failure envelop surface is fixed in $\sqrt{J_{2D}} - J_1$ space, and therefore does not harden unless kinematic hardening is present. Next, there is a cap surface, denoted f_2 in the figure, with f_2 given by

$$f_2 = \sqrt{J_{2D}} - F_c(J_1, \kappa)$$

where F_c is defined by

$$F_c(J_1, \kappa) \equiv \frac{1}{R} \sqrt{[X(\kappa) - L(\kappa)]^2 - [J_1 - L(\kappa)]^2},$$

$X(\kappa)$ is the intersection of the cap surface with the J_1 axis

$$X(\kappa) = \kappa + RF_e(\kappa),$$

and $L(\kappa)$ is defined by

$$L(\kappa) \equiv \begin{cases} \kappa \text{ if } \kappa > 0 \\ 0 \text{ if } \kappa \le 0 \end{cases}.$$

The hardening parameter κ is related to the plastic volume change ε_v^p through the hardening law

$$\varepsilon_v^p = W\left\{1 - \exp\left[-D\left(X(\kappa) - X_0\right)\right]\right\}$$

Geometrically, κ is seen in the figure as the J_1 coordinate of the intersection of the cap surface and the failure surface. Finally, there is the tension cutoff surface, denoted f_3 in the figure. The function f_3 is given by

$$f_3 \equiv T - J_1,$$

where T is the input material parameter which specifies the maximum hydrostatic tension sustainable by the material. The elastic domain in $\sqrt{J_{2D}} - J_1$ space is then bounded by the failure envelope surface above, the tension cutoff surface on the left, and the cap surface on the right.

An additive decomposition of the strain into elastic and plastic parts is assumed:

$$\varepsilon = \varepsilon + \varepsilon^P ,$$

where $\boldsymbol{\varepsilon}^e$ is the elastic strain and $\boldsymbol{\varepsilon}^p$ is the plastic strain. Stress is found from the elastic strain using Hooke's law,

$$\sigma = C\left(\varepsilon - \varepsilon^P\right) ,$$

where $\boldsymbol{\sigma}$ is the stress and $\mathbf{C}$ is the elastic constitutive tensor.

The yield condition may be written

$$f_1(\sigma) \le 0$$
$$f_2(\sigma, \kappa) \le 0$$
$$f_3(\sigma) \le 0$$

and the plastic consistency condition requires that

$$\dot{\lambda}_k f_k = 0$$
$$k = 1, 2, 3$$
$$\dot{\lambda}_k \ge 0$$

where λ_k is the plastic consistency parameter for surface k. If $f_k < 0$ then, $\dot{\lambda}_k = 0$ and the response is elastic. If $f_k > 0$ then surface k is active and $\dot{\lambda}_k$ is found from the requirement that $\dot{f}_k = 0$.

Associated plastic flow is assumed, so using Koiter's flow rule the plastic strain rate is given as the sum of contribution from all of the active surfaces,

$$\dot{\varepsilon}^p = \sum_{k=1}^{3} \dot{\lambda}_k \frac{\partial f_k}{\partial s} .$$

One of the major advantages of the cap model over other classical pressure-dependent plasticity models is the ability to control the amount of dilatency produced under shear loading. Dilatency is produced under shear loading as a result of the yield surface having a positive slope in $\sqrt{J_{2D}} - J_1$ space, so the assumption of plastic flow in the direction normal to the yield surface produces a plastic strain rate vector that has a component in the volumetric (hydrostatic) direction (see Figure 3.10). In models such as the Drucker-Prager and Mohr-Coulomb, this dilatency continues as long as shear loads are applied, and in many cases produces far more dilatency than is experimentally observed in material tests. In the cap model, when the failure surface is active, dilatency is produced just as with the Drucker-Prager and Mohr-Columb models. However, the hardening law permits the cap surface to contract until the cap intersects the failure envelope at the stress point, and the cap remains at that point. The local normal to the yield surface is now vertical, and therefore the normality rule assures that no further plastic volumetric strain (dilatency) is created. Adjustment of the parameters that control the rate of cap contractions permits experimentally observed amounts of dilatency to be incorporated into the cap model, thus producing a constitutive law which better represents the physics to be modeled.

Another advantage of the cap model over other models such as the Drucker-Prager and Mohr-Coulomb is the ability to model plastic compaction. In these models all purely volumetric response is elastic. In the cap model, volumetric response is elastic until the stress point hits the cap surface. Therefore, plastic volumetric strain (compaction) is generated at a rate controlled by the hardening law. Thus, in addition to controlling the amount of dilatency, the introduction of the cap surface adds another experimentally observed response characteristic of geological material into the model.

The inclusion of kinematic hardening results in hysteretic energy dissipation under cyclic loading conditions. Following the approach of Isenberg, et. al. [1978] a nonlinear kinematic hardening law is used for the failure envelope surface when nonzero values of and N are specified. In this case, the failure envelope surface is replaced by a family of yield surfaces bounded by an initial yield surface and a limiting failure envelope surface. Thus, the shape of the yield surfaces described above remains unchanged, but they may translate in a plane orthogonal to the J axis,

Translation of the yield surfaces is permitted through the introduction of a "back stress" tensor, $\boldsymbol{\alpha}$. The formulation including kinematic hardening is obtained by replacing the stress $\boldsymbol{\sigma}$ with the translated stress tensor $\eta \equiv \sigma - \alpha$ in all of the above equation. The history tensor $\boldsymbol{\alpha}$ is assumed deviatoric, and therefore has only 5 unique components. The evolution of the back stress tensor is governed by the nonlinear hardening law

$$\alpha = \bar{c}\bar{F}(\sigma,\alpha)\dot{e}^{p}$$

where $\bar{c}$ is a constant, $\bar{F}$ is a scalar function of $\boldsymbol{\sigma}$ and $\boldsymbol{\alpha}$ and $\dot{e}^{p}$ is the rate of deviator plastic strain. The constant may be estimated from the slope of the shear stress - plastic shear strain curve at low levels of shear stress.

The function $\bar{F}$ is defined as

$$\bar{F} \equiv \max\left(0,1-\frac{(\sigma-\alpha)\bullet\alpha}{2NF_e\left(J_1\right)}\right)$$

where N is a constant defining the size of the yield surface. The value of N may be interpreted as the radial distant between the outside of the initial yield surface and the inside of the limit surface. In order for the limit surface of the kinematic hardening cap model to correspond with the failure envelope surface of the standard cap model, the scalar parameter a must be replaced α - N in the definition F_e.

The cap model contains a number of parameters which must be chosen to represent a particular material, and are generally based on experimental data. The parameters α, β, θ, and γ are usually evaluated by fitting a curve through failure data taken from a set of triaxial compression tests. The parameters W, D, and X_0 define the cap hardening law. The value W represent the void fraction of the uncompressed sample and D governs the slope of the initial loading curve in hydrostatic compression. The value of R is the ration of major to minor axes of the quarter ellipse defining the cap surface. Additional details and guidelines for fitting the cap model to experimental data re found in (Chen and Baladi, 1985).

Material Type 26 (Metallic Honeycomb)

Columns		Quantity	Format
1-10	Card 3	Young's modulus (for honeycomb material), E	E10.0
11-20		Poisson's ratio (for honeycomb material), ν	E10.0
21-30		Yield stress for fully compacted honeycomb material, σ_y	E10.0
31-40		LCA, load curve number for sigma-aa versus either relative volume or volumetric strain **(See Figure 3.11.)**	E10.0
41-50		LCB, load curve number for sigma-bb versus either relative volume or volumetric strain. (default: LCB=LCA)	E10.0
51-60		LCC, load curve number for sigma-cc versus either relative volume or volumetric strain. (default: LCC=LCA)	E10.0
61-70		LCS, load curve number for shear stress versus either relative volume or volumetric strain. (default LCS=LCA). Each component of shear stress may have its own load curve via Card 5 input.	E10.0
71-80		Relative volume at which the honeycomb is fully compacted, V_f	E10.0
	Card 4	The following honeycomb parameters must be defined for there are no defaults.	
1-10		Elastic modulus E_{aau} in uncompressed configuration	E10.0
11-20		Elastic modulus E_{bbu} in uncompressed configuration	E10.0
21-30		Elastic modulus E_{ccu} in uncompressed configuration	E10.0
31-40		Elastic shear modulus G_{abu} in uncompressed configuration	E10.0
41-50		Elastic shear modulus G_{bcu} in uncompressed configuration	E10.0
51-60		Elastic shear modulus G_{cau} in uncompressed configuration	E10.0
61-70		μ, material viscosity coefficient. (default=.05)	E10.0

Columns		Quantity	Format
71-80		Bulk viscosity flag. EQ.0.0: bulk viscosity is not used. This is recommended. EQ.1.0: bulk viscosity is active and μ=0 This will give results identical to previous versions of LS-DYNA .	E10.0
1-10	Card 5	LCAB, load curve number for sigma-ab versus either relative volume or volumetric strain (default: LCAB=LCS)	E10.0
11-20		LCBC, load curve number for sigma-bc versus either relative volume or volumetric strain. (default: LCBC=LCS)	E10.0
21-30		LCCA, load curve number for sigma-ca versus either relative volume or volumetric strain. (default: LCCA=LCS)	E10.0
31-40		LCSR, optional load curve number for strain-rate effects	E10.0
41-50		Tensile strain at element failure (element will erode)	E10.0
51-60		Shear strain at element failure (element will erode)	E10.0
1-10	Card 6	Material axes option, AOPT EQ.0.0: locally orthotropic with material axes determined by element nodes as shown in Figure 3.5. Cards 7 and 8 are blank with this option. EQ.1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center. Card 8 below is blank. EQ.2.0: globally orthotropic with material axes determined by vectors defined on Cards 7 and 8	E10.0
1-10	Card 7	x_p, define for AOPT = 1.0	E10.0
11-20		y_p, define for AOPT = 1.0	E10.0
21-30		z_p, define for AOPT = 1.0	E10.0
1-10	Card 7	a_1, define for AOPT = 2.0	E10.0
11-20		a_2, define for AOPT = 2.0	E10.0
21-30		a_3, define for AOPT = 2.0	E10.0
1-10	Card 8	d_1, define for AOPT = 2.0	E10.0

Columns	Quantity	Format
11-20	d_2, define for AOPT = 2.0	E10.0
21-30	d_3, define for AOPT = 2.0	E10.0

For efficiency it is strongly recommended that the load curve ID's: LCA, LCB, LCC, LCS, LCAB, LCBC, and LCCA, contain exactly the same number of points with corresponding strain values on the abcissa. If this recommendation is followed the cost of the table lookup is insignificant. Conversely, the cost increases significantly if the abcissa strain values are not consistent between load curves.

The behavior before compaction is orthotropic where the components of the stress tensor are uncoupled, i.e., an a component of strain will generate resistance in the local *a*-direction with no coupling to the local *b* and *c* directions. The elastic modulii vary from their initial values to the fully compacted values linearly with the relative volume:

$$E_{aa} = E_{aau} + \beta(E - E_{aau})$$

$$E_{bb} = E_{bbu} + \beta(E - E_{bbu})$$

$$E_{cc} = E_{ccu} + \beta(E - E_{ccu})$$

$$G_{ab} = G_{abu} + \beta(G - G_{abu})$$

$$G_{bc} = G_{bcu} + \beta(G - G_{bcu})$$

$$G_{ca} = G_{cau} + \beta(G - G_{cau})$$

where

$$\beta = \max\left[\min\left(\tfrac{1-V}{1-V_f}, 1\right), 0\right]$$

and G is the elastic shear modulus for the fully compacted honeycomb material

$$G = \frac{E}{2(1+\nu)}\ .$$

The relative volume, V, is defined as the ratio of the current volume over the initial volume, and typically, V=1 at the beginning of a calculation. The two bulk viscosity coefficients on the first card in columns 46-65 of Card 1 should be set to very small numbers to prevent the development of spurious pressure that may lead to undesirable and confusing results.

The load curves define the magnitude of the average stress as the material changes density (relative volume). Each curve related to this model must have the same number of points and the same abscissa values. There are two ways to define these curves, **a)** as a function of relative volume (V) or **b)** as a function of volumetric strain defined as:

$$\varepsilon_V = 1 - V$$

In the former, the first value in the curve should correspond to a value of relative volume slightly less than the fully compacted value. In the latter, the first value in the curve should be less than or equal to zero corresponding to tension and increase to full compaction. **Care should be taken when defining the curves so the extrapolated values do not lead to negative yield stresses.**

At the beginning of the stress update we transform each element's stresses and strain rates into the local element coordinate system. For the uncompacted material, the trial stress components are updated using the elastic interpolated modulii according to:

$$\sigma_{aa}^{n+1^{trial}} = \sigma_{aa}^{n} + E_{aa}\Delta\varepsilon_{aa}$$

$$\sigma_{bb}^{n+1^{trial}} = \sigma_{bb}^{n} + E_{bb}\Delta\varepsilon_{bb}$$

$$\sigma_{cc}^{n+1^{trial}} = \sigma_{cc}^{n} + E_{cc}\Delta\varepsilon_{cc}$$

$$\sigma_{ab}^{n+1^{trial}} = \sigma_{ab}^{n} + 2G_{ab}\Delta\varepsilon_{ab}$$

$$\sigma_{bc}^{n+1^{trial}} = \sigma_{bc}^{n} + 2G_{bc}\Delta\varepsilon_{bc}$$

$$\sigma_{ca}^{n+1^{trial}} = \sigma_{ca}^{n} + 2G_{ca}\Delta\varepsilon_{ca}$$

We then independently check each component of the updated stresses to ensure that they do not exceed the permissible values determined from the load curves, e.g., if

$$\left|\sigma_{ij}^{n+1^{trial}}\right| > \lambda\sigma_{ij}(V)$$

then

$$\sigma_{ij}^{n+1} = \sigma_{ij}(V)\frac{\lambda \sigma_{ij}^{n+1^{trial}}}{\left|\sigma_{ij}^{n+1^{trial}}\right|}$$

On Card 3 σ_{ij} (V) is defined in the load curve specified in columns 31-40 for the aa stress component, 41-50 for the bb component, 51-60 for the cc component, and 61-70 for the ab, bc, cb shear stress components. The parameter λ is either unity or a value taken from the load curve number, LCSR, that defines λ as a function of strain-rate. Strain-rate is defined here as the Euclidean norm of the deviatoric strain-rate tensor.

For fully compacted material we assume that the material behavior is elastic-perfectly plastic and updated the stress components according to:

$$s_{ij}^{trial} = s_{ij}^{n} + 2G\Delta\varepsilon_{ij}^{dev^{n+1/2}}$$

where the deviatoric strain increment is defined as

$$\Delta\varepsilon_{ij}^{dev} = \Delta\varepsilon_{ij} - \tfrac{1}{3}\Delta\varepsilon_{kk}\delta_{ij} \ .$$

We now check to see if the yield stress for the fully compacted material is exceeded by comparing

$$s_{eff}^{trial} = \left(\tfrac{3}{2}s_{ij}^{trial}s_{ij}^{trial}\right)^{1/2}$$

the effective trial stress to the yield stress, σ_y (Card 3, field 21-30). If the effective trial stress exceeds the yield stress we simply scale back the stress components to the yield surface

$$s_{ij}^{n+1} = \frac{\sigma_y}{s_{eff}^{trial}} s_{ij}^{trial} \ .$$

We can now update the pressure using the elastic bulk modulus, K

$$p^{n+1} = p^{n} - K\Delta\varepsilon_{kk}^{n+1/2}$$

$$K = \frac{E}{3(1-2\nu)}$$

and obtain the final value for the Cauchy stress

$$\sigma_{ij}^{n+1} = s_{ij}^{n+1} - p^{n+1}\delta_{ij} \ .$$

After completing the stress update we transform the stresses back to the global configuration.

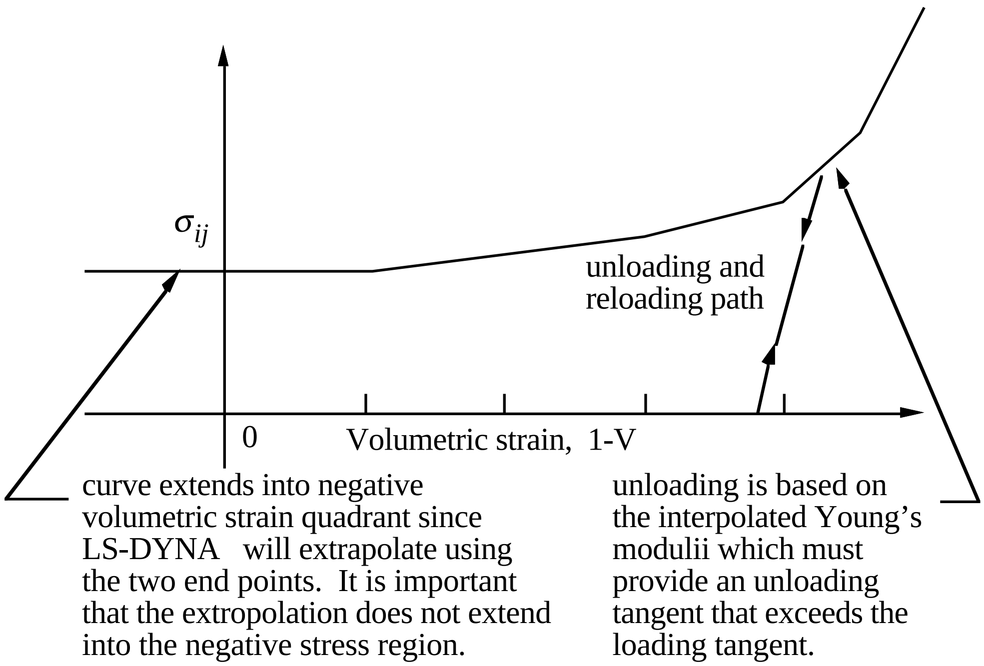


Figure 3.11. Stress quantity versus volumetric strain. Note that the "yield stress" at a volumetric strain of zero is nonzero. In the load curve definition the "time" value is the volumetric strain and the "function" value is the yield stress.

Material Type 27 (Slightly Compressible Mooney-Rivlin Rubber)

This material model provides an alternative to the Blatz-Ko rubber model. The implementation is due to [Maker 1987].

Columns		Quantity	Format
1-10	Card 3	A (If both *A* and *B* are input as 0.0 define the data on Card 4 below.)	E10.0
11-20		*B*	E10.0
21-30		υ, Poisson's ratio (> .49 is recommended-smaller values may not work.)	E10.0
31-40		Use reference geometry to initial stresses. EQ.0.0: off EQ.1.0: on	E10.0

If *A*=*B*=0.0 then a least square fit is computed with uniaxial data. Card 4 should contain the following information. Also see Figure 3.12.

Columns		Quantity	Format
1-10	Card 4	Specimen gauge length	E10.0
11-20		Specimen width	E10.0
21-30		Specimen thickness	E10.0
31-40		Load curve ID giving the force versus actual change in the gauge length.	E10.0
	Card 5	Blank	
.		.	.
.		.	.
.		.	.
	Card 8	Blank	

The strain energy density function is defined as:

$$W = A(\ I\text{-}3) + B(\ II\text{-}3) + C(\ III^{-2}\ \text{-}1\ ) + D(\ III\text{-}1)^2$$

where

$$C = 0.5\,A + B$$

$$D = \frac{A(5\upsilon - 2) + B(11\upsilon - 5)}{2(1 - 2\upsilon)}$$

υ = Poisson’s ratio

$2(A+B)$ = shear modulus of linear elasticity

I, II, III = invariants of right Cauchy-Green Tensor $\underset{\sim}{C}$.

The load curve definition that provides the uniaxial data should give the change in gauge length, ΔL, in columns 1-10 and the corresponding force in columns 11-20 if a 2E10.0 format is used. In compression both the force and the change in gauge length must be specified as negative values. In tension the force and change in gauge length should be input as positive values. The principal stretch ratio in the uniaxial direction, λ_1, is then given by

$$\lambda_1 = \frac{L_O + \Delta L}{L_O} \quad .$$

Alternatively, the stress versus strain curve can also be input by setting the gauge length, thickness, and width to unity (1.0) and defining the engineering strain in place of the change in gauge length and the nominal (engineering) stress in place of the force. See Figure 3.13.

The least square fit to the experimental data is performed during the initialization phase and is a comparison between the fit and the actual input is provided in the printed file. It is a good idea to visually check to make sure it is acceptable. The coefficients *A* and *B* are also printed in the output file.

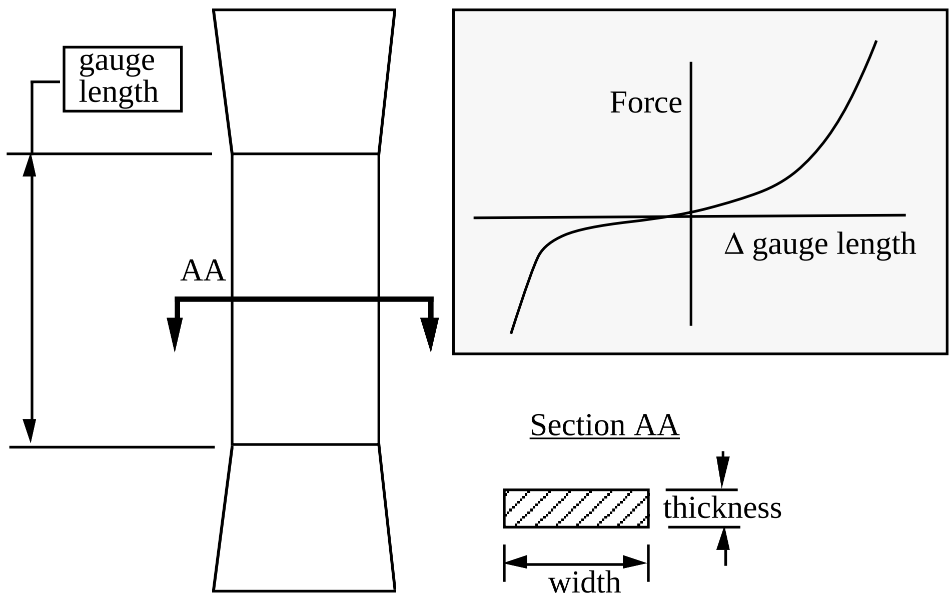


Figure 3.12. Uniaxial specimen for experimental data.

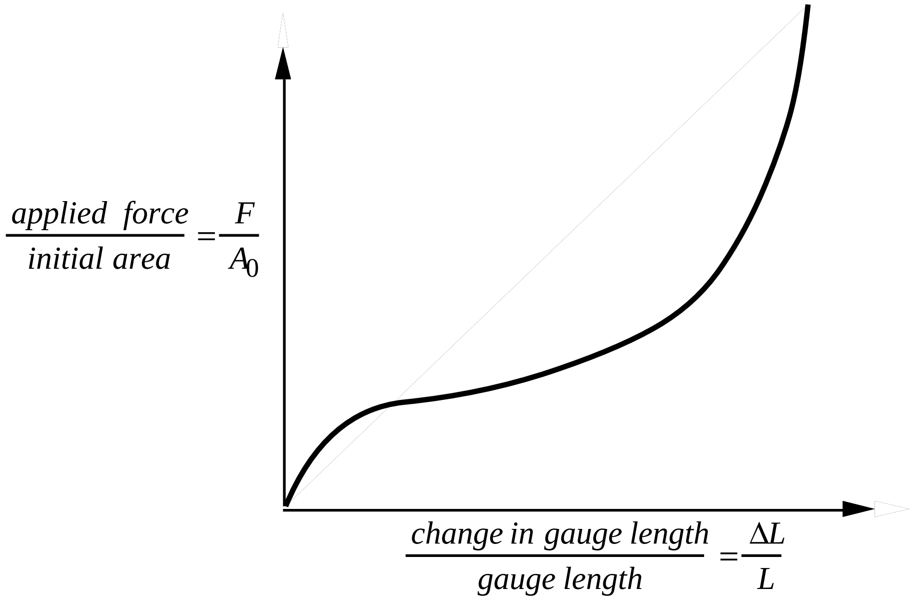


Figure 3.13. The stress versus strain curve can used instead of the force versus the change in the gauge length by setting the gauge length, thickness, and width to unity (1.0) and defining the engineering strain in place of the change in gauge length and the nominal (engineering) stress in place of the force.

Material Type 28 (Resultant Plasticity)

This model is available for the Belytschko-Schwer beam and the Belytschko-Tsay shell. For beams, the treatment is elastic-perfectly plastic, but for shell elements isotropic hardening is approximately modeled.

Columns		Quantity	Format
1-10	Card 3	Young’s modulus	E10.0
1-10	Card 4	Poisson’s ratio	E10.0
1-10	Card 5	Yield stress	E10.0
1-10	Card 6	Hardening modulus, E_t (shells only)	E10.0
1-10	Card 7	Blank	
1-10	Card 8	Blank	

Material Type 29 (FORCE LIMITED Resultant Formulation)

This material model is available for the Belytschko beam element only.

Plastic hinges form at the ends of the beam when the moment reaches the plastic moment. The moment-versus-rotation relationship is specified by the user in the form of a load curve and scale factor. The points of the load curve are (plastic rotation in radians, plastic moment). Both quantities should be positive for all points, with the first point being (zero, initial plastic moment). Within this constraint any form of characteristic may be used including flat or falling curves. Different load curves and scale factors may be specified at each node and about each of the local s and t axes.

Axial collapse occurs when the compressive axial load reaches the collapse load. The collapse load-versus-collapse deflection is specified in the form of a load curve. The points of the load curve are (true strain, collapse force). Both quantities should be entered as positive for all points, and will be interpreted as compressive (collapse does not occur in tension). The first point should be (zero, initial collapse load).

The collapse load may vary with end moment as well as with deflection. In this case, several load-deflection curves are defined, each corresponding to a different end moment. Each load curve should have the same number of points and the same deflection values. The end moment is defined as the average of the absolute moments at each end of the beam and is always positive.

It is not possible to make the plastic moment vary with axial load.

Columns		Quantity	Format
1-10	Card 3	Young's modulus	E10.0
11-20		Poisson's ratio	E10.0
21-30		Blank	10X
31-40		Damping factor, λ	E10.0

Leave this card blank if the axial collapse force is independent of the bending moment.

Columns		Quantity	Format
1-10	Card 4	End moment for first force versus strain curve, m_1	E10.0
11-20		End moment for second force versus strain curve, m_2	E10.0
.		.	.
.		.	.
.		.	.
71-80		End moment for eighth force versus strain curve, m_8	E10.0

Enter only one load curve if the axial collapse force is independent of bending moment.

Columns		Quantity	Format
1-10	Card 5	Load curve for collapse load versus strain, at end moment, m_1 (default: no axial collapse)	E10.0
11-20		Load curve for collapse load versus strain, at end moment, m_2 (default: collapse load is independent of moment)	E10.0
.		.	.
.		.	.
.		.	.
71-80		Load curve for collapse load versus strain, at end moment, m_8	E10.0

Note: The above curves are force versus strain unless the axial option on Card 6 is set to 1.0 in which case they give force versus change in length.

Columns		Quantity	Format
1-10	Card 6	Axial option EQ.0.0: axial load curves are collapse load versus strain EQ.1.0: axial load curves are collapse load versus change in length	E10.0

Columns	Quantity		Format
11-20		Flag to allow beam to yield in tension EQ.0.0: no yield EQ.1.0: can yield	E10.0
21-30	Load curve for torsional moment versus rotation		E10.0
31-40	Scale factor on torsional moment (default = 1.0)		E10.0
41-50	Torsional yield moment for interaction calculations (default set to le20 to prevent interaction)		E10.0
51-60	Axial elastic softening factor when hinge forms		E10.0

Card 7 is for moments about the s-axis.

1-10	Card 7	Load curve for plastic moment versus rotation at at node 1 (default: no hinge about s-axis)	E10.0
11-20		Scale factor on plastic moment at node 1 (default = 1.0)	E10.0
21 -30		Load curve for plastic moment versus rotation at node 2 (default: same as at node 1)	E10.0
31-40		Scale factor on plastic moment at node 2 (default: same as at node 1)	E10.0
41-50		Yield moment at node 1 for interaction calculations (default: set to le20 to prevent interaction)	E10.0
51-60		Yield moment at node 2 for interaction calculations (default: same as at node 1)	E10.0

Card 8 is for moments about the t-axis.

1-10	Card 8	Load curve for plastic moment versus rotation at at node 1 (default: no hinge about t-axis)	E10.0
11-20		Scale factor on plastic moment at node 1 (default = 1.0)	E10.0
21 -30		Load curve for plastic moment versus rotation at node 2 (default: same as at node 1)	E10.0
31-40		Scale factor on plastic moment at node 2 (default: same as at node 1)	E10.0
41-50		Yield moment at node 1 for interaction calculations (default: set to le20 to prevent interaction)	E10.0

Columns	Quantity	Format
51-60	Yield moment at node 2 for interaction calculations (default: same as at node 1)	E10.0

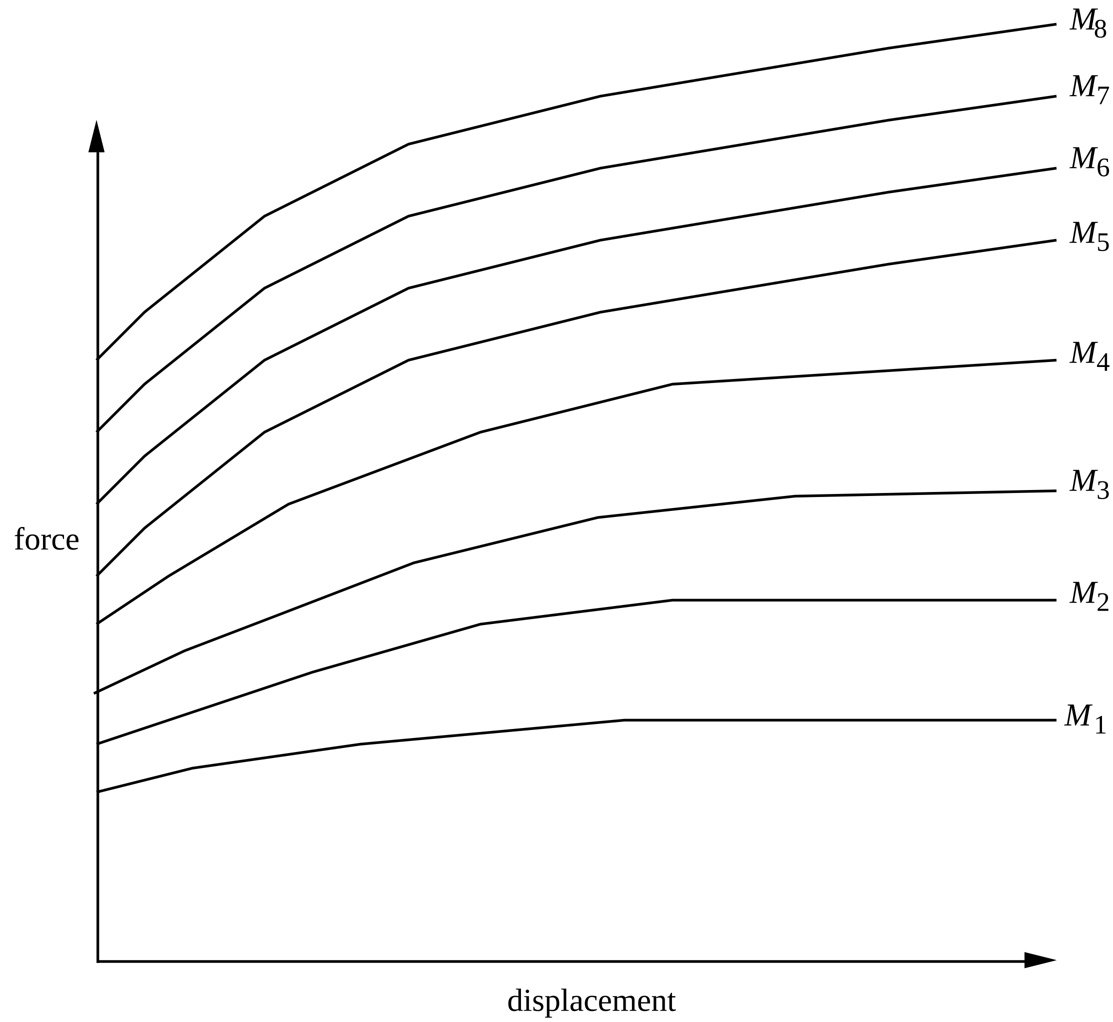


Figure 3.14. The force magnitude is limited by the applied end moment. For an intermediate value of the end moment LS-DYNA interpolates between the curves to determine the allowable force value.

Material Type 30 (Closed-Form Update Shell Plasticity)

This model implements a closed form solution for the plane stress constitutive update under conditions of perfect plasticity or kinematic hardening for a bilinear von Mises model. The implementation is described in [Whirley et al. 1989].

Columns	Quantity		Format
1-10	Card 3	Young’s modulus	E10.0
1-10	Card 4	Poisson’s ratio	E10.0
1-10	Card 5	Yield stress	E10.0
1-10	Card 6	Hardening modulus, E_t	E10.0
	Card 7	Blank	
	Card 8	Blank	

Material Type 31 (Slightly Compressible Rubber Model)

This model implements a modified form of the hyperelastic constitutive law first described in [Kenchington 1988].

Columns		Quantity	Format
1-10	Card 3	Poisson’s ratio. Values between .49 and less than .50 are suggested.	E10.0

The constants can be defined directly or a least squares fit can be performed if the uniaxial data is available. If a least squares fit is chosen, then flag the terms to be included in the energy functional by setting their corresponding coefficients to unity. If all coefficients are zero the default is to use only the terms involving I_1 and I_2. C_{100} defaults to unity if the least square fit is used.

Columns		Quantity	Format
11-20		C_{100} (EQ.1.0 if term is included in the least squares fit.)	E10.0
21-30		C_{200} (EQ.1.0 if term is included in the least squares fit.)	E10.0
31-40		C_{300} (EQ.1.0 if term is included in the least squares fit.)	E10.0
41-50		C_{400} (EQ.1.0 if term is included in the least squares fit.)	E10.0
1-10	Card 4	C_{110} (EQ.1.0 if term is included in the least squares fit.)	E10.0
11-20		C_{210} (EQ.1.0 if term is included in the least squares fit.)	E10.0
21-30		C_{010} (EQ.1.0 if term is included in the least squares fit.)	E10.0
31-40		C_{020} (EQ.1.0 if term is included in the least squares fit.)	E10.0
1-10	Card 5	Exit or continue option EQ.0.0: continue if strain limits are exceeded EQ.1.0: stop if strain limits are exceeded	E10.0
1-10	Card 6	Maximum strain limit, (Green-St. Venant Strain)	E10.0
11-20		Minimum strain limit, (Green-St. Venant Strain)	E10.0

If Card 7 is defined then a least squares fit is performed to determine the constants which are input as 1.0.

Material Type 31 (Slightly Compressible Rubber Model)

Columns		Quantity	Format
1-10	Card 7	Specimen gauge length	E10.0
11-20		Specimen width	E10.0
21-30		Specimen thickness	E10.0
31-40		Load curve ID giving the force versus actual change in the gauge length.	E10.0
1-10	Card 8	Use reference geometry to initial stresses. EQ.0.0: off EQ.1.0: on	E10.0

The strain energy functional, *U*, is defined in terms of the input constants as:

$$
\begin{aligned}
U = {} & C_{100}\, I_1 + C_{200}\, I_1^{\,2} + C_{300}\, I_1^{\,3} + C_{400}\, I_1^{\,4} + C_{110} I_1 I_2 + \\
& C_{210} I_1^{\,2} I_2 + C_{010}\, I_2 + C_{020} I_2^{\,2} \; + f(J)
\end{aligned}
$$

where the invarients can be expressed in terms of the deformation gradient matrix, F_{ij}, and the Green-St. Venant strain tensor, E_{ij} :

$$
\begin{aligned}
J \; &= \left| F_{ij} \right| \\
I_1 &= E_{ii} \\
I_2 &= \frac{1}{2!} \delta^{ij}_{pq} E_{pi} E_{qj}
\end{aligned}
$$

The derivative of U with respect to a component of strain gives the corresponding component of stress

$$
S_{ij} = \frac{\partial U}{\partial E_{ij}}
$$

here, S_{ij}, is the second Piola-Kirchhoff stress tensor.

The load curve definition that provides the uniaxial data should give the change in gauge length, ΔL, in columns 1-10 and the corresponding force in column s 11-20 if a 2E10.0 format is used. In compression both the force and the change in gauge length must be specified as negative values. In tension the force and change in gauge length should be input as positive values. The principal stretch ratio in the uniaxial direction, λ_1, is then given by

$$\lambda_1 = \frac{L_O + \Delta L}{L_O} \quad .$$

Alternatively, the stress versus strain curve can also be input by setting the gauge length, thickness, and width to unity and defining the engineering strain in place of the change in gauge length and the nominal (engineering) stress in place of the force. See Figure 3.13.

The least square fit to the experimental data is performed during the initialization phase and is a comparison between the fit and the actual input is provided in the printed file. It is a good idea to visually check the fit to make sure it is acceptable. The coefficients C_{100} - C_{020} are also printed in the output file.

Material Type 32 (Laminated Glass Model)

Columns		Quantity	Format
1-10	Card 3	Young's modulus (glass)	E10.0
11-20		Poisson's ratio (glass)	E10.0
21-30		Yield stress (glass)	E10.0
31-40		Hardening modulus, E_t (glass)	E10.0
41-50		Plastic strain at failure (glass)	E10.0
1-10	Card 4	Young's modulus (polymer)	E10.0
11-20		Poisson's ratio (polymer)	E10.0
21-30		Yield stress (polymer)	E10.0
31-40		Hardening modulus, E_t (polymer)	E10.0
1-10	Card 5	1st integration point material (glass=0, polymer=1)	E10.0
11-20		2nd integration point material	E10.0
21-30		3rd integration point material	E10.0
31-40		4th integration point material	E10.0

Isotropic hardening is assumed. The material to which the glass is bonded is assumed to stretch plastically without failure. A user defined integration rule specifies the thickness of the layers making up the glass. On cards 5-8, columns 1-80 (*E10.0) define whether the integration point is glass (0.0) or polymer (1.0). Define the material for the same number of integration points as specified in the rule. Insert blank cards as necessary.

Material Type 33 (Barlat's Anisotropic Plasticity Model)

This model was developed by Barlat, Lege, and Brem [1991] for modelling material behavior in forming processes. The finite element implementation of this model is described in detail by Chung and Shah [1992] and is used here.

Columns		Quantity	Format
1-10	Card 3	Young's modulus, E	E10.0
11-20		Poisson's ratio, ν	E10.0
21-30		k	E10.0
31-40		ε_0	E10.0
41-50		n	E10.0
51-60		m, flow potential exponent in Barlat's Model	E10.0
1-10	Card 4	a, anisotropy coefficient in Barlat's Model	E10.0
11-20		b, anisotropy coefficient in Barlat's Model	E10.0
21-30		c anisotropy coefficient in Barlat's Model	E10.0
31-40		f, anisotropy coefficient in Barlat's Model	E10.0
41-50		g, anisotropy coefficient in Barlat's Model	E10.0
51-60		h, anisotropy coefficient in Barlat's Model	E10.0
1-10	Card 5	Blank	E10.0
1-10	Card 6	Material axes option, AOPT EQ.0.0: locally orthotropic with material axes determined by element nodes as shown in Figure 3.5. Cards 7 and 8 are blank with this option. EQ.1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center. Card 8 below is blank. EQ.2.0: globally orthotropic with material axes determined by vectors defined on Cards 7 and 8	E10.0

Columns		Quantity	Format
		EQ.3.0: applicable to shell elements only. This option determines locally orthotropic material axes by offsetting the material axes by an angle (Card 8) from a line in the plane of the shell determined by taking the cross product of the vector defined on Card 7 with the shell normal vector.	
1-10	Card 7	x_p, define for AOPT = 1.0	E10.0
11-20		y_p, define for AOPT = 1.0	E10.0
21-30		z_p, define for AOPT = 1.0	E10.0
1-10	Card 7	a_1, define for AOPT = 2.0	E10.0
11-20		a_2, define for AOPT = 2.0	E10.0
21-30		a_3, define for AOPT = 2.0	E10.0
1-10	Card 7	v_1, define for AOPT = 3.0	E10.0
11-20		v_2, define for AOPT = 3.0	E10.0
21-30		v_3, define for AOPT = 3.0	E10.0
1-10	Card 8	d_1, define for AOPT = 2.0	E10.0
11-20		d_2, define for AOPT = 2.0	E10.0
21-30		d_3, define for AOPT = 2.0	E10.0

The yield function Φ is defined as

$$\Phi = |S_1 - S_2|^m + |S_2 - S_3|^m + |S_3 - S_1|^m = 2^m$$

where $\bar{\sigma}$ is the effective stress and $S_{i=1,2,3}$ are the principal values of the symmetric matrix $S_{\alpha\beta}$,

$$S_{xx} = [c(\sigma_{xx} - \sigma_{yy}) - b(\sigma_{zz} - \sigma_{xx})]/3$$
$$S_{yy} = [a(\sigma_{yy} - \sigma_{zz}) - c(\sigma_{xx} - \sigma_{yy})]/3$$
$$S_{zz} = [b(\sigma_{zz} - \sigma_{xx}) - a(\sigma_{yy} - \sigma_{zz})]/3$$
$$S_{yz} = f\sigma_{yz}$$
$$S_{zx} = g\sigma_{zx}$$
$$S_{xy} = h\sigma_{xy}$$

The material constants a, b, c, f, g and h represent anisotropic properties. When $a = b = c = f = g = h = 1$, the material is isotropic and the yield surface reduces to the Tresca yield surface for m=1 and von Mises yield surface for $m = 2 or 4$.

For FCC materials m=8 is recommended and for BCC materials m=6 is used.
The yield strength of the material is

$$\sigma_y = k(1 + \varepsilon_0)^n \quad .$$

Material Type 34 (Fabric)

The fabric model is a variation on the layered orthotropic composite model of material 22 and is valid for 3 and 4 node membrane elements only and is strongly recommended for modeling airbags and seatbelts. In addition to being a constitutive model, this model also invokes a special membrane element formulation which is more suited to the deformation experienced by fabrics under large deformation. For thin fabrics, buckling can result in an inability to support compressive stresses; thus a flag is included for this option. A linear elastic liner is also included which can be used to reduce the tendency for these elements to be crushed when the no-compression option is invoked.

Columns		Quantity	Format
1-10	Card 3	E_a , longitudinal direction	E10.0
11-20		E_b, transverse direction EQ.0.0: Isotropic elastic (Version 950 onward)	E10.0
21-30		E_c, normal direction EQ.0.0: Isotropic elastic (Version 950 onward)	E10.0
31-40		Fabric leakage coefficient (optional), FLC LT.0.0: |FLC| is the load curve ID of the curve defining FLC versus time. See notes below.	E10.0
41-50		Fabric area coefficient (optional), FAC LT.0.0: |FAC| is the load curve ID of the curve defining FAC versus <u>absolute</u> pressure. See notes below.	E10.0
51-60		Effective leakage area for blocked fabric, ELA LT.0.0: |ELA| is the load curve ID of the curve defining ELA versus time. The default value of zero assumes that no leakage occurs. A value of .10 would assume that 10% of the blocked fabric is leaking gas.	E10.0
61-70		Flag to turn off compression in liner until the reference geometry is reached, i.e., the fabric element becomes tensile. EQ.0.0: off. EQ.1.0: on.	E10.0

Columns		Quantity	Format
1-10	Card 4	υ_{ba}, Poisson’s ratio	E10.0
11-20		υ_{ca} EQ.0.0: Isotropic elastic (Version 950 onward)	E10.0
21-30		υ_{cb} EQ.0.0: Isotropic elastic (Version 950 onward)	E10.0
31-40		Compressive stress flag (default = 0.0). This option does not apply to the liner. EQ.0.0: don’t eliminate compressive stresses EQ.1.0: eliminate compressive stresses	E10.0
41-50		Flag to modify membrane formulation for fabric material: EQ.0.0:default. Least costly and very reliable. EQ.1.0:invarient local membrane coordinate system EQ.2.0:Green-Largrange strain formulation (available for testing but not yet recommended.)	
1-10	Card 5	G_{ab}	E10.0
11-20		G_{bc}	E10.0
21-30		G_{ca}	E10.0
31-40		Young's modulus for elastic liner (optional)	E10.0
41-50		Poisson’s ratio for elastic liner (optional)	E10.0
51-60		Ratio of liner thickness to total fabric thickness	E10.0
61-70		Rayleigh damping coefficient. A value of 0.05 generally works well. This roughly corresponds to 5 percent structural damping.	E10.0

For an elastic isotropic fabric cards 6-8 below can be left blank. For orthotropic behavior an accurate definition of the material directions below is very important in obtaining correct.results.

Columns		Quantity	Format
1-10	Card 6	Material axes option, AOPT EQ.0.0: locally orthotropic with material axes determined by element nodes n_1, n_2, and n_4 as shown in Figure 3.5. Cards 7 and 8 below are blank with this option.	E10.0

Columns		Quantity	Format
		EQ.1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center. Card 8 below is blank. EQ.2.0: globally orthotropic with material axes determined by vectors defined on Cards 7 and 8. EQ.3.0: applicable to shell elements only. This option determines locally orthotropic material axes by offsetting the material axes by an angle (Card 12) from a line in the plane of the shell determined by taking the cross product of the vector defined on Card 7 with the shell normal vector.	
1-10	Card 7	x_p define for AOPT=1.0	E10.0
11-20		y_p define for AOPT=1.0	E10.0
21-30		z_p define for AOPT=1.0	E10.0
1-10	Card 7	a_1 define for AOPT=2.0	E10.0
11-20		a_2 define for AOPT=2.0	E10.0
21-30		a_3 define for AOPT=2.0	E10.0
1-10	Card 7	v_1 define for AOPT=3.0	E10.0
11-20		v_2 define for AOPT=3.0	E10.0
21-30		v_3 define for AOPT=3.0	E10.0
1-10	Card 8	d_1 define for AOPT=2.0	E10.0
11-20		d_2 define for AOPT=2.0	E10.0
21-30		d_3 define for AOPT=2.0	E10.0

If the airbag material is to be approximated as an isotropic material, then only one Young's modulus and Poisson's ratio should be defined. The elastic approximation is very cost efficient due to the fact that the local transformations to the material coordinate system may be skipped. If orthotropic constants are defined, it is very important to consider the orientation of the local material system and employ great care in setting up the finite element mesh.

The parameters FLC, FAC, and ELA are optional for the Wang-Nefske and hybrid inflation models. It is possible for the airbag to be constructed of multiple fabrics having different values for porosity and permeability. The leakage of gas through the fabric in an airbag then requires an accurate determination of the areas by part ID available for leakage. The leakage area may change over time due to stretching of the airbag fabric or blockage when the bag contacts the structure. LS-DYNA can check the interaction of the bag with the structure and split the areas into regions that are blocked and unblocked depending on whether the regions are in or not in contact, respectively. Typically, FLC and FAC must be determined experimentally and there variation in time with pressure are optional to allow for maximum flexibility.

Material Type 35 (Kinematic/Isotropic Elastic-Plastic Green-Naghdi Rate)

Columns		Quantity	Format
1-10	Card 3	Young's modulus	E10.0
11-20		Strain rate parameter, C	E10.0
21-30		Strain rate parameter, p	E10.0
1-10	Card 4	Poisson's ratio	E10.0
1-10	Card 5	Yield stress	E10.0
1-10	Card 6	Hardening modulus, E_t	E10.0
1-10	Card 7	Hardening parameter, β'	E10.0
		$0 < \beta' < 1$	
	Card 8	Blank	

This model is available only for brick elements and is similar to model 3 but uses the Green-Naghdi Rate formulation rather than the Jaumann rate.

Material Type 36 (Barlat's 3-Parameter Plasticity Model)

This model was developed by Barlat and Lian [1989] for modelling sheets under plane stress conditions.

Columns		Quantity	Format
1-10	Card 3	Young's modulus, E	E10.0
11-20		Poisson's ratio, ν	E10.0
21-30		Hardening rule EQ.1.0: linear EQ.2.0: exponential EQ.3.0: load curve	E10.0
For linear hardening (1.0) in columns 31-50 define:			
31-40		Tangent modulus	E10.0
41-50		Yield stress	E10.0
For exponential hardening (2.0) in columns 31-50 define:			
31-40		k, strength coefficient for exponential hardening	E10.0
41-50		n, exponent	E10.0
51-60		load curve ID for the load curve hardening rule	E10.0
61-70		ε_0 for determining initial yield stress for exponential hardening. (Default=0.0)	E10.0
71-80		spi, if ε_0 is zero above. (Default=0.0) EQ.0.0: $\varepsilon_0 = (E/k)**[1/(n-1)]$ LE..02: $\varepsilon_0 = spi$ GT..02: $\varepsilon_0 = (spi/k)**[1/n]$	E10.0
1-10	Card 4	m, exponent in Barlat's yield surface	E10.0
11-20		R_{00}	E10.0
21-30		R_{45}	E10.0
31-40		R_{90}	E10.0
1-10	Card 5	Blank	E10.0

Columns		Quantity	Format
1-10	Card 6	Material axes option, AOPT EQ.0.0: locally orthotropic with material axes determined by element nodes as shown in Figure 3.5. Cards 7 and 8 are blank with this option. EQ.1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center. Card 8 below is blank. EQ.2.0: globally orthotropic with material axes determined by vectors defined on Cards 7 and 8 EQ.3.0: applicable to shell elements only. This option determines locally orthotropic material axes by offsetting the material axes by an angle (Card 8) from a line in the plane of the shell determined by taking the cross product of the vector defined on Card 7 with the shell normal vector.	E10.0
1-10	Card 7	x_p, define for AOPT = 1.0	E10.0
11-20		y_p, define for AOPT = 1.0	E10.0
21-30		z_p, define for AOPT = 1.0	E10.0
1-10	Card 7	a_1, define for AOPT = 2.0	E10.0
11-20		a_2, define for AOPT = 2.0	E10.0
21-30		a_3, define for AOPT = 2.0	E10.0
1-10	Card 7	v_1, define for AOPT = 3.0	E10.0
11-20		v_2, define for AOPT = 3.0	E10.0
21-30		v_3, define for AOPT = 3.0	E10.0
1-10	Card 8	d_1, define for AOPT = 2.0	E10.0
11-20		d_2, define for AOPT = 2.0	E10.0
21-30		d_3, define for AOPT = 2.0	E10.0

The anisotopic yield criterion Φ for plane stress is defined as:

$$\Phi = a\left|K_1 + K_2\right|^m + a\left|K_1 - K_2\right|^m + c\left|2K_2\right|^m = 2\sigma_Y^m$$

where σ_Y is the yield stress and $K_{i=1,2}$ are given by:

$$K_1 = \frac{\sigma_x - h\sigma_y}{2}$$

$$K_2 = \sqrt{\left(\frac{\sigma_x - h\sigma_y}{2}\right)^2 + p^2\tau_{xy}^2}$$

The anisotropic material constants a, c, h, and p are obtained through R_{00}, R_{45}, and R_{90}:

$$a = 2 - 2\sqrt{\frac{R_{00}}{1+R_{00}}\frac{R_{90}}{1+R_{90}}}$$

$$c = 2 - a$$

$$h = \sqrt{\frac{R_{00}}{1+R_{00}}\frac{1+R_{90}}{R_{90}}}$$

The anisotropy parameter p is calculated implicitly. According to Barlat and Lian the R value, width to thickness strain ratio, for any angle ϕ can be calculated from:

$$R_\phi = \frac{2m\sigma_Y^m}{\left(\frac{\partial\Phi}{\partial\sigma_x} + \frac{\partial\Phi}{\partial\sigma_y}\right)\sigma_\phi} - 1$$

where σ_ϕ is the uniaxial tension in the ϕ direction. This expression can be used to iteratively calculate the value of p. Let ϕ=45 and define a function g as

$$g(p) = \frac{2m\sigma_Y^m}{\left(\frac{\partial\Phi}{\partial\sigma_x} + \frac{\partial\Phi}{\partial\sigma_y}\right)\sigma_\phi} - 1 - R_{45}$$

An iterative search is used to find the value of p.

For FCC materials m=8 is recommended and for BCC materials m=6 may be used. The yield strength of the material can be expressed in terms of k and n:

$$\sigma_y = k\,\varepsilon^n = k\left(\varepsilon_{yp} + \bar{\varepsilon}^p\right)^n$$

where ε_{yp} is the elastic strain to yield and $\bar{\varepsilon}^p$is the effective plastic strain (logrithmic). If SIGY is set to zero, the strain to yield if found by solving for the intersection of the linearly elastic loading equation with the strain hardening equation:

$$\sigma = E\,\varepsilon$$
$$\sigma = k\,\varepsilon^n$$

which gives the elastic strain at yield as:

$$\varepsilon_{yp} = \left(\frac{E}{k}\right)^{\left[\frac{1}{n-1}\right]}$$

If SIGY yield is nonzero and greater than 0.02 then:

$$\varepsilon_{yp} = \left(\frac{\sigma_y}{k}\right)^{\left[\frac{1}{n}\right]}$$

Material Type 37 (Transversely Anisotropic Elastic-Plastic)

Columns		Quantity	Format
1-10	Card 3	Young's modulus	E10.0
1-10	Card 4	Poisson's ratio	E10.0
1-10	Card 5	Yield stress	E10.0
1-10	Card 6	Hardening modulus, E_t	E10.0
1-10	Card 7	Anisotropic hardening parameter, R	E10.0
1-10	Card 8	Load curve number defining effective stress versus effective plastic strain. The yield stress and hardening modulus are ignored with this option.	E10.0

This plasticity model is fully iterative and is available only for shell elements.

Consider Cartesian reference axes which are parallel to the three symmetry planes of anisotropic behavior. Then, the yield function suggested by [Hill 1948] can be written

$$F(\sigma_{22}-\sigma_{33})^2 + G(\sigma_{33}-\sigma_{11})^2 + H(\sigma_{11}-\sigma_{22})^2 + 2L\sigma_{23}^2 + 2M\sigma_{31}^2 + 2N\sigma_{12}^2 - 1 = 0$$

where σ_{y1}, σ_{y2}, and σ_{y3}, are the tensile yield stresses and σ_{y12}, σ_{y23}, and σ_{y31} are the shear yield stresses. The constants *F*, *G*, *H*, *L*, *M*, and *N* are related to the yield stress by

$$2L = \frac{1}{\sigma_{23}^2}$$

$$2M = \frac{1}{\sigma_{y31}^2}$$

$$2N = \frac{1}{\sigma_{y12}^2}$$

$$2F = \frac{1}{\sigma_{y2}^2} + \frac{1}{\sigma_{y3}^2} - \frac{1}{\sigma_{y1}^2}$$

$$2G = \frac{1}{\sigma_{y3}^2} + \frac{1}{\sigma_{y1}^2} - \frac{1}{\sigma_{y2}^2}$$

$$2H = \frac{1}{\sigma_{y1}^2} + \frac{1}{\sigma_{y2}^2} - \frac{1}{\sigma_{y3}^2}\ .$$

The isotropic case of von Mises plasticity can be recovered by setting $F = G = H = \ {}^{1}\!/_{2\sigma_y^2}$ and $L = M = N = {}^{3}\!/_{2\sigma_y^2}$.

For the particular case of transverse anisotropy, where properties do not vary in the x_1-x_2 plane, the following relations hold:

$$2F = 2G = \frac{1}{\sigma_{y3}^2}$$

$$2H = \frac{2}{\sigma_y^2} - \frac{1}{\sigma_{y3}^2}$$

$$N = \frac{2}{\sigma_y^2} - \frac{1}{2}\frac{1}{\sigma_{y3}^2}$$

where it has been assumed that $\sigma_{y1} = \sigma_{y2} = \sigma_y$.

Letting $K = {}^{\sigma_y}\!/_{\sigma_{y3}}$, the yield criteria can be written

$$F(\sigma) = \sigma_e = \sigma_y,$$

where

$$F(\sigma) \equiv \Big[\sigma_{11}^2 + \sigma_{22}^2 + K^2\sigma_{33}^2 - K^2\sigma_{33}\left(\sigma_{11}+\sigma_{22}\right) - \left(2-K^2\right)\sigma_{11}\sigma_{22}$$
$$+ 2L\sigma_y^2\left(\sigma_{23}^2+\sigma_{31}^2\right) + 2\left(2-\frac{1}{2}K^2\right)\sigma_{12}^2\Big]^{1/2}$$

The rate of plastic strain is assumed to be normal to the yield surface so $\dot{\varepsilon}_{ij}^{p}$ is found from

$$\dot{\varepsilon}_{ij}^{p} = \lambda\frac{\partial F}{\partial \sigma_{ij}}\ .$$

Now consider the case of plane stress, where $\sigma_{33} = 0$. Also, define the anisotropy input parameter (see Card 7 above) R as the ratio of the in-plane plastic strain rate to the out-of-plane plastic strain rate,

$$R = \frac{\dot{\varepsilon}_{22}^{p}}{\dot{\varepsilon}_{33}^{p}} .$$

It then follows that

$$R = \frac{2}{K^2} - 1$$

Using the plane stress assumption and the definition of R, the yield function may now be written

$$F(\sigma) = \left[\sigma_{11}^2 + \sigma_{22}^2 - \frac{2R}{R+1} \sigma_{11}\sigma_{22} + 2\frac{2R+1}{R+1} \sigma_{12}^2 \right]^{½} .$$

Material Type 38 (Blatz-Ko Compressible Foam)

Columns		Quantity	Format
1-10	Card 3	Shear modulus, *G*	E10.0
11-20		Use reference geometry to initial stresses. EQ.0.0: off EQ.1.0: on	E10.0
	Card 4	Blank	
.		.	.
.		.	.
.		.	.
	Card 8	Blank	

The strain energy functional for the compressible foam model is given by

$$W = \frac{G}{2}\left(\tfrac{\mathrm{II}}{\mathrm{III}} + 2\sqrt{\mathrm{III}} - 5\right)$$

Blatz and Ko [1962] suggested this form for a 47 percent volume polyurethane foam rubber with a Poisson's ratio of 0.25. In terms of the strain invarients, I, II, and III, the second Piola-Kirchhoff stresses are given as

$$S^{ij} = G\left[\left(\mathrm{I}\delta_{ij} - C_{ij}\right)\frac{1}{\mathrm{III}} + \left(\sqrt{\mathrm{III}} - \frac{\mathrm{II}}{\mathrm{III}}\right)C_{ij}^{-1}\right]$$

where C_{ij} is the right Cauchy-Green strain tensor. This stress measure is transformed to the Cauchy stress, σ_{ij}, according to the relationship

$$\sigma^{ij} = \mathrm{III}^{-1/2} F_{ik} F_{jl} S_{lk}$$

where F_{ij} is the deformation gradient tensor.

Material Type 39 (Transversely Anisotropic Elastic-Plastic with FLD)

Columns		Quantity	Format
1-10	Card 3	Young's modulus	E10.0
11-20	Card 3	Curve ID defining the Flow Limit Diagram. Minor strains in percent are defined as abcissa values Major strains in percent are defined as ordinate values The flow limit diagram is shown in Figure 3.15. In defining the curve, list pairs of minor and major strains starting with the left most point and ending with right most point.	E10.0
1-10	Card 4	Poisson's ratio	E10.0
1-10	Card 5	Yield stress	E10.0
1-10	Card 6	Hardening modulus, E_t	E10.0
1-10	Card 7	Anisotropic hardening parameter, R	E10.0
1-10	Card 8	Load curve ID defining effective stress versus effective plastic strain. The yield stress and hardening modulus are ignored with this option.	E10.0

See material model 37 for the theoretical basis. The first history variable is the maximum strain ratio defined by:

$$\frac{\varepsilon_{major_{workpiece}}}{\varepsilon_{major_{fld}}}$$

corresponding to $\varepsilon_{minor_{workpiece}}$.

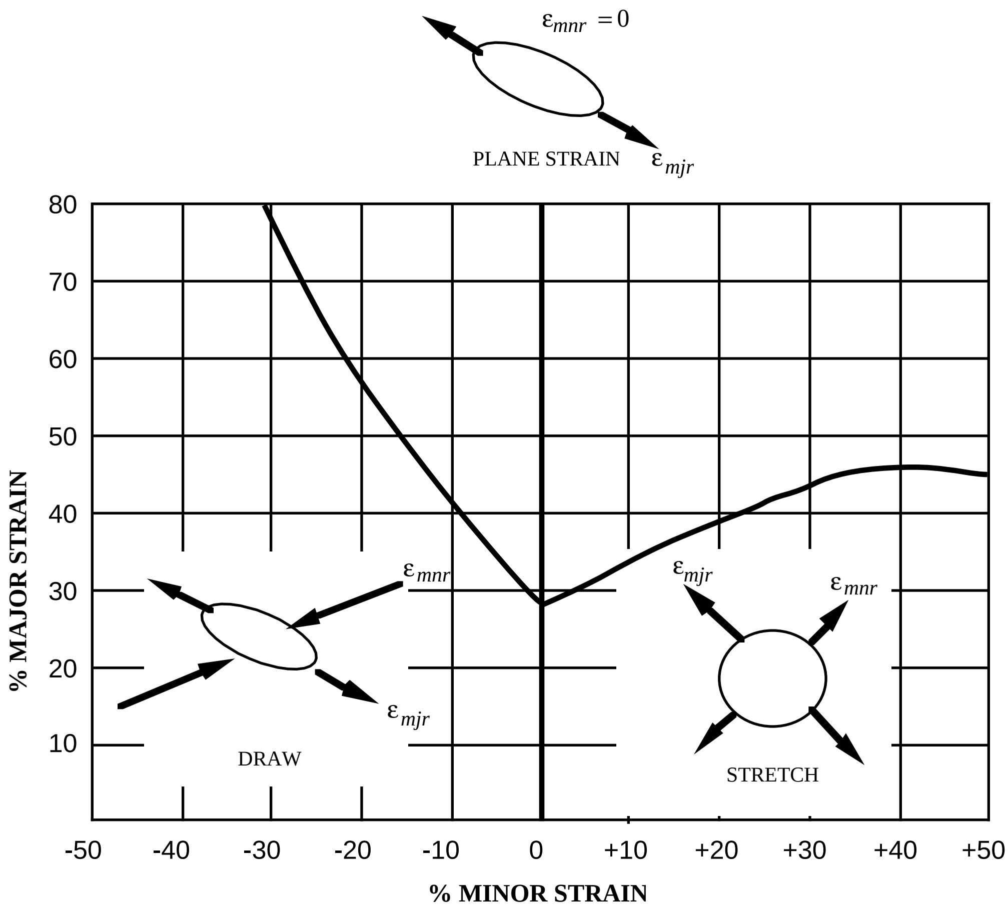


Figure 3.15. Flow limit diagram.

Material Type 40 (Nonlinear Elastic Orthotropic Material)

Columns		Quantity	Format
1-10	Card 3	E_{a0}, modulus-longitudinal direction	E10.0
11-20		E_{b0}, modulus-transverse direction	E10.0
21-30		E_{c0}, modulus-normal direction	E10.0
	The following input on Card 1 is optional and applies to solid elements only.		
31-40		Load curve ID defining the nominal stress versus strain along c-axis. Strain is defined as λ_c-1 where λ_c is the stretch ratio along the c axis.	E10.0
41-50		Load curve ID defining the nominal ab shear stress versus ab-strain in the ab-plane. Strain is defined as the sin(γ_{ab}) where γ_{ab} is the shear angle.	E10.0
51-60		Load curve ID defining the nominal ab shear stress versus ab-strain in the bc-plane. Strain is defined as the sin(γ_{bc}) where γ_{bc} is the shear angle.	E10.0
61-70		Load curve ID defining the nominal ab shear stress versus ab-strain in the ca-plane. Strain is defined as the sin(γ_{ca}) where γ_{ca} is the shear angle.	E10.0
1-10	Card 4	ν_{ba}	E10.0
11-20		ν_{ca}	E10.0
21-30		ν_{cb}	E10.0
31-40		ΔT, temperature increment for stress initialization.	E10.0
41-50		T_{ramp}, time to ramp up to the final temperature	E10.0
51-60		α, thermal expansion coefficient	E10.0
1-10	Card 5	G_{ab}	E10.0
11-20		G_{bc}	E10.0
21-30		G_{ca}	E10.0

Material Type 40 (Nonlinear Elastic Orthotropic Material)

Columns		Quantity	Format
1-10	Card 6	Material axes option, AOPT EQ.0.0: Locally orthotropic with material axes determined by element nodes n_1, n_2, and n_4, as shown in Figure 3.5. The AOPT data on cards 7 and 8 below are blank with this option. EQ.1.0: Locally orthotropic with material axes determined by a point in space and the global location of the element center. The AOPT data on card 8 below is blank. EQ.2.0: Globally orthotropic with material axes determined by vectors defined on Card 7 and 8. EQ.3.0: Applicable to shell elements only. This option determines locally orthotropic material axes by offsetting the material axes by an angle (Card 12) from a line in the plane of the shell determined by taking the cross product of the vector defined on Card 7 with the shell normal vector.	E10.0
1-10	Card 7	x_p define for AOPT=1.0	E10.0
11-20		y_p define for AOPT=1.0	E10.0
21-30		z_p define for AOPT=1.0	E10.0
1-10	Card 7	a_1 define for AOPT=2.0	E10.0
11-20		a_2 define for AOPT=2.0	E10.0
21-30		a_3 define for AOPT=2.0	E10.0
1-10	Card 7	v_1 define for AOPT=3.0	E10.0
11-20		v_2 define for AOPT=3.0	E10.0
21-30		v_3 define for AOPT=3.0	E10.0
1-10	Card 8	d_1 define for AOPT=2.0	E10.0
11-20		d_2 define for AOPT=2.0	E10.0
21-30		d_3 define for AOPT=2.0	E10.0
31-40		Load curve ID defining the nominal stress versus strain along a-axis. Strain is defined as λ_a-1 where λ_a is the stretch ratio along the a axis.	E10.0

Columns	Quantity	Format
41-50	Load curve ID defining the nominal stress versus strain along b-axis. Strain is defined as λ_b-1 where λ_b is the stretch ratio along the b axis.	E10.0
51-60	ε_{fail}, failure strain	E10.0
61-70	Δt_{fail}, time step for automatic element erosion	E10.0
71-80	C_{damp}, damping coefficient	E10.0

Material Types 41-50 (User Defined Material Models)

Define the number of fields specified in the control section using only as many cards as needed with eight parameters per card with format 8E10.0. The locations of the bulk modulus, shear modulus, and the orientation set must be consistent with the control card defined for the material type.

When the user defined material requires the deformation gradient, nine extra history variables must be allocated for its storage and a call to compute_f must be made as described in Appendix A.

Columns	Quantity	Format
1-10	First material parameter	E10.0
11-20	Second material parameter	E10.0
21-30	Third material parameter	E10.0
31-40	Fourth material parameter	E10.0
.	.	.
.	.	.
.	.	.

The orientation information begins at the specified address and contains the following information.in the order given.below.

1.	Material axes option, AOPT EQ.0.0: locally orthotropic with material axes determined by element nodes n_1, n_2, and n_4 as shown in Figure 3.5. Fields 3-8 below are blank with this option. EQ.1.0: locally orthotropic with material axes determine by a point in space and the global location of the element center. Fields 6-8 below are not defined. EQ.2.0: globally orthotropic with material axes determined by vectors defined in fields 3-8 below. EQ.3.0: applicable to shell elements only. This option determines locally orthotropic material axes by offsetting the material axes by an angle from a line in the plane of the shell determined by taking the cross product of the vector defined in fields 3-5 below with the shell normal vector.	E10.0
2.	Material axes change flag for brick elements	E10.0

EQ.1.0: default
EQ.2.0: switch material axes a and b
EQ.3.0: switch material axes a and c

3. x_p define for AOPT=1.0
a_1 define for AOPT=2.0
v_1 define for AOPT=3.0

4. y_p define for AOPT=1.0
a_2 define for AOPT=2.0
v_2 define for AOPT=3.0

5. z_p define for AOPT=1.0
a_3 define for AOPT=2.0
v_3 define for AOPT=3.0

6. d_1 define for AOPT=2.0

7. d_2 define for AOPT=2.0

8. d_3 define for AOPT=2.0

Material Type 42 (Planar Anisotropic Plasticity Model)

This model is built into LS-DYNA as a user material model for modeling plane stress anisotropic plasticity in shells. Please note that only three cards are input here. The orthotropic angles must be defined later as for all materials of this type. This model is currently not vectorized.

Columns		Quantity	Format
1-10	Card 3	E, Young's modulus	E10.0
11-20		PR, Poisson's ratio	E10.0
21-30		K, bulk modulus	E10.0
31-40		G, shear modulus	E10.0
41-50		Yield parameter, A	E10.0
51-60		Plastic hardening coefficient, m	E10.0
61-70		Strain rate coefficient, n	E10.0
71-80		Optional load curve ID defining yield stress versus ε_{eff}^{p}	E10.0
1-10	Card 4	R00	E10.0
11-20		R45	E10.0
21-30		R90	E10.0
31-40		$\dot{\varepsilon}_{min}$ minimum strain rate (must be nonzero)	E10.0
1-10	Card 5	Material axes option, AOPT EQ.0.0: locally orthotropic with material axes determined by element nodes n_1, n_2, and n_4 as shown in Figure 3.5. Cards 7 and 8 below are blank with this option. EQ.1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center. Card 8 below is blank. EQ.2.0: globally orthotropic with material axes determined by vectors defined on Cards 7 and 8.	E10.0

Columns	Quantity	Format
	EQ.3.0: applicable to shell elements only. This option determines locally orthotropic material axes by offsetting the material axes by an angle (Card 12) from a line in the plane of the shell determined by taking the cross product of the vector defined on Card 7 with the shell normal vector.	
11-20.	Material axes change flag for brick elements EQ.1.0: default EQ.2.0: switch material axes a and b EQ.3.0: switch material axes a and c	E10.0
21-30	x_p define for AOPT=1.0	E10.0
31-40	y_p define for AOPT=1.0	E10.0
41-50	z_p define for AOPT=1.0	E10.0
21-30	a_1 define for AOPT=2.0	E10.0
31-40	a_2 define for AOPT=2.0	E10.0
41-50	a_3 define for AOPT=2.0	E10.0
21-30	v_1 define for AOPT=3.0	E10.0
31-40	v_2 define for AOPT=3.0	E10.0
41-50	v_3 define for AOPT=3.0	E10.0
51-60	d_1 define for AOPT=2.0	E10.0
61-70	d_2 define for AOPT=2.0	E10.0
71-80	d_3 define for AOPT=2.0	E10.0

This is an implementation of an anisotropic plasticity model for plane stress where the flow rule, see Material Type 37, simplifies to:

$$F\left(\sigma_{22}\right)^2 + G\left(\sigma_{11}\right)^2 + H\left(\sigma_{11} - \sigma_{22}\right)^2 + 2N\sigma_{12}^2 - 1 = 0 \ .$$

The anisotropic parameters R00, R45, and R90 are defined in terms of *F*, *G*, *H*, and *N* as [Hill, 1989]:

$$2R00 = \frac{H}{G}$$

$$2R45 = \frac{2N}{(F+G)} - 1.$$

$$2R90 = \frac{H}{F}$$

The yield function for this model is given as:

$$\sigma_y = A\varepsilon^m \dot{\varepsilon}^n$$

To avoid numerical problems the minimum strain rate, $\dot{\varepsilon}_{\min}$ must be defined and the initial yield stress σ_0, is calculated as

$$\sigma_0 = A\varepsilon_0{}^m \dot{\varepsilon}_{\min}^n = E\varepsilon_0$$

$$\varepsilon_0 = \left(\frac{E}{A\dot{\varepsilon}_{\min}^n} \right).$$

Material Type 48 (Strain Rate Dependent Plasticity with Size Dependent Failure)

Columns	Quantity	Format
1-10	Bulk modulus, K	E10.0
11-20	c1, strain rate parameter (see equation 3.3)	E10.0
21-30	p1, strain rate parameter (see equation 3.3)	E10.0
31-40	c2, strain rate parameter (see equation 3.4)	E10.0
41-50	p2, strain rate parameter (see equation 3.4)	E10.0

Card 4 (8E10.0)

Columns	Quantity	Format
1-10	Shear modulus, G	E10.0
11-20	Curve number giving failure strain as a function of element size (EQ.0.0: ignore this option)	E10.0
21-30	Method for determining element size EQ.0.0: size=$\sqrt{\text{area}}$ EQ.1.0: size=minimum side length used for time step calculation EQ.2.0: size=maximum side length (area/mimimum side) EQ.3.0: size is based on the direction of strains within the element	E10.0
31-40	Fully integrated element failure option: This is the maximum number of integration's points in a layer which can fail before the layer looses strength. (default=4.0)	E10.0
41-50	Option for including pressure effects in fracture EQ.0.0: ignore pressure sensitivity (default) EQ.1.0: include pressure sensitivity	E10.0

Card 5 (8E10.0)

Columns	Quantity	Format
1-10	Yield strength	E10.0
11-20	Optional load curve number defining effective stress versus effective plastic strain. Cards 7 and 8 are ignored with this option.	E10.0

Card 6 (8E10.0)

Columns	Quantity	Format
1-10	Tangent modulus, ignored if stress strain curve is defined	E10.0
11-20	Plastic strain at failure (ignored if size dependent failure option is input).	E10.0
21-30	Time step size for automatic element deletion	E10.0
31-40	Load curve number to scale yield stress to account for strain rate effects, lc1.	E10.0
41-50	Load curve number to scale yield stress to account for strain rate effects, lc2.	E10.0

Card 7 (8E10.0)

Columns	Quantity	Format
1-80	Effective plastic strain values (define up to 8 points)	E10.0

Card 8 (8E10.0)

Columns	Quantity	Format
1-10	Corresponding yield stress values (define up to 8 points)	E10.0

Material Type 51 (Temperature and Rate Dependent Plasticity)

Columns		Quantity	Format
1-10	Card 3	E, Young's modulus (psi)	E10.0
11-20		PR, Poisson's ratio (-)	E10.0
21-30		T, initial temperature (^oR)	E10.0
31-40		HC, heat generation coefficient ($^oR/psi$)	E10.0
1-10	Card 4	C_1 (psi)	E10.0
11-20		C_2 (^oR)	E10.0
21-30		C_3 (psi)	E10.0
31-40		C_4 (^oR)	E10.0
1-10	Card 5	C_5 ($1/s$)	E10.0
11-20		C_6 (^oR)	E10.0
21-30		C_7 ($1/psi$)	E10.0
31-40		C_8 (^oR)	E10.0
41-50		C_{13} ($1/psi$)	E10.0
51-60		C_{14} (^oR)	E10.0
1-10	Card 6	C_9 (psi)	E10.0
11-20		C_{10} (^oR)	E10.0
21-30		C_{15} (psi)	E10.0
31-40		C_{16} (^oR)	E10.0
41-50		C_{11} ($1/psi\text{-}s$)	E10.0
51-60		C_{12} (^oR)	E10.0
61-70		C_{17} ($1/psi\text{-}s$)	E10.0
71-80		C_{18} (^oR)	E10.0

Columns	Quantity		Format
1-10	Card 7	α_1, initial value of internal state variable 1	E10.0
11-20		α_2, initial value of internal state variable 2	E10.0
21-30		α_4, initial value of internal state variable 3	E10.0
31-40		α_5, initial value of internal state variable 4	E10.0
41-50		α_6, initial value of internal state variable 5	E10.0
51-60		κ, initial value of internal state variable 6	E10.0
	Card 8	Blank	

sec-psi-^{o}R	sec-MPa-^{o}R	sec-MPA-^{o}K
C_1	$*1/145$	$*1/145$
C_2	—	$*5/9$
C_3	$*1/145$	$*1/145$
C_4	—	$*5/9$
C_5	—	—
C_6	—	$*5/9$
C_7	*145	*145
C_8	—	$*5/9$
C_9	$*1/145$	$*1/145$
C_{10}	—	$*5/9$
C_{11}	*145	*145
C_{12}	—	$*5/9$
C_{13}	*145	*145
C_{14}	—	$*5/9$
C_{15}	$*1/145$	$*1/145$
C_{16}	—	$*5/9$
C_{17}	*145	*145
C_{18}	—	$*5/9$
C0=HC	*145	$*145*5/9$
E	$*1/145$	$*1/145$
υ	—	—
T	—	$*5/9$

The kinematics associated with the model are discussed in references [Hill 1948, Bammann and Aifantis 1987, Bammann 1989]. The description below is taken nearly verbatim from Bammann [Hill 1948].

With the assumption of linear elasticity we can write,

$$\overset{o}{\sigma} = \lambda \, \mathrm{tr}\left(D^e\right)1 + 2\mu D^e$$

where,the Cauchy stress σ is convected with the elastic spin W^e as,

$$\overset{o}{\sigma} = \dot{\sigma} - W^e \sigma + \sigma W^e$$

This is equivalent to writing the constitutive model with respect to a set of directors whose direction is defined by the plastic deformation [Bammann and Aifantis 1987, Bammann and Johnson 1987]. Decomposing both the skew symmetric and symmetric parts of the velocity gradient into elastic and plastic parts we write for the elastic stretching D^e and the elastic spin W^e,

$$D^e = D - D^p - D^{th}, W^e = W = W^p.$$

Within this structure it is now necessary to prescribe an equation for the plastic spin W^p in addition to the normally prescribed flow rule for D^p and the stretching due to the thermal expansion D^{th}. As proposed, we assume a flow rule of the form,

$$D^p = f(T)\sinh\left[\frac{|\xi| - \kappa - Y(T)}{V(T)}\right]\frac{\xi'}{|\xi'|}.$$

Where T is the temperate, κ is the scalar hardening variable, ξ′ is the difference between the deviatoric Cauchy stress σ′ and the tensor variable α′,

$$\xi' = \sigma' - \alpha'$$

and $f(T), Y(T), V(T)$ are scalar functions whose specific dependence upon the temperature is given below. Assuming isotropic thermal expansion, and introducing the expansion coefficient $\dot{A}$, the thermal stretching can be written,

$$D^{th} = \dot{A}\dot{T}1.$$

The evolution of the internal variables α and κ are prescribed in a hardening minus recovery format as,

$$\overset{o}{\alpha} = h(T)D^p - \left[r_d\ (T)\left|D^p\right| + r_s\,(T)\right]|\alpha|\,\alpha,$$

$$\dot{\kappa} = H(T)D^p - \left[R_d\,(T)\left|D^p\right| - R_s\,(T)\right]\kappa^2$$

where h and H are the hardening moduli, r_s (T) and R_s (T) are scalar functions describing the diffusion controlled 'static' or 'thermal' recovery, and r_d (T) and R_d (T) are the functions describing dynamic recovery.

If we assume that $W^p = 0$, we recover the Jaumann stress rate which results in the prediction of an oscillatory shear stress response in simple shear when coupled with a Prager kinematic hardening assumption [Johnson and Bammann 1984]. Alternatively we can choose,

$$W^p = R^T \dot{U} U^{-1} R,$$

which recovers the Green-Naghdi rate of Cauchy stress and has been shown to be equivalent to Mandel's isoclinic state [Bammann and Aifantis 1987]. The model employing this rate allows a reasonable prediction of directional softening for some materials but in general under-predicts the softening and does not accurately predict the axial stresses which occur in the torsion of the thin walled tube.

The final equation necessary to complete our description of high strain rate deformation is one which allows us to compute the temperature change during the deformation. In the absence of a coupled thermomechanical finite element code we assume adiabatic temperature change and follow the empirical assumption that 90 -95% of the plastic work is dissipated as heat. Hence,

$$\dot{T} = \frac{.9}{\rho C_v}\left(\sigma \cdot D^p\right),$$

where ρ is the density of the material and C_v the specific heat.

In terms of the input parameters the functions defined above become:

V(T) = C1 exp(-C2/T)	h(T) = C9 exp(C10/T)
Y(T) = C3 exp(C4/T)	rs(T) = C11exp(-C12/T)
f(T) = C5 exp(-C6/T)	RD(T) = C13exp(-C14/T)
rd(T) = C7 exp(-C8/T)	H(T) = C15exp(C16/T)
	RS(T) = C17exp(-C18/T)

and the heat generation coefficient is

$$HC = \frac{.9}{\rho C_V} .$$

Material Type 52 (Sandia's Damage Model)

Columns		Quantity	Format
1-10	Card 3	E, Young's modulus	E10.0
11-20		PR, Poisson's ratio	E10.0
21-30		T, initial temperature	E10.0
31-40		HC, heat generation coefficient	E10.0
1-10	Card 4	C_1	E10.0
11-20		C_2	E10.0
21-30		C_3	E10.0
31-40		C_4	E10.0
1-10	Card 5	C_5	E10.0
11-20		C_6	E10.0
21-30		C_7	E10.0
31-40		C_8	E10.0
41-50		C_{13}	E10.0
51-60		C_{14}	E10.0
1-10	Card 6	C_9	E10.0
11-20		C_{10}	E10.0
21-30		C_{15}	E10.0
31-40		C_{16}	E10.0
41-50		C_{11}	E10.0
51-60		C_{12}	E10.0
61-70		C_{17}	E10.0
71-80		C_{18}	E10.0

Columns		Quantity	Format
1-10	Card 7	α_1, initial value of internal state variable 1	E10.0
11-20		α_2, initial value of internal state variable 2	E10.0
21-30		α_3, initial value of internal state variable 3	E10.0
31-40		α_4, initial value of internal state variable 4	E10.0
41-50		α_5, initial value of internal state variable 5	E10.0
51-60		α_6, initial value of internal state variable 6	E10.0
51-60		N, exponent in damage evolution	E10.0
61-70		Do initial damage (porosity)	E10.0
1-10	Card 8	Failure strain for erosion	E10.0

The evolution of the damage parameter, ϕ, is defined by [Bammann, et al. 1990]

$$\dot{\phi} = \beta \left[\frac{1}{(1-\phi)^N} - (1-\phi) \right]^{\left| D^p \right|}$$

in which

$$\beta = \sinh \left[\frac{2(2N-1)p}{(2N-1)\overline{\sigma}} \right]$$

where p is the pressure and $\overline{\sigma}$ is the effective stress.

Material Type 53 (Low Density Closed Cell Polyurethane Foam)

Columns		Quantity	Format
1-10	Card 3	E, Young's modulus	E10.0
11-20		a	E10.0
21-30		b	E10.0
31-40		c	E10.0
41-50		p_0, initial foam pressure	E10.0
51-60		ϕ, ratio of foam to polymer density	E10.0
61-70		γ_0, initial volumetric strain	E10.0
	Card 4	Blank	
	.	.	
	.	.	
	Card 8	Blank	

A rigid, low density, closed cell, polyurethane foam model developed at Sandia Laboratories [Neilsen et al. 1987] has been recently implemented for modeling impact limiters in automotive applications. A number of such foams were tested at Sandia and reasonable fits to the experimental data were obtained.

In some respects this model is similar to the crushable honeycomb model type 26 in that the components of the stress tensor are uncoupled until full volumetric compaction is achieved. However, unlike the honeycomb model this material possesses no directionality but includes the effects of confined air pressure in its overall response characteristics..

$$\sigma_{ij} = \sigma_{ij}^{sk} - \delta_{ij}\sigma^{air}$$

where σ_{ij}^{sk} is the skeletal stress and σ^{air} is the air pressure computed from the equation:

$$\sigma^{air} = -\frac{p_0\gamma}{1+\gamma-\phi}$$

where p_0 is the initial foam pressure usually taken as the atmospheric pressure and γ defines the volumetric strain

$$\gamma = V - 1 + \gamma_0$$

where V is the relative volume and γ_0 is the initial volumetric strain which is typically zero. The yield condition is applied to the principal skeletal stresses which are updated independently of the air pressure. We first obtain the skeletal stresses:

$$\sigma_{ij}^{sk} = \sigma_{ij} + \sigma_{ij}\,\sigma^{air}$$

and compute the trial stress, σ_i^{skt}

$$\sigma_{ij}^{skt} = \sigma_{ij}^{sk} + E\,\dot{\varepsilon}_{ij}\,\Delta t$$

where E is Young's modulus. Since Poisson's ratio is zero, the update of each stress component is uncoupled and $2G=E$ where G is the shear modulus. The yield condition is applied to the principal skeletal stresses such that if the magnitude of a principal trial stress component, σ_i^{skt}, exceeds the yield stress, σ_y, then

$$\sigma_i^{sk} = \min(\sigma_y, \left|\sigma_i^{skt}\right|)\frac{\sigma_i^{skt}}{\left|\sigma_i^{skt}\right|} \quad .$$

The yield stress is defined by

$$\sigma_y = a + b(1 + c\gamma)$$

where a, b, and c are user defined input constants. After scaling the principal stresses they are transformed back into the global system and the final stress state is computed

$$\sigma_{ij} = \sigma_{ij}^{sk} - \delta_{ij}\sigma^{air} \, .$$

Material Type 54 and 55 (Enhanced Composite Damage Model)

Material 54 uses the Chang matrix failure criterion (as Material 22);
Material 55 uses the Tsay-Wu criterion for matrix failure.

Arbitrary orthothropic materials, e.g., unidirectional layers in composite shell structures can be defined. Optionally, various types of failure can be specified following either the suggestions of [Chang and Chang, 1984] or [Tsai and Wu, 1981]. In addition special measures are taken for failure under compression. See [Matzenmiller and Schweizerhof, 1990]. This model is only valid for thin shell elements.

Columns		Quantity	Format
1-10	Card 3	E_a, Young's modulus - longitudinal direction	E10.0
11-20		E_b, Young's modulus - transverse direction	E10.0
21-40		Blank	E10.0
41-50		β, weighting factor for shear term in tensile fiber mode ($0.0 \le \beta \le 1.0$)	E10.0
51-60		YCFAC, factor for the remainder longitudinal compressive strength after compressive matrix matrix failure. X_c=YCFAC*Y_c	E10.0
1-10	Card 4	ν_{yx}	E10.0
11-20		ν_{zx}	E10.0
21-30		ν_{zy}	E10.0
1-10	Card 5	G_{yx}	E10.0
11-20		G_{yx}	E10.0
21-30		G_{zx}	E10.0
1-10	Card 6	Material axes option, AOPT EQ.0.0: Locally orthotropic with material axes determined by element nodes n_1, n_2, and n_4, as shown in Figure 3.5. The AOPT data on cards 7 and 8 below are blank with this option.	E10.0

Columns	Quantity		Format
		EQ.1.0: Locally orthotropic with material axes determined by a point in space and the global location of the element center. The AOPT data on card 8 below is blank. EQ.2.0: Globally orthotropic with material axes determined by vectors defined on Card 7 and 8. EQ.3.0: Applicable to shell elements only. This option determines locally orthotropic material axes by offsetting the material axes by an angle (Card 12) from a line in the plane of the shell determined by taking the cross product of the vector defined on Card 7 with the shell normal vector.	
31-40	t_{fail}	Time step size for element deletion ≤ 0: no element deletion by time step size The crashfront algorithm only works if t_{fail} is set to a value above zero. $0 \leq t_{fail} \leq .1$: element is deleted, when its time step is smaller than the given value > .1: element is deleted, when the quotient of the actual time step and the original time step drops below the given value	E10.0
41-50	X_c	Longitudinal compressive strength	E10.0
51-60	SOFT	Softening reduction factor for material strength in crashfront elements (default = 1.0). TFAIL must be greater than zero to activate this option.	E10.0
61-70	FBRT	Softening for fiber tensile strength EQ.0.0: tensile strength = X_t GT:0.0: tensile strength = X_t, reduced to X_t*FBRT after failure has occurred in compressive matrix mode.	E10.0

Tension cutoff failure occurs when the tensile strength is reached if DFAILT is equal to 0.0. If DFAILT is greater than 0.0, the tensile stress is maintained at the tensile strength and the element does not fail.

1-10	Card 7	x_p define for AOPT=1.0	E10.0
11-20		y_p define for AOPT=1.0	E10.0
21-30		z_p define for AOPT=1.0	E10.0

Columns		Quantity	Format
1-10	Card 7	a_1 define for AOPT=2.0	E10.0
11-20		a_2 define for AOPT=2.0	E10.0
21-30		a_3 define for AOPT=2.0	E10.0
1-10	Card 7	v_1 define for AOPT=3.0	E10.0
11-20		v_2 define for AOPT=3.0	E10.0
21-30		v_3 define for AOPT=3.0	E10.0
31-40		DFAILT, failure strain for tensile fiber mode EQ.0.0: fiber rupture with tension cutoff GT.0.0: stress = FBRT * X_t after failure, tension cutoff if tensile strain > DFAILT	E10.0
41-50		DFAILC, failure strain (<0) for compressive fiber mode. Only active if DFAILT.GT.0.0.	E10.0
51-60		ERODS, erosion strain (>0). If the effective strain in an element exceeds ERODS, the element is eroded. Note that an element is also eroded if each ply through the thickness fails; however, this may fail to work and excessive element stretching results.	E10.0
61-70		DFAILM, Maximum strain for matrix straining in tension or compression. The layer in the element is completely removed after the maximum strain in the matrix direction is reached. The input value is always positive.	E10.0
71-80		DFAILS, Maximum shear strain. The layer in the element is completely removed after the maximum shear strain is reached. The input value is always positive.	E10.0
1-10	Card 8	d_1 define for AOPT=2.0	E10.0
11-20		d_2 define for AOPT=2.0	E10.0
21-30		d_3 define for AOPT=2.0	E10.0
31-40		S_C shear strength, ab plane	E10.0
41-50		x_t longitudinal tensile strength, a-axis	E10.0
51-60		y_t transverse tensile strength, b-axis	E10.0
61-70		y_c transverse compressive strength	E10.0
71-80		α, Nonlinear shear stress parameter	E10.0

\ The Chang/Chang criteria is given as follows:

for the tensile fiber mode,

$$\sigma_{aa} > 0 \quad then \quad e_f^2 = \left(\frac{\sigma_{aa}}{X_t}\right)^2 + \beta\left(\frac{\sigma_{ab}}{S_c}\right) - 1 \begin{cases} \geq 0 \; failed \\ < 0 \; elastic \end{cases},$$

$$E_a = E_b = G_{ab} = \nu_{ba} = \nu_{ab} = 0,$$

for the compressive fiber mode,

$$\sigma_{aa} < 0 \quad then \quad e_c^2 = \left(\frac{\sigma_{aa}}{X_c}\right)^2 - 1 \begin{cases} \geq 0 \; failed \\ < 0 \; elastic \end{cases},$$

$$E_a = \nu_{ba} = \nu_{ab} = 0.$$

for the tensile matrix mode,

$$\sigma_{bb} > 0 \quad then \quad e_m^2 = \left(\frac{\sigma_{bb}}{Y_t}\right)^2 + \left(\frac{\sigma_{ab}}{S_c}\right)^2 - 1 \begin{cases} \geq 0 \; failed \\ < 0 \; elastic \end{cases},$$

$$E_b = \nu_{ba} = 0. \quad \rightarrow G_{ab} = 0,$$

and for the compressive matrix mode,

$$\sigma_{bb} < 0 \quad then \quad e_d^2 = \left(\frac{\sigma_{bb}}{2S_c}\right)^2 + \left[\left(\frac{Y_c}{2S_c}\right)^2 - 1\right]\frac{\sigma_{bb}}{Y_c} + \left(\frac{\sigma_{ab}}{S_c}\right)^2 - 1 \begin{cases} \geq 0 \; failed \\ < 0 \; elastic \end{cases},$$

$$_b = \nu_{ba} = \nu_{ab} = 0. \quad \rightarrow G_{ab} = 0$$
$$X_c = 2Y_c \quad for\ 50\%\ fiber\ volume.$$

In the Tsay/Wu criteria the tensile and compressive fiber modes are treated as in the Chang/Chang criteria. The failure criterion for the tensile and compressive matrix mode is given as:

$$e_{md}^2 = \frac{\sigma_{bb}^2}{Y_c Y_t} + \left(\frac{\sigma_{ab}}{S_c}\right)^2 + \frac{(Y_c - Y_t)\sigma_{bb}}{Y_c Y_t} - 1 \begin{cases} \geq 0 \; failed \\ < 0 \; elastic \end{cases}$$

For β =1 we get the original criterion of Hashin [1980] in the tensile fiber mode. For β =0 we get the maximum stress criterion which is found to compare better to experiments.

Failure can occur in any of four different ways:

1. If DFAILT is zero, failure occurs if the Chang/Chang failure criterion is satisfied in the tensile fiber mode.

2. If DFAILT is greater than zero, failure occurs if the tensile fiber strain is greater than DFAILT or less than DFAILC.

3. If EFS is greater than zero, failure occurs if the effecive strain is greater than EFS.

4. If TFAIL is greater than zero, failure occurs according to the element timestep as described in the definition of TFAIL above.

When failure has occurred in all the composite layers (through-thickness integration points), the element is deleted. Elements which share nodes with the deleted element become “crashfront” elements and can have their strengths reduced by using the SOFT parameter with TFAIL greater than zero.

Information about the status in each layer (integration point) and element can be plotted using additional integration point variables. The number of additional integration point variables for shells written to the LS-TAURUS database is input by the *DATABASE_BINARY definition as variable NEIPS. For Models 54 and 55 these additional variables are tabulated below (*i* = shell integration point):

History Variable	Description	Value	LS-TAURUS Component
1.$ef(i)$	*tensile fiber mode*	1 - elastic 0 - failed	*81*
2.$ec(i)$	*compressive fiber mode*		*82*
3.$em(i)$	*tensile matrix mode*		*83*
4.$ed(i)$	*compressive matrix mode*		*84*
5.*efail*	*max[ef(ip)]*		*85*

6.*dam*	*damage parameter*	*-1 - element intact* *10^{-8} - element in crashfront* *+1 - element failed*	*86*

These variables can be plotted in LS-TAURUS as element components 81, 82, ..., 80+ NEIPS. The following components, defined by the sum of failure indicators over all through-thickness integration points, are stored as element component 7 instead of the effective plastic strain.:

Description	Integration point
$\frac{1}{nip}\sum_{i=1}^{nip} ef(i)$	*1*
$\frac{1}{nip}\sum_{i=1}^{nip} ec(i)$	*2*
$\frac{1}{nip}\sum_{i=1}^{nip} cm(i)$	*3*

Examples:

a) Fringe of tensile fiber mode for integration point 3:

LS-TAURUS commands in Phase I: *intg 3 frin 81*

LS-TAURUS commands in Phase II: *etime 81 n e1 e2 ... en*

b) Sum of failure indicator of compressive fiber mode:

LS-TAURUS commands in Phase I: *intg 2 frin 7*

LS-TAURUS commands in Phase II: *etime 7 3 e e e* (with *e* ... element number)

c) Visualization of crashfront via *dam* parameter

LS-TAURUS commands: *frin 86*

Material Type 57 (Low Density Urethane Foam)

Columns		Quantity	Format
1-10	Card 3	E, Young's modulus	E10.0
11-20		Curve number of nominal stress versus strain	E10.0
21-30		Tension cut-off stress	E10.0
31-40		Hysteretic unloading factor between 0 and 1 (Default=1, i.e., no energy dissipation)	E10.0
41-50		β, decay constant (Default=0., i.e., no relaxation)	E10.0
51-60		Viscous coefficient (.05<recommended value<.50) for stress oscillations and shock waves.	E10.0
61-70		Shape factor for unloading. Active for nonzero values of the hysteretic unloading factor. Values less than one reduces the energy dissipation and greater than one increases dissipation.	E10.0
71-80		Failure option after cutoff stress is reached. EQ.0: tensile stress remains at cut-off value EQ.1: tensile stress is reset to zero.	E10.0
1-10	Card4	Bulk viscosity activation flag. EQ.0.0: no bulk viscosity (recommended) EQ.1.0: bulk viscosity active	E10.0
11-20		Optional Young's relaxation modulus, E_d, for rate effects.	E10.0
21-30		Optional decay constant, β_1.	E10.0
1-10	Card5	Stiffness coefficient for contact interface stiffness. EQ.0.0: Maximum slope in stress vs. strain curve is used. When the maximum slope is taken for the contact, the time step size for this material is reduced for stability. In some cases Δt may be significantly smaller, and defining a reasonable stiffness is recommended.	E10.0
1-10	Card6	Use reference geometry to initial stresses. EQ.0.0: off EQ.1.0: on	E10.0
	Card 7	Blank	
	.	.	
	.	.	
	Card 8	Blank	

This urethane foam model is available to model highly compressible foams such as those used in seat cushions and as padding on the Side Impact Dummy (SID). The compressive behavior is illustrated in Figure 3.16 where hysteresis on unloading is shown. This behavior under uniaxial loading is assumed not to significantly couple in the transverse directions. In tension the material behaves in a linear fashion until tearing occurs. Although our implementation may be somewhat unusual, it was first motivated by Shkolnikov [1991] and a paper by Storakers [1986]. The recent additions necessary to model hysteretic unloading and rate effects are due to Chang, et. al. [1994]. These latter additions have greatly expanded the usefulness of this model.

The model uses tabulated input data for the loading curve where the nominal stresses are defined as a function of the elongations, ε_i, which are defined in terms of the principal stretches, λ_i, as:

$$\varepsilon_i = \lambda_i - 1$$

The stretch ratios are found by solving for the eigenvalues of the left stretch tensor, V_{ij}, which is obtained via a polar decomposition of the deformation gradient matrix, F_{ij} Recall that,

$$F_{ij} = R_{ik} U_{kj} = V_{ik} R_{kj}$$

The update of V_{ij} follows the numerically stable approach of Taylor and Flanagan [1989]. After solving for the principal stretches, the elongations are computed and, if the elongations are compressive, the corresponding values of the nominal stresses, τ_i, are interpolated. If the elongations are tensile, the nominal stresses are given by

$$\tau_i = E\varepsilon_i$$

The Cauchy stresses in the principal system become

$$\sigma_i = \frac{\tau_i}{\lambda_j \lambda_k} \quad .$$

The stresses are then transformed back into the global system for the nodal force calculations.

When hysteretic unloading is used, the reloading will follow the unloading curve if the decay constant, β, is set to zero. If β is nonzero the decay to the original loading curve is governed by the expression:

$$1 - e^{-\beta t}$$

The bulk viscosity, which generates a rate dependent pressure, may cause an unexpected volumetric response and, consequently, it is optional with this model.

Rate effects are accounted for through linear viscoelasticity by a convolution integral of the form

$$\sigma_{ij}^{r} = \int_{0}^{t} g_{ijkl}(t-\tau)\frac{\partial \varepsilon_{kl}}{\partial \tau}\, d\tau$$

where $g_{ijkl}(t-\tau)$ is the relaxation function. The stress tensor, σ_{ij}^{r}, augments the stresses determined from the foam, σ_{ij}^{f}; consequently, the final stress, σ_{ij}, is taken as the summation of the two contributions:

$$\sigma_{ij} = \sigma_{ij}^{f} + \sigma_{ij}^{r}.$$

Since we wish to include only simple rate effects, the relaxation function is represented by one term from the Prony series:

$$g(t) = \alpha_0 + \sum_{m=1}^{N} \alpha_m e^{-\beta t}$$

given by,

$$g(t) = E_d e^{-\beta_1 t}$$

This model is effectively a Maxwell fluid which consists of a damper and spring in series. We characterize this in the input by a Young's modulus, E_d, and decay constant, β_1. The formulation is performed in the local system of principal stretches where only the principal values of stress are computed and triaxial coupling is avoided. Consequently, the one-dimensional nature of this foam material is unaffected by this addition of rate effects. The addition of rate effects necessitates twelve additional history variables per integration point. The cost and memory overhead of this model comes primarily from the need to “remember” the local system of principal stretches.

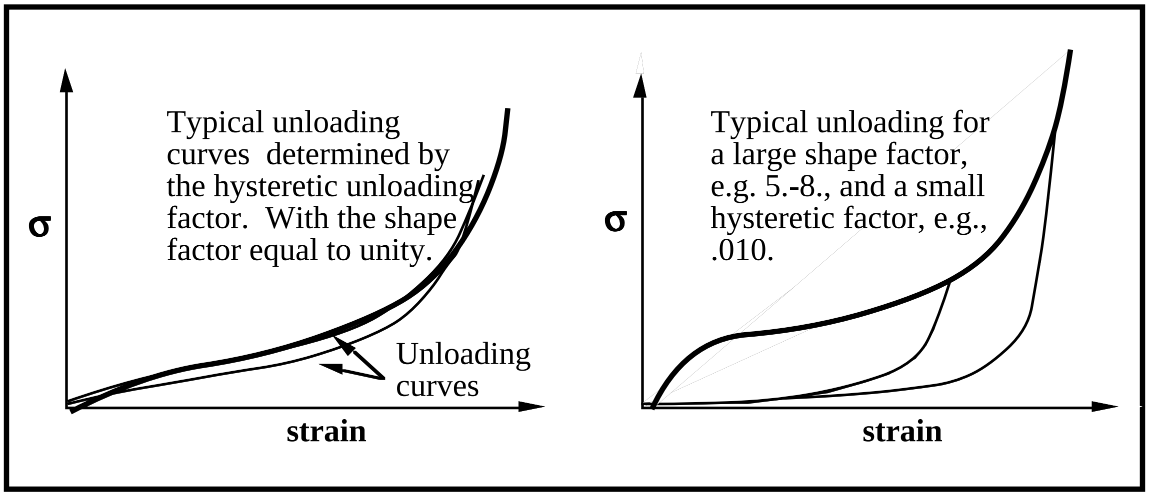


Figure 3.16. Behavior of the low density urethane foam model.

Material Type 58 (Laminated Composite/Fabric)

Depending on the type of failure surface, this model may be used to model composite materials with unidirectional layers, complete laminates, and fabrics. This model is implemented only for shell elements

Columns		Quantity	Format
1-10	Card 3	E_a, longitudinal direction	E10.0
11-20		E_b, transverse direction	E10.0
21-30		E_c, normal direction which is currently not used.	E10.0
1-10	Card 4	υ_{ba}	E10.0
11-20		$\tau 1$, stress limit of the first slightly nonlinear part of the of the shear stress versus shear strain curve. The values $\tau 1$ and $\gamma 1$ are used to define a curve of shear stress versus shear strain. These values are input if FS, defined below, is set to a value of -1.	E10.0
21-30		$\gamma 1$, strain limit of the first slightly nonlinear part of the of the shear stress versus shear strain curve.	E10.0
1-10	Card 5	G_{ab}	E10.0
11-20		G_{bc}	E10.0
21-30		G_{ca}	E10.0
31-40		SLIMT1, minimum stress limit after stress maximum. (fiber tension) This is a factor between 0.0 and 1.0. Default=1.0e-08.	E10.0
41-50		SLIMC1, minimum stress limit after stress maximum. (fiber compression) This is a factor between 0.0 and 1.0. Default=1.0e-08.	E10.0
51-60		SLIMT2, minimum stress limit after stress maximum. (matrix tension) This is a factor between 0.0 and 1.0. Default=1.0e-08.	E10.0
61-70		SLIMC2, minimum stress limit after stress maximum. (matrix compression) This is a factor between 0.0 and 1.0. Default=1.0e-08.	E10.0
71-80		SLIMS, minimum stress limit after stress maximum. (shear) This is a factor between 0.0 and 1.0. Default=1.0e-08.	E10.0

Columns		Quantity	Format
1-10	Card 6	Material axes option, AOPT EQ.0.0: locally orthotropic with material axes determined by element nodes n_1, n_2, and n_4 as shown in Figure 3.5. Cards 7 and 8 below are blank with this option. EQ.1.0:locally orthotropic with material axes determine by a point in space and the global location of the element center.Card 8 below is blank. EQ.2.0:globally orthotropic with material axes determined by vectors defined on Cards 7 and 8. EQ.3.0:applicable to shell elements only. This option determines locally orthotropic material axes by offsetting the material axes by an angle (Card 8) from a line in the plane of the shell determined by taking the cross product of the vector defined on Card 7 with the shell normal vector.	
11-20		Blank	E10.0
21-30		Blank	E10.0
31-40		TSIZE, time step for automatic element deletion	E10.0
41-50		ERODS, maximum effective strain for element layer failure. A value of 1 = 100% strain.	E10.0
51-60		SOFT, softening reduction factor for strength in crash	E10.0
61-70		FS, Failure surface type: EQ.1.0:smooth failure surface with a quadratic criterion for both the fiber (a) and transverse (b) directions. EQ.0.0:smooth failure surface in the transverse (b) direction with a limiting value in the fiber (a) direction. This model is appropiate for unidirectional (UD) layered composites. EQ.-1.:faceted failure surface. When the strength values are reached then damage evolves in tension and compression for both the fiber and transverse direction. Shear behavior is also considered.	E10.0

Columns		Quantity	Format
1-10	Card 7	x_p define for AOPT=1.0	E10.0
11-20		y_p define for AOPT=1.0	E10.0
21-30		z_p define for AOPT=1.0	E10.0
1-10	Card 7	a_1 define for AOPT=2.0	E10.0
11-20		a_2 define for AOPT=2.0	E10.0
21-30		a_3 define for AOPT=2.0	E10.0
1-10	Card 7	v_1 define for AOPT=3.0	E10.0
11-20		v_2 define for AOPT=3.0	E10.0
21-30		v_3 define for AOPT=3.0	E10.0
31-40	Card 7	e_{11c}, strain at longitudinal compressive strength, a-axis.	E10.0
41-50		e_{11t}, strain at longitudinal tensile strength, a-axis.	E10.0
51-60		e_{22c}, strain at transverse compressive strength, b-axis.	E10.0
61-70		e_{22t} , strain at transverse tensile strength, b-axis.	E10.0
71-80		e_s, strain at shear strength, ab plane.	E10.0
1-10	Card 8	d_1 define for AOPT=2.0	E10.0
11-20		d_2 define for AOPT=2.0	E10.0
21-30		d_3 define for AOPT=2.0	E10.0
31-40		x_c longitudinal compressive strength, a-axis	E10.0
41-50		x_t longitudinal tensile strength, a-axis	E10.0
51-60		y_c transverse compressive strength, b-axis	E10.0
61-70		y_t transverse tensile strength, b-axis	E10.0
71-80		S_c shear strength, ab plane	E10.0

Material Type 59 (Composite Failure Model - Plasticity Based)

Columns		Quantity	Format
1-10	Card 3	E_a, longitudinal direction	E10.0
11-20		E_b, transverse direction	E10.0
21-30		E_c, normal direction	E10.0
31-40		K_f, bulk modulus of failed material	E10.0
	The following parameters (cols. 41-60) apply to shell elements only.		
41-50		s_r, reduction factor(default=0.447)	E10.0
51-60		s_f, softening factor(default=0.0)	E10.0
1-10	Card 4	υ_{ba}	E10.0
11-20		υ_{ca}	E10.0
21-30		υ_{cb}	E10.0
	The following parameters (cols. 31-60) apply to brick elements only.		
31-40	Card 4	s_{ba}, in plane shear strength	E10.0
41-50		s_{ca}, transverse shear strength	E10.0
51-60		s_{cb}, transverse shear strength	E10.0
1-10	Card 5	G_{ab}	E10.0
11-20		G_{bc}	E10.0
21-30		G_{ca}	E10.0
	The following parameters (cols. 31-60) apply to brick elements only.		
31-40	Card 5	x_c, longitudinal compressive strength, a-axis	E10.0
41-50		y_c, transverse compressive strength, b-axis	E10.0
51-60		z_c, normal compressive strength, c-axis	E10.0

Material Type 59 (Composite Failure)

Columns		Quantity	Format
1-10	Card 6	Material axes option, AOPT EQ.0.0: locally orthotropic with material axes determined by element nodes n_1, n_2, and n_4 as shown in Figure 3.5. The AOPT data on cards 7 and 8 below are blank with this option. EQ.1.0: locally orthotropic with material axes determine by a point in space and the global location of the element center. The AOPT data on card 8 below is blank. EQ.2.0: globally orthotropic with material axes determined by vectors defined on Cards 7 and 8. EQ.3.0: applicable to shell elements only. This option determines locally orthotropic material axes by offsetting the material axes by an angle (Card 8) from a line in the plane of the shell determined by taking the cross product of the vector defined on Card 7 with the shell normal vector.	
11-20		Material axes change flag for brick elements EQ.1.0: default EQ.2.0: switch material axes a and b EQ.3.0: switch material axes a and c	E10.0
21-30			E10.0
31-40		tsize, time step for automatic element deletion	E10.0
41-50		alp, nonlinear shear stress parameter	E10.0
51-60		soft, softening reduction factor for strength in crash	E10.0
61-70		fbrt, softening of fiber tensile strength	E10.0
1-10	Card 7	x_p define for AOPT=1.0	E10.0
11-20		y_p define for AOPT=1.0	E10.0
21-30		z_p define for AOPT=1.0	E10.0
1-10	Card 7	a_1 define for AOPT=2.0	E10.0
11-20		a_2 define for AOPT=2.0	E10.0
21-30		a_3 define for AOPT=2.0	E10.0

Columns	Quantity		Format
1-10	Card 7	v_1 define for AOPT=3.0	E10.0
11-20		v_2 define for AOPT=3.0	E10.0
21-30		v_3 define for AOPT=3.0	E10.0
1-10	Card 8	d_1 define for AOPT=2.0	E10.0
11-20		d_2 define for AOPT=2.0	E10.0
21-30		d_3 define for AOPT=2.0	E10.0
	The following parameters (cols. 31-60) apply to brick elements only.		
31-40		x_t longitudinal tensile strength, a-axis	E10.0
41-50		y_t transverse tensile strength, b-axis	E10.0
51-60		z_t normal tensile strength, c-axis	E10.0
	The following parameters (cols. 31-80) apply to shell elements only.		
31-40		x_c longitudinal compressive strength, a-axis	E10.0
41-50		x_t longitudinal tensile strength, a-axis	E10.0
51-60		y_c transverse compressive strength, b-axis	E10.0
61-70		y_t transverse tensile strength, b-axis	E10.0
71-80		S_c shear strength, ab plane if $S_c > 0$ ==> faceted failure surface theory if $S_c < 0$ ==> ellipsoidal failure surface theory.	E10.0

Material Type 60 (Elastic With Viscosity)

Card 3

Columns	Quantity	Format
1-10	Young's modulus (ignored if Card 7 is not blank)	E10.0
11-20	Viscosity V_o (see note below)	E10.0
21-30	Viscosity coefficient A (see note below)	E10.0
31-40	Viscosity coefficient B	E10.0
41-50	Viscosity coefficient C	E10.0
51-60	Load curve defining factor on viscosity versus time (default: factor = 1.0)	E10.0
21-30	Viscosity coefficient A (see note below)	E10.0

Card 4

If only one value is defined Poisson's ratio is independent of temperature.

Columns	Quantity	Format
1-10	Poisson's ratio at T_1	E10.0
.	.	.
.	.	.
.	.	.
71-80	Poisson's ratio at T_8	E10.0

Card 5

Temperature dependence is optional. If this card is blank the material is not temperature dependent.

Columns	Quantity	Format
1-10	Temperature at T_1	E10.0
.	.	.
.	.	.
.	.	.
71-80	Temperature at T_8	E10.0

Card 6

Columns	Quantity	Format
1-10	Viscosity at T_1 (see note below)	E10.0
.	.	.
.	.	.
.	.	.
71-80	Viscosity at T_8	E10.0

Card 7

Columns	Quantity	Format
1-10	Young’s modulus at T_1	E10.0
.	.	.
.	.	.
.	.	.
71-80	Young’s modulus at T_8	E10.0

Card 8

Columns	Quantity	Format
1-10	Coefficient of thermal expansion at T_1	E10.0
.	.	.
.	.	.
.	.	.
71-80	Coefficient of thermal expansion at T_8	E10.0

In this material model, the strain has both elastic and viscous components. The model was developed to represent glass at temperatures around 600°C, to allow glass forming processed to be simulated.

Any or all of the properties can vary with temperature. Temperatures can be input by nay of the methods available on Control Card 5.

Viscosity is not active during dynamic relaxation, i.e., the material becomes linear elastic.

Notes:

1. The variation of viscosity with temperature can be defined in any of the 3 ways.
   (i) Constant, $V = V_0$ (use card 3, columns 11-20).
   (ii) $V = V_0 \times 10 ** (A / (T - B) + C)$ (use card 3, columns 11-20, 21-30, 31-40 and 41-50; leave card 6 blank).
   (iii) Piecewise-linear: use card 6, and leave card 3 columns 11-50 blank.

Material Type 61 (Maxwell/Kelvin Viscoelastic with Maximum Strain)

This model outputs strain data, see Bandak [1991], that is used to predict damage in special types of viscoelastic material behavior.

Columns		Quantity	Format
1-10	Card 3	Bulk modulus (elastic)	E10.0
1-10	Card 4	Short-time shear modulus, G_0	E10.0
1-10	Card 5	Long-time shear modulus, G_∞	E10.0
1-10	Card 6	Maxwell decay constant, β [FOPT=0.0] or Kelvin relaxation constant, τ [FOPT=1.0]	E10.0
1-10	Card 7	Formulation option, FOPT EQ.0.0: Maxwell EQ.1.0: Kelvin	E10.0
1-10	Card 8	Strain output option to be plotted as component 7 in LS-TAURUS which is the effective plastic strain component. The maximum values are updated for each element each time step. EQ.0.0: maximum principal strain that occurs during the calculation. EQ.1.0: maximum magnitude of the principal strain that occurs during the calculation. Both positive and negative values are examined.. EQ.2.0: maximum effective strain that occurs during the calculation.	E10.0

The shear relaxation behavior is described for the Maxwell model by:

$$G(t) = G_\infty + (G_0 - G_\infty)e^{-\beta t}.$$

A Jaumann rate formulation is used

$$\overset{\nabla}{s'_{ij}} = 2 \int_0^t G(t-\tau)\, \dot{\varepsilon}'_{ij}(\tau) dt$$

where the prime denotes the deviatoric part of the stress rate, $\overset{\nabla}{s'_{ij}}$, and $\dot{\varepsilon}'_{ij}$ is the deviatoric the strain rate.

For the Kelvin model the stress evolution equation is defined as:

$$\dot{s}_{ij}+\frac{1}{\tau}s_{ij}=\left(1+\delta_{ij}\right)G_0\dot{\varepsilon}'_{ij}+\left(1+\delta_{ij}\right)\frac{G_\infty}{\tau}\varepsilon'_{ij}$$

where δ_{ij} is the Kronecker delta, G_0 is the instantaneous shear modulus, G_∞ is the long term shear modulus, and τ is the decay constant.

The pressure is determined from the bulk modulus and the volumetric strain:

$$p=-K\varepsilon_v$$

where

$$\varepsilon_v=\ln\left(\frac{V}{V_0}\right)$$

defines the logrithmic volumetric strain from the relative volume.

Bandak's [1991] calculation of the total strain tensor, ε_{ij}, for output uses an incremental update based on Jaumann rate:

$$\varepsilon_{ij}^{n+1}=\varepsilon_{ij}^{n}+r_{ij}^{n}+\varepsilon_{ij}^{\nabla n+½}\Delta t^{n+½}$$

where

$$\Delta\varepsilon_{ij}^{n+½}=\dot{\varepsilon}_{ij}^{\,n+½}\,\Delta t^{n+½}$$

and r_{ij}^{n} gives the rotation of the stain tensor at time tt^n to the configuration at t^{n+1}

$$r_{ij}^{n}=\left(\varepsilon_{ip}^{n}\omega_{pj}^{n+½}+\varepsilon_{jp}^{n}\omega_{pi}^{n+½}\right)\Delta t^{n+½}\ .$$

Material Type 62 (Viscous Foam, Ove Arup & Partners Model)

Columns		Quantity	Format
1-10	Card 3	Initial Young's modulus (E_1)	E10.0
11-20		Power law for Young's modulus (n_1)	E10.0
21-30		Viscous coefficient (V_2)	E10.0
31-40		Elastic modulus for viscosity (E_2)	E10.0
41-50		Power law for viscosity (n_2)	E10.0
1-10	Card 4	Posson's ratio, ν	
	Card 5	Blank	
	.	.	
	.	.	
	Card 8	Blank	

This model was written to represent the energy absorbing foam found on certain crash dummies. This model was added to model the 'Confor Foam' on the ribs of the Eurosid.

The model consists of a nonlinear elastic stiffness in parallel with a viscous damper. The elastic stiffness is intended to limit total crush while the viscosity absorbs energy. The stiffness E_2 exists to prevent timestep problems.

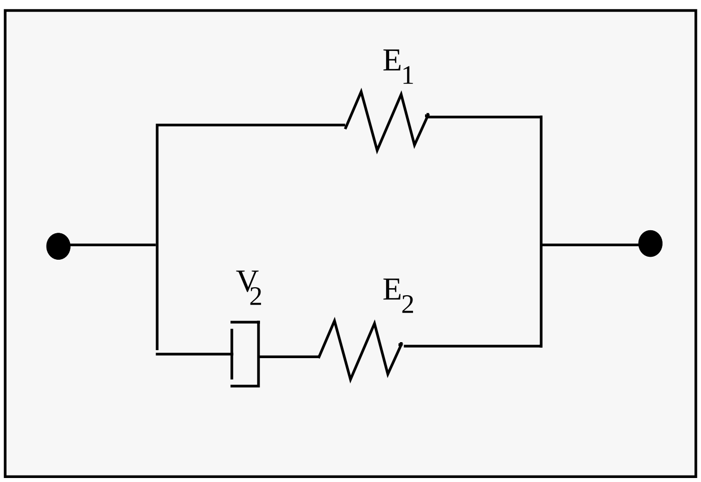


Figure 3.17. Schematic of material model 62.

Both E_1 and V_2 are nonlinear with crush as follows:

$$E_1^t = E_1\left(V^{-n_1}\right)$$

$$V_2^t = V_2\left(abs(1-V)\right)^{n_2}$$

where V is the relative volume defined by the ratio of the current to initial volume. Typical values are (units of N, mm, s)

$$E_1 = 0.0036$$
$$n_1 = 4.0$$
$$V_2 = 0.0015$$
$$E_2 = 100.0$$
$$n_2 = 0.2$$
$$\nu = 0.05$$

Material Type 63 (Isotropic Crushable Foam)

Columns	Quantity		Format
1-10	Card 3	Young's modulus (E)	E10.0
11-20		Poisson's ratio (generally 0, but <.50)	E10.0
21-30		Load curve number defining yield stress versus volumetric strain, γ, see Figure 3.18.	E10.0
31-40		Cutoff value for tensile stress.(> 0.0)	E10.0
41-50		Viscous coefficient (.05<recommended value <.50)	E10.0
	Card 4	Blank	
	.	.	
	.	.	
	Card 8	Blank	

The volumetric strain is defined in terms of the relative volume, V, as:

$$\gamma = 1. - V \ .$$

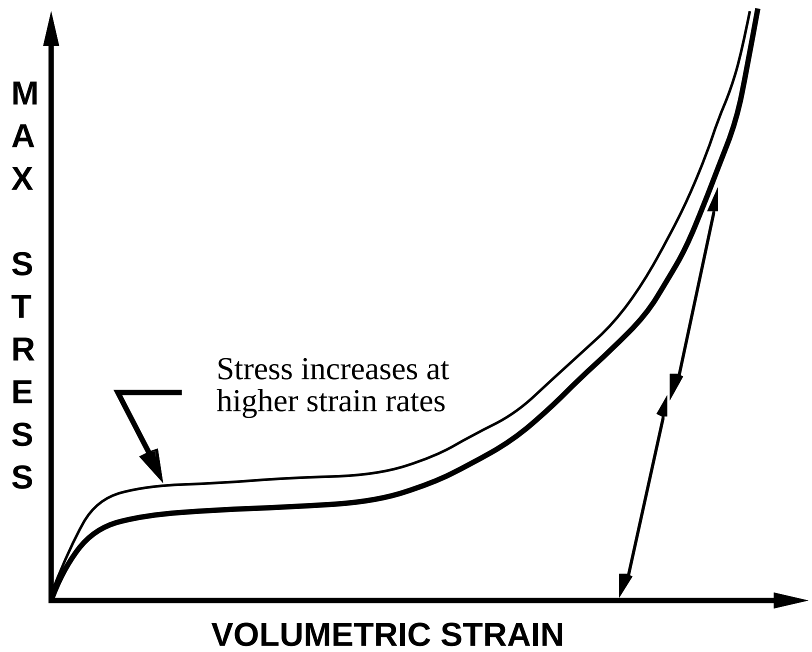


Figure 3.18. Behavior of strainrate sensitive crushable foam. Unloading is elastic to the tension cutoff. Subsequent reloading follows the unloading curve.

Material Type 64 (Strain Rate Sensitive Power-Law Plasticity)

Columns		Quantity	Format
1-10	Card 3	Modulus of elasticity	E10.0
11-20		Poisson's ratio	E10.0
21-30		Material constant, *k*. If *k*<0 the absolute value of *k* is taken as the load curve number that defines *k* as a function of plastic strain.	E10.0
31-40		Strain hardening coefficient, *m*. If *m*<0 the absolute value of *m* is taken as the load curve number that defines *m* as a function of plastic strain	E10.0
41-50		Strain rate sensitivity coefficient, *n*. If *n*<0 the absolute value of *n* is taken as the load curve number that defines *n* as a function of plastic strain	E10.0
51-60		Initial strain rate (0.0002)	E10.0
61-70		Formulation for rate effects: EQ.0.0: Scale yield stress (default) EQ.1.0: Viscoplastic formulation	E10.0
71-80		Factor to normalize strain rate EQ.1.0: Time units of seconds (default) EQ.1.E-3: Time units of milliseconds EQ.1.E-6: Time units of microseconds	E10.0
	Card 4	Blank	
	Card 5	Blank	
	Card 6	Blank	
	Card 7	Blank	
	Card 8	Blank	

This material model follows a constitutive relationship of the form:

$$\sigma = k\varepsilon^m \dot{\varepsilon}^n$$

where σ is the yield stress, ε is the effective plastic strain, $\dot{\varepsilon}$ is the normalized effective plastic strain rate, and the constants *k*, *m*, and *n* can be expressed as functions of effective plastic strain or can be constant with respect to the plastic strain. The case of no strain hardening can be obtained by setting the exponent of the plastic strain equal to a very small positive value, i.e. 0.0001.

This model can be combined with the superplastic forming input to control the magnitude of the pressure in the pressure boundary conditions in order to limit the effective plastic strain rate so that it does not exceed a maximum value at any integration point within the model.

A fully viscoplastic formulation is optional. An additional cost is incurred but the improvement is results can be dramatic.

Material Type 65 (Modified Zerilli/Armstrong)

Columns		Quantity	Format
1-10	Card 3	G, shear modulus	E10.0
11-20		$\dot{\varepsilon}_0$, factor to normalize strain rate	E10.0
21-30		n	E10.0
31-40		T_r, room temperature	E10.0
41-50		pc, pressure cutoff	E10.0
51-60		Spall Type EQ.0.0: default set to "2.0" EQ.1.0: minimum pressure limit EQ.2.0: maximum principal stress EQ.3.0: minimum pressure cutoff	E10.0
61-70		Failure strain for erosion	E10.0
61-70		Formulation for rate effects: EQ.0.0: Scale yield stress (default) EQ.1.0: Viscoplastic formulation	E10.0
1-10	Card 4	C_1	E10.0
11-20		C_2	E10.0
21-30		C_3	E10.0
31-40		C_4	E10.0
41-50		C_5	E10.0
51-60		C_6	E10.0
1-10	Card 5	B_1	E10.0
11-20		B_2	E10.0
21-30		B_3	E10.0
1-10	Card 6	G_1	E10.0
11-20		G_2	E10.0

Material Type 65 (Modified Zerilli/Armstrong)

Columns	Quantity	Format
21-30	G_3	E10.0
31-40	G_4	E10.0
41-50	BULK, bulk modulus defined for shell elements only. Do not input for solid elements.	E10.0

The Armstrong-Zerilli Material Model expresses the flow stress as follows.

For fcc metals,

$$\sigma = C_1 + \left\{ C_2 \left(\varepsilon^p \right)^{\frac{1}{2}} \left[e^{\left(-C_3 + C_4 \ln\left(\dot{\varepsilon}^* \right) \right) T} \right] + C_5 \right\} \left(\frac{\mu(T)}{\mu(293)} \right)$$

ε^p = effective plastic strain

$\dot{\varepsilon}^* = \frac{\dot{\varepsilon}}{\dot{\varepsilon}_0}$ effective plastic strain rate where $\dot{\varepsilon}_0$ = 1, 1e-3, 1e-6 for time units of seconds, milliseconds, and microseconds, respectively.

For bcc metals,

$$\sigma = C_1 + C_2 e^{\left(-C_3 + C_4 \ln\left(\dot{\varepsilon}^* \right) \right) T} + \left[C_5 \left(\varepsilon^p \right)^n + C_6 \right] \left(\frac{\mu(T)}{\mu(293)} \right)$$

where

$$\left(\frac{\mu(T)}{\mu(293)} \right) = B_1 + B_2 T + B_3 T^2 \quad .$$

The relationship between heat capacity (specific heat) and temperature may be characterized by a cubic polynomial equation as follows:

$$C_p = G_1 + G_2 T + G_3 T^2 + G_4 T^3$$

A fully viscoplastic formulation is optional. An additional cost is incurred but the improvement in results can be dramatic.

Material Type 66 (Linear Stiffness/Linear Viscous 3D Discrete Beam)

The formulation of the discrete beam (type 6) assumes that the beam is of zero length and requires no orientation node. A small distance between the nodes joined by the beam is permitted. The local coordinate system which determines (r,s,t) is given by the coordinate ID in the cross sectional input where the global system is the default. The local coordinate system axes rotate with the average of the rotations of the two nodes that define the beam.

Columns		Quantity	Format
1-10	Card 3	Translational stiffness about local r-axis	E10.0
11-20		Translational stiffness about local s-axis	E10.0
21-30		Translational stiffness about local t-axis	E10.0
31-40		Rotational stiffness about the local r-axis	E10.0
41-50		Rotational stiffness about the local s-axis	E10.0
51-60		Rotational stiffness about the local t-axis	E10.0
1-10	Card 4	Translational viscous damper about local r-axis	E10.0
11-20		Translational viscous damper about local s-axis	E10.0
21-30		Translational viscous damper about local t-axis	E10.0
31-40		Rotational viscous damper about the local r-axis	E10.0
41-50		Rotational viscous damper about the local s-axis	E10.0
51-60		Rotational viscous damper about the local t-axis	E10.0
	Card 5	Blank	
	Card 6	Blank	
	Card 7	Blank	
	Card 8	Blank	

For null stiffness coefficients, no forces corresponding to these null values will develop. The viscous damping coefficients are optional.

Material Type 67 (Nonlinear Stiffness/Viscous 3D Discrete Beam)

The formulation of the discrete beam (type 6) assumes that the beam is of zero length and requires no orientation node. A small distance between the nodes joined by the beam is permitted. The local coordinate system which determines (r,s,t) is given by the coordinate ID in the cross sectional input where the global system is the default. The local coordinate system axes rotate with the average of the rotations of the two nodes that define the beam.

Columns		Quantity	Format
1-10	Card 3	Load curve ID-translational force along the local r-axis versus relative r-displacement. See Figure 3.19 for an explanation on how to define the load curve.	E10.0
11-20		Load curve ID-translational force along the local s-axis versus relative s-displacement.	E10.0
21-30		Load curve ID-translational force along the local t-axis versus relative t-displacement.	E10.0
31-40		Load curve ID-rotational moment about the local r-axis versus relative r-rotation.	E10.0
41-50		Load curve ID-rotational moment about the local s-axis versus relative s-rotation.	E10.0
51-60		Load curve ID-rotational moment about the local t-axis versus relative t-rotation.	E10.0
1-10	Card 4	Load curve ID-translational viscous damping force along the r-axis versus relative r-rotational velocity.	E10.0
11-20		Load curve ID-translational viscous damping force along the s-axis versus relative s-rotational velocity.	E10.0
21-30		Load curve ID-translational viscous damping force along the t-axis versus relative t-rotational velocity.	E10.0
31-40		Load curve ID-rotational viscous damping moment about the r-axis versus relative r-rotational velocity.	E10.0
41-50		Load curve ID-rotational viscous damping moment about the s-axis versus relative s-rotational velocity.	E10.0
51-60		Load curve ID-rotational viscous damping moment about the t-axis versus relative t-rotational velocity.	E10.0

Columns	Quantity	Format
	Card 5 Blank	
	Card 6 Blank	
	Card 7 Blank	
	Card 8 Blank	

For null load curve ID’s, no forces are computed.

The formulation of the discrete beam (type 6) assumes that the beam is of zero length and requires no orientation node. A small distance between the nodes joined by the beam is permitted. The local coordinate system which determines (r,s,t) is given by the coordinate ID, in the cross sectional input where the global system is the default. The local coordinate system axes rotate with the average of the rotations of the two nodes that define the beam.

If different behavior in tension and compression is desired in the calculation of the force resultants, the load curve(s) must be defined in the negative quadrant starting with the most negative displacement then increasing monotonically to the most positive. If the load curve behaves similarly in tension and compression, define only the positive quadrant. Whenever displacement values fall outside of the defined range, the resultant forces will be extrapolated. Figure 3.19 depicts a typical load curve for a force resultant. Load curves used for determining the damping forces and moment resultants always act identically in tension and compression, since only the positive quadrant values are considered, i.e., start the load curve at the origin [0,0].

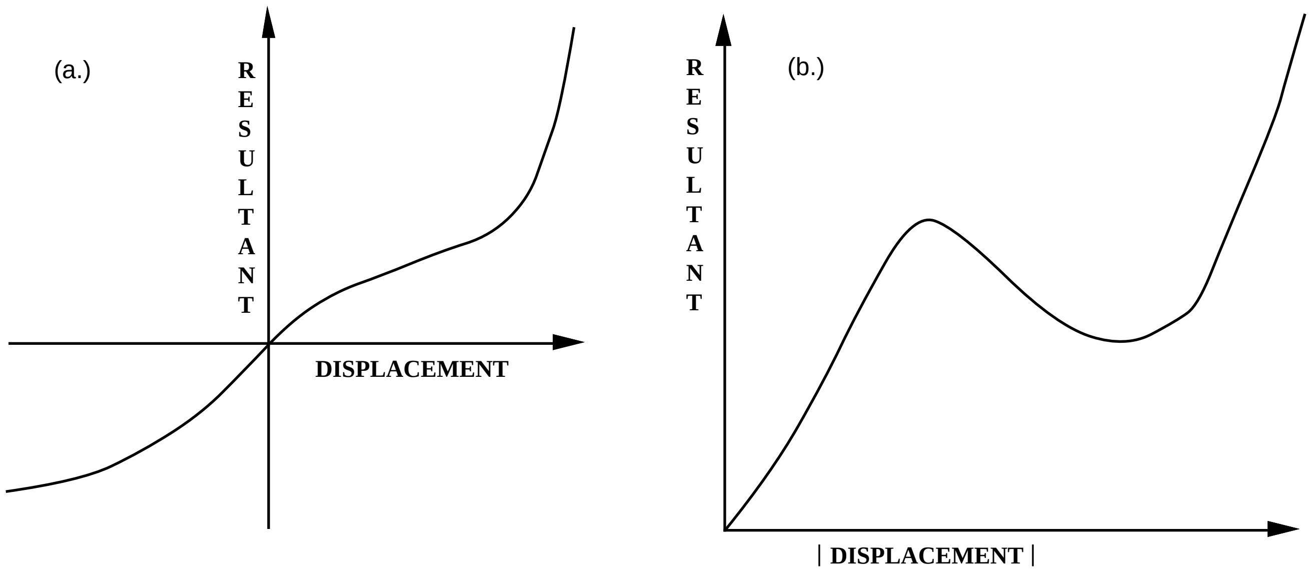


Figure 3.19. The resultant forces and moments are determined by a table lookup. If the origin of the load curve is at [0,0] as in (b.) and tension and compression responses are symmetric.

Material Type 68 (Nonlinear Plastic/Linear Viscous 3D Discrete Beam)

The formulation of the discrete beam (type 6) assumes that the beam is of zero length and requires no orientation node. A small distance between the nodes joined by the beam is permitted. The local coordinate system which determines (*r*,*s*,*t*) is given by the coordinate ID in the cross sectional input where the global system is the default. The local coordinate system axes rotate with the average of the rotations of the two nodes that define the beam.

Columns		Quantity	Format
1-10	Card 3	Translational stiffness along local r-axis	E10.0
11-20		Translational stiffness along local s-axis	E10.0
21-30		Translational stiffness along local t-axis	E10.0
31-40		Rotational stiffness about the local r-axis	E10.0
41-50		Rotational stiffness about the local s-axis	E10.0
51-60		Rotational stiffness about the local t-axis	E10.0
1-10	Card 4	Translational viscous damper along local r-axis	E10.0
11-20		Translational viscous damper along local s-axis	E10.0
21-30		Translational viscous damper along local t-axis	E10.0
31-40		Rotational viscous damper about the local r-axis	E10.0
41-50		Rotational viscous damper about the local s-axis	E10.0
51-60		Rotational viscous damper about the local t-axis	E10.0
1-10	Card 5	Load curve ID-yield force versus plastic displacement r-axis. See Figure 3.20 for an explanation on how to define the load curve.	E10.0
11-20		Load curve ID-yield force versus plastic displacement s-axis	E10.0
21-30		Load curve ID-yield force versus plastic displacement t-axis	E10.0
31-40		Load curve ID-yield moment versus plastic rotation r-axis	E10.0

Columns		Quantity	Format
41-50		Load curve ID-yield moment versus plastic rotation s-axis	E10.0
51-60		Load curve ID-yield moment versus plastic rotation t-axis	E10.0

If Card 6 below is blank failure based on force resultants is not included. If any parameter below is nonzero, failure will be considered.

Columns		Quantity	Format
1-10	Card 6	F_r^{fail}, optional failure parameter. If zero, the corresponding force, F_r, is not considered in the failure calculation.	E10.0
11-20		F_s^{fail}, optional failure parameter. If zero, the corresponding force, F_s, is not considered in the failure calculation.	E10.0
21-30		F_t^{fail}, optional failure parameter. If zero, the corresponding force, F_t, is not considered in the failure calculation.	E10.0
31-40		M_r^{fail}, optional failure parameter. If zero, the corresponding force, M_r, is not considered in the failure calculation.	E10.0
41-50		M_s^{fail}, optional failure parameter. If zero, the corresponding force, M_s, is not considered in the failure calculation.	E10.0
51-60		M_t^{fail}, optional failure parameter. If zero, the corresponding force, M_t, is not considered in the failure calculation.	E10.0

If Card 7 (below) is blank failure based on displacements is not included. If any parameter below is nonzero, failure will be considered.

Columns		Quantity	Format
1-10	Card 7	u_r^{fail}, optional failure parameter. If zero, the corresponding displacement, u_r, is not considered in the failure calculation.	E10.0

11-20	u_s^{fail}, optional failure parameter. If zero, the corresponding displacement, u_S, is not considered in the failure calculation.	E10.0

Columns	Quantity	Format
21-30	u_t^{fail}, optional failure parameter. If zero, the corresponding displacement, u_t, is not considered in the failure calculation.	E10.0
31-40	θ_r^{fail}, optional failure parameter. If zero, the corresponding rotation, θ_r, is not considered in the failure calculation.	E10.0
41-50	θ_s^{fail}, optional failure parameter. If zero, the corresponding rotation, θ_s, is not considered in the failure calculation.	E10.0
51-60	θ_t^{fail}, optional failure parameter. If zero, the corresponding rotation, θ_t, is not considered in the failure calculation.	E10.0

Card 8 Blank

For the translational and rotational degrees of freedom where elastic behavior is desired, set the load curve ID to zero.

Catastrophic failure based on force resultants occurs if the following inequality is satisfied.

$$\left(\frac{F_r}{F_r^{fail}}\right)^2+\left(\frac{F_s}{F_s^{fail}}\right)^2+\left(\frac{F_t}{F_t^{fail}}\right)^2+\left(\frac{M_r}{M_r^{fail}}\right)^2+\left(\frac{M_s}{M_s^{fail}}\right)^2+\left(\frac{M_t}{M_t^{fail}}\right)^2-1.\geq 0.$$

After failure the discrete element is deleted. Likewise, catastrophic failure based on displacement resultants occurs if the following inequality is satisfied:

$$\left(\frac{u_r}{u_r^{fail}}\right)^2+\left(\frac{u_s}{u_s^{fail}}\right)^2+\left(\frac{u_t}{u_t^{fail}}\right)^2+\left(\frac{\theta_r}{\theta_r^{fail}}\right)^2+\left(\frac{\theta_s}{\theta_s^{fail}}\right)^2+\left(\frac{\theta_t}{\theta_t^{fail}}\right)^2-1.\geq 0.$$

After failure the discrete element is deleted. If failure is included either one or both of the criteria may be used.

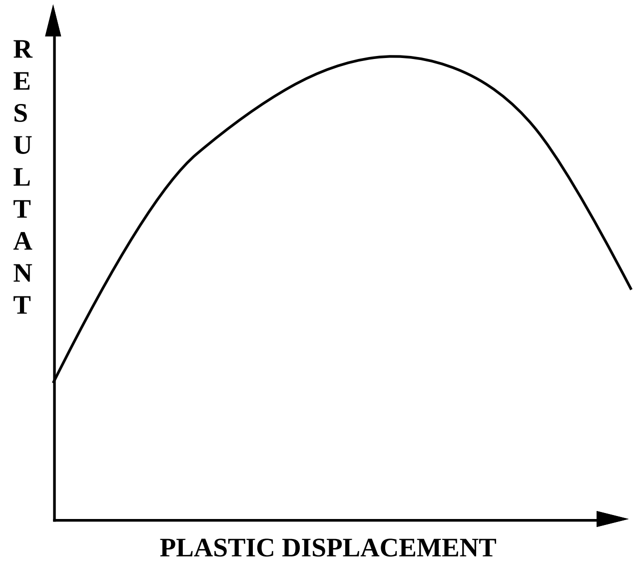


Figure 3.20. The resultant forces and moments are limited by the yield definition. The initial yield point corresponds to a plastic displacement of zero.

Material Type 69 (Side Impact Dummy Damper, SID Damper)

The side impact dummy uses a damper that is not adequately treated by the nonlinear force versus relative velocity curves since the force characteristics are dependent on the displacement of the piston.

Columns		Quantity	Format
1-10	Card 3	S_t, piston stroke. S_t must equal or exceed the length of the beam element. (See Figure 3.21 below)	E10.0
11-20		d, piston diameter	E10.0
21-30		R, default orifice radius	E10.0
31-40		h, orifice controller position	E10.0
41-50		n, number of orifices, not to exceed 15.	E10.0
51-60		K, damping constant LT.0.0: |K| is the load curve number ID defining the damping coefficient as a function of the absolute value of the relative velocity	E10.0
61-70		C, discharge coefficient	E10.0
71-80		k, stiffness coefficient if piston bottoms out	E10.0
1-10	Card 4	d_1, orifice location relative to fixed end	E10.0
11-20		d_2, orifice location relative to fixed end	E10.0
21-30		d_3, orifice location relative to fixed end	E10.0
31-40		d_4, orifice location relative to fixed end	E10.0
	.	.	
	.	.	
1-10	Card 5	d_9, orifice location relative to fixed end	E10.0
11-20		d_{10}, orifice location relative to fixed end	E10.0
21-30		d_{11}, orifice location relative to fixed end	E10.0
31-40		d_{12}, orifice location relative to fixed end	E10.0
	.	.	
	.	.	

Material Type 69 (Side Impact Dummy Damper)

Columns		Quantity	Format
71-80	Card 5	SF, scale factor on calculated force. EQ.0.0: default is set to 1.0	E10.0
1-10	Card 6	R_1, orifice radius corresponding to d_1	E10.0
11-20		R_2, orifice radius corresponding to d_2	E10.0
21-30		R_3, orifice radius corresponding to d_3	E10.0
31-40		R_4, orifice radius corresponding to d_4	E10.0
	.	.	
	.	.	
1-10	Card 7	R_9, orifice radius corresponding to d_9	E10.0
11-20		R_{10}, orifice radius corresponding to d_{10}	E10.0
21-30		R_{11}, orifice radius corresponding to d_{11}	E10.0
31-40		R_{12}, orifice radius corresponding to d_{12}	E10.0
	.	.	
	.	.	
71-80	Card7	c, linear viscous damping coefficient used after damper bottoms out either in tension or compression.	E10.0
1-10	Card 8	ρ_{fluid}, fluid density	E10.0
11-20		C_1, coefficient for linear velocity term	E10.0
21-30		C_2, coefficient for quadratic velocity term	E10.0
31-40		Load curve number ID defining force versus piston displacement, s, i.e., term $f(s+s_0)$. Compressive behavior is defined in the positive quadrant of the force displacement curve. Displacements falling outside of the defined force displacement curve are extrapolated. Care must be taken to ensure that extrapolated values are reasonable.	E10.0
41-50		Load curve number ID defining damping coefficient versus piston displacement, s, i.e., $g(s+s_0)$. Displacements falling outside the defined curve are extrapolated. Care must be taken to ensure that extrapolated values are reasonable.	E10.0

Columns	Quantity	Format
51-60	Initial displacement s_0, typically set to zero. A positive displacement corresponds to compressive behavior.	E10.0
61-70	C_3, coefficient for fluid inertia term	E10.0

Cards 4-5 give the n (<16) orifice locations d_1, d_2, d_3, d_4, ..., d_n relative to the fixed end using a (8e10.0) format. Cards 6-7 gives the corresponding radii for each orifice: if zero, the default radius, R, is used. If necessary insert blank cards.

As the damper moves, the fluid flows through the open orifices to provide the necessary damping resistance. While moving as shown in Figure 3.21, the piston gradually blocks off and effectively closes the orifices. The number of orifices and the size of their opening control the damper resistance and performance. The damping force is computed from

$$F = SF\left\{KA_pV_p\left\{\frac{C_1}{A_0^t} + C_2\left|V_p\right|\rho_{fluid}\left[\left(\frac{A_p}{CA_0^t}\right)^2 - 1\right]\right\} - f\left(s + s_0\right) + V_p g(s + s_0)\right\}$$

where K is a user defined constant or a tabulated function of the absolute value of the relative velocity, V_p is the piston velocity, C is the discharge coefficient, A_p is the piston area, A_0^t is the total open areas of orifices at time t, ρ_{fluid} is the fluid density, C_1 is the coefficient for the linear term, and C_2 is the coefficient for the quadratic term.

In the implementation, the orifices are assumed to be circular with partial covering by the orifice controller. As the piston closes, the closure of the orifice is gradual. This gradual closure is properly taken into account to insure a smooth response. If the piston stroke is exceeded, the stiffness value, k, limits further movement, i.e., if the damper bottoms out in tension or compression the damper forces are calculated by replacing the damper by a bottoming out spring and damper, k and c, respectively. The piston stroke must exceed the initial length of the beam element. The time step calculation is based in part on the stiffness value of the bottoming out spring. A typical force versus displacement curve at constant relative velocity is shown in Figure 3.22.

The factor, SF, which scales the force defaults to 1.0 and is analogous to the adjusting ring on the damper.

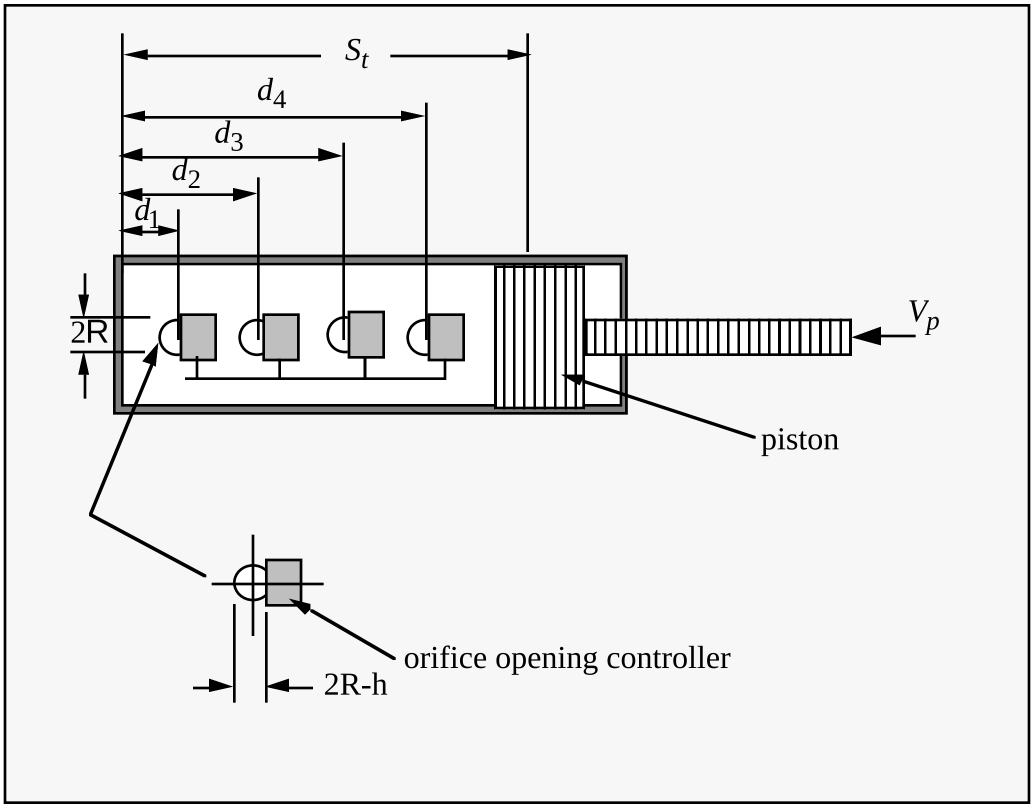


Figure 3.21. Mathematical model for the Side Impact Dummy damper.

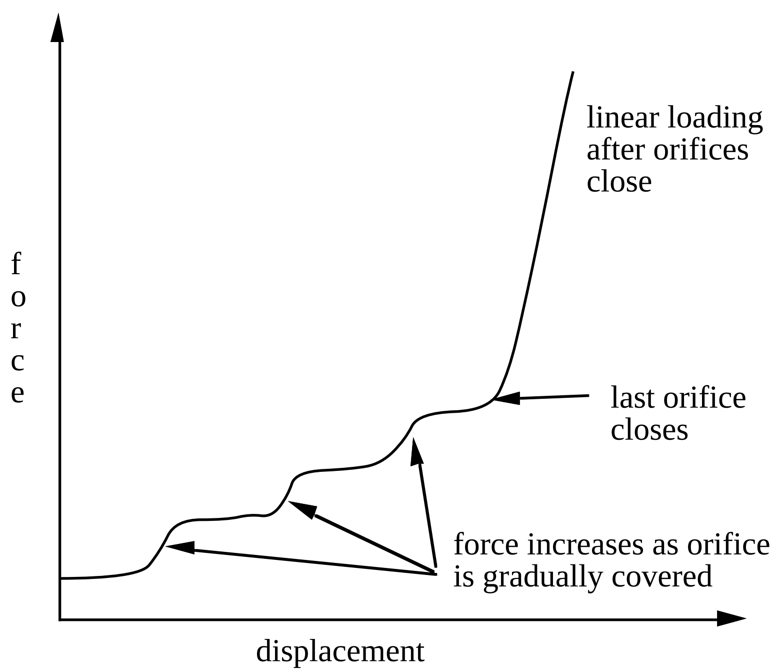


Figure 3.22. Force versus displacement as orifices are covered at a constant relative velocity. Only the linear velocity term is active.

Material Type 70 (Hydraulic/Gas Damper)

This special purpose element represents a combined hydraulic and gas-filled damper which has a variable orifice coefficient. A schematic of the damper is shown in Figure 3.23. Dampers of this type are sometimes used on buffers at the end of railroad tracks and as aircraft undercarriage shock absorbers. This material can be used only as a discrete beam element.

Columns		Quantity	Format
1-10	Card 3	C_0, length of gas column.	E10.0
11-20		n, adiabatic constant	E10.0
21-30		P_0, initial gas pressure	E10.0
31-40		P_a, atmospheric pressure	E10.0
41-50		A_p, piston cross sectional area	E10.0
51-60		K_h, hydraulic constant	E10.0
61-70		N, load curve number defining the orifice area, a_0, versus element deflection.	E10.0
71-80		F_r, return factor on orifice force. This acts as a factor on the hydraulic force only and is applied when unloading. It is intended to represent a valve that opens when the piston unloads to relieve hydraulic pressure. Set it to 1.0 for no such relief.	E10.0
1-10	Card 4	SCLF, scale factor on force. (Default = 1.0)	E10.0
11-20		Clearance (if nonzero, no tensile force develops for positive displacements and negative forces develop only after the clearance is closed.	E10.0
	Card 5	Blank	
	Card 6	Blank	
	Card 7	Blank	
	Card 8	Blank	

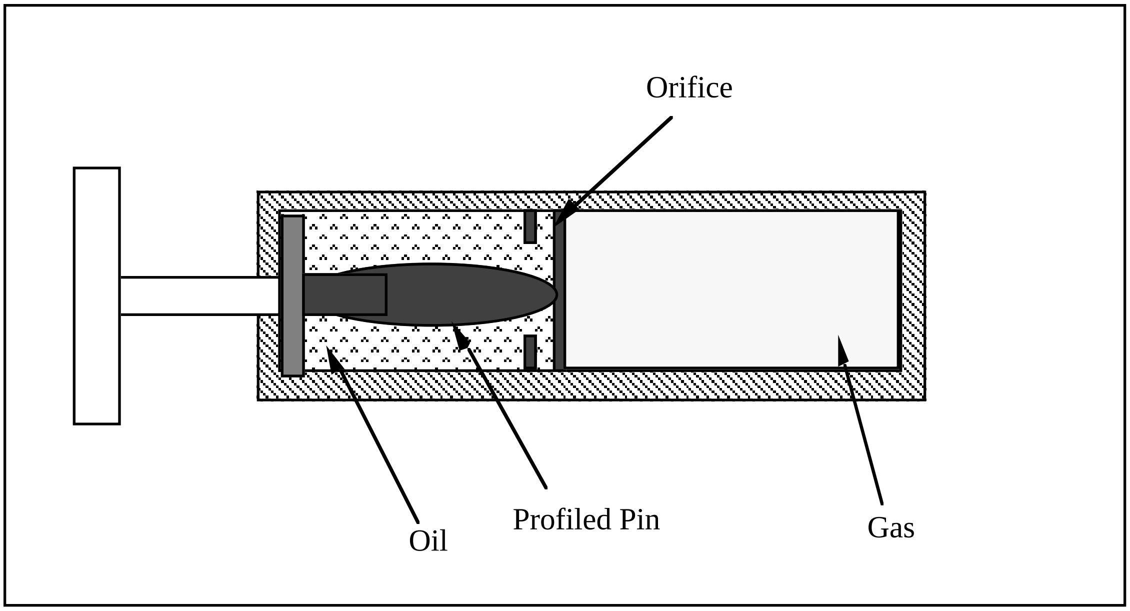


Figure 3.23. Schematic of Hydraulic/Gas damper.

As the damper is compresses two actions contribute to the force developed. First, the gas is adiabatically compressed into a smaller volume. Secondly, oil is forced through an orifice. A profiled pin may occupy some of the cross-sectional area of the orifice; thus, the orifice area available for the oil varies with the stroke. The force is assumed proportional to the square of the velocity and inversely proportional to the available area.

The equation for this element is:

$$F = SCLF \cdot \left\{ K_h \left(\frac{V}{a_0} \right)^2 + \left[P_0 \left(\frac{C_0}{C_0 - S} \right)^n - P_a \right] \cdot A_p \right\}$$

where S is the element deflection and V is the relative velocity across the element.

Material Type 71 (Cable)

This material can be used only as a discrete beam element.

Columns		Quantity	Format
1-10	Card 3	GT.0.0 : *E*, Young's modulus LT. 0.0 : Stiffness	E10.0
11-20		*n*, optional load curve ID giving stress versus strain	E10.0
1-10	Card 4	F0, initial tensile force. If F0 is defined, an offset is not needed for an initial tensile force.	E10.0
	Card 5	Blank	
	Card 6	Blank	
	Card 7	Blank	
	Card 8	Blank	

The force, *F*, generated by the cable is nonzero if and only if the cable is tension. The force is given by:

$$F = \max(F_0 + K\Delta L, 0.)$$

where ΔL is the change in length

$$\Delta L = current\ length - \left(initial\ length - offset\right)$$

and the stiffness (E > 0.0 only) is defined as:

$$K = \frac{E \cdot area}{\left(initial\ length - offset\right)}$$

Note that a constant force element can be obtained by setting:

$$F_0 > 0 \quad and \;\; K = 0$$

although the application of such an element is unknown.

The area and offset are defined on either the cross section or element cards. For a slack cable the offset should be input as a negative length. For an initial tensile force the offset should be positive.

If a load curve is specified the Young's modulus will be ignored and the load curve will be used instead. The points on the load curve are defined as engineering stress versus engineering strain, i.e., the change in lenght over the initial length. The unloading behavior follows the loading.

Material Type 72 (Concrete Damage Model)

Columns		Quantity	Format
1-10	Card 3	υ (constant Poisson's ratio model)	E10.0
11-20		Maximum principal stress failure, sigf	E10.0
21-30		Cohesion (a_0)	E10.0
31-40		Pressure hardening coefficient (a_1)	E10.0
41-50		Pressure hardening coefficient (a_2)	E10.0
51-60		Damage scaling factor b_1	E10.0
61-70		Blank	10X
71-80		Pressure hardening coefficient for failed material (a_{lf})	E10.0
1-10	Card 4	Percent reinforcement (0 - 100%)	E10.0
11-20		Elastic modulus for reinforcement, E_r	E10.0
21-30		Poisson's ratio for reinforcement, υ_r	E10.0
31-40		Initial yield stress for reinforcement	E10.0
41-50		Tangent modulus for reinforcement	E10.0
51-60		Load curve giving rate sensitivity for principal material	E10.0
61-70		Load curve giving rate sensitivity for reinforcement	E10.0
1-10	Card 5	1st tabulated value for damage function λ_1	E10.0
11-20		λ_2	.
.		.	.
.		.	.
71-80		λ_8	E10.0

Material Type 72 **(Concrete Damage Model)**

Columns		Quantity	Format
1-10	Card 6	λ_9	E10.0
.		.	.
.		.	.
.		.	.
41-50		λ_{13}	E10.0
51-60		Damage scaling factor for triaxial tensile path, b_3	E10.0
61-70		Cohesion for yield limit, a_{0y}	E10.0
71-80		Pressure hardening coefficient for yield limit a_{1y}	E10.0
1-10	Card 7	1st tabulated value of scale factor η_1	E10.0
.		.	.
.		.	.
.		.	.
71-80		η_8	E10.0
1-10	Card 8	η_9	E10.0
.		.	.
.		.	.
.		.	.
41-50		η_{13}	E10.0
51-60		Damage scaling factor for triaxial tensile path, b_2	E10.0
61-70		Pressure hardening coefficient for failed material a_{2f}	E10.0
71-80		Pressure hardening coefficient for yield limit a_{2y}	E10.0

Notes: Cohesion for failed material $a_{0f} = 0.0$

b3 must be positive or zero.

$\lambda_n < \lambda_{n+1}$. The first point must be zero.

Material Type 73 (Low Density Viscoelastic Foam)

Columns		Quantity	Format
1-10	Card 3	E, Young's modulus	E10.0
11-20		Curve number of nominal stress versus strain	E10.0
21-30		Tension cut-off stress	E10.0
31-40		Blank	
41-50		Blank	
51-60		Viscous coefficient (.05<recommended value<.50) for stress oscillations and shock waves.	E10.0
61-70		Blank	
71-80		Failure option after cutoff stress is reached. EQ.0: tensile stress remains at cut-off value EQ.1: tensile stress is reset to zero.	E10.0
1-10	Card4	Bulk viscosity activation flag. EQ.0.0: no bulk viscosity (recommended) EQ.1.0: bulk viscosity active	E10.0
1-10	Card5	Stiffness coefficient for contact interface stiffness. EQ.0.0: Maximum slope in stress vs. strain curve is used. When the maximum slope is taken for the contact, the time step size for this material is reduced for stability. In some cases Δt may be significantly smaller, and defining a reasonable stiffness is recommended.	E10.0
11-20		Load curve ID if constants $\beta\iota$ are determined via a least squares fit. This relaxation curve is shown in Figure 3.25. This model ignores the constant stress	E10.0
21-30		BSTART. In the fit, β_1 is set to zero, β_2 is set to BKSTART, β_3 is 10 times $\beta\kappa_2$, β_4 is 100 times greater than β_3, and so on. If zero, BSTART= .01.	E10.0
31-40		TRAMP, optional ramp time for loading.	E10.0

Material Type 73 (Low Density Viscoelastic Foam)

Columns		Quantity	Format
41-50		Number of terms in fit. If zero, the default is 6. Currently, the maximum number is set to 6. Values of 2 are 3 are recommended, since each term used adds significantly to the cost. Caution should be exercised when taking the results from the fit. Preferably, all generated coefficients should be positive. Negative values may lead to unstable results. Once a satisfactory fit has been achieved it is recommended that the coefficients which are written into the output file be input in future runs.	E10.0
1-10	Card 6	Use reference geometry to initial stresses. EQ.0.0: off EQ.1.0: on	E10.0

If the viscous effects are active and if a load curve ID is not defined on card 4 giving the relaxation data, then define the following input on cards 7 and 8. If not leave cards 7 and 8 blank. Up to 6 pairs may be defined.

1-10	Card 7	G_1, Maxwell consant, optional	E10.0
11-20		β_1, decay constant, optional	E10.0
21-30		G_2 , Maxwell consant	E10.0
31-40		β_2, decay constant	E10.0
41-50		G_3, Maxwell consant	E10.0
51-60		β_3 , decay constant	E10.0
61-70		G_4, Maxwell consant	E10.0
71-80		β_4, decay constant	E10.0
1-10	Card 8	G_5, Maxwell consant, optional	E10.0
11-20		β_5, decay constant, optional	E10.0
21-30		G_6 , Maxwell consant	E10.0
31-40		β_6, decay constant	E10.0

This viscoelastic foam model is available to model highly compressible viscous foams. The hyperelastic formulation of this models follows that of material 57.

Rate effects are accounted for through linear viscoelasticity by a convolution integral of the form

$$\sigma_{ij}^{r} = \int_0^t g_{ijkl}(t-\tau)\frac{\partial \varepsilon_{kl}}{\partial \tau}d\tau$$

where $g_{ijkl}(t-\tau)$ is the relaxation function. The stress tensor, σ_{ij}^{r}, augments the stresses determined from the foam, σ_{ij}^{f}; consequently, the final stress, σ_{ij}, is taken as the summation of the two contributions:

$$\sigma_{ij} = \sigma_{ij}^{f} + \sigma_{ij}^{r}.$$

Since we wish to include only simple rate effects, the relaxation function is represented by up to six terms of the Prony series:

$$g(t) = \alpha_0 + \sum_{m=1}^{N}\alpha_m e^{-\beta t}$$

This model is effectively a Maxwell fluid which consists of a dampers and springs in series. The formulation is performed in the local system of principal stretches where only the principal values of stress are computed and triaxial coupling is avoided. Consequently, the one-dimensional nature of this foam material is unaffected by this addition of rate effects. The addition of rate effects necessitates 42 additional history variables per integration point. The cost and memory overhead of this model comes primarily from the need to "remember" the local system of principal stretches and the evaluation of the viscous stress components..

Material Type 75 (Bilkhu/Dubois Foam Model)

This model uses uniaxial and triaxial test data to provide a more realistic treatment of crushable foam. The Poisson's ratio is set to zero for the elastic response.

Columns		Quantity	Format
1-10	Card 3	Young's modulus (E)	E10.0
11-20		Load curve ID giving pressure for plastic yielding versus volumetric strain. See Figure 3.24.	E10.0
21-30		Load curve ID giving uniaxial yield stress versus volumetric strain. See Figure 3.24.	E10.0
31-40		Viscous coefficient (.05<recommended value <.50)	E10.0
41-50		Pressure cutoff. If zero, the default is set to one-tenth of p_0, the yield pressure corresponding to a volumetric strain of zero.	E10.0
51-60		Pressure cutoff as a fraction of pressure yield value the default is set to 0.1 if non-zero this will override the pressure cutoff value given in colums 41-50	E10.0
61-70		Cutoff value for uniaxial tensile stress if non-zero this will override the pressure cutoff values in columns 41-60	E10.0
71-80		Cutoff value for uniaxial tensile stress as a fraction of the uniaxial compressive yield strength, if non-zero this overrides pressure cutoff values in colums 41-60	E10.0
1-10	Card 4	Load curve ID giving a scale factor for the previous yield curves, dependent upon the volumetric plastic strain.	E10.0
1-10	Card 5	Poisson coefficient, will apply to both elastic and plastic deformation.	E10.0
	Card 6	Blank	
	Card 8	Blank	

The volumetric strain is defined in terms of the relative volume, V, as:

$$\gamma = -\ln(V)$$

In defining the curves the stress and strain pairs should be positive values starting with a volumetric strain value of zero.

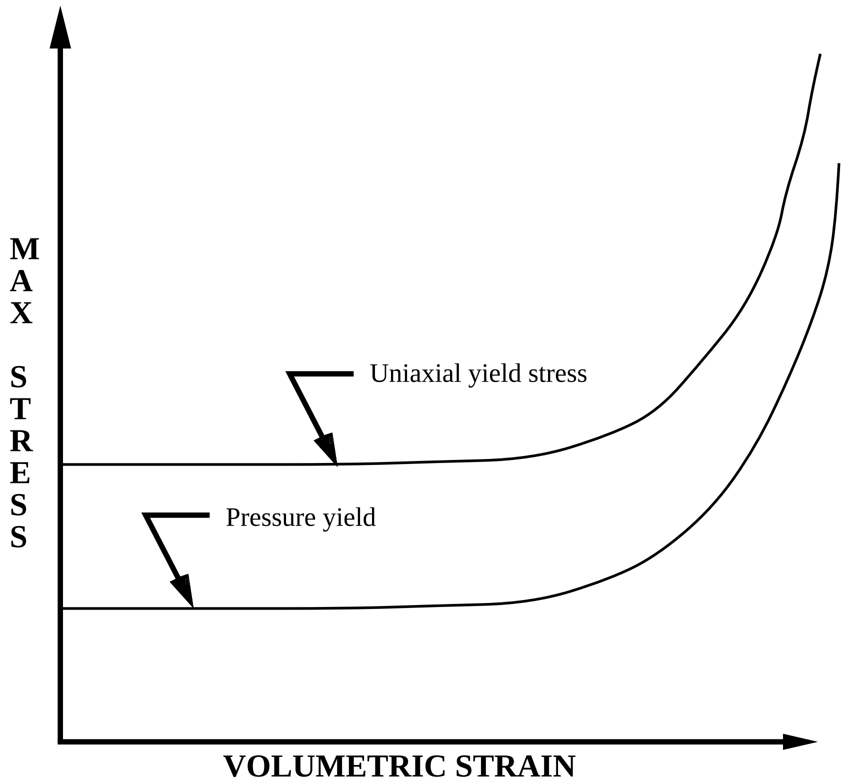


Figure 3.24. Behavior of crushable foam. Unloading is elastic.

Material Type 76 (General Viscoelastic)

Columns		Quantity	Format
1-10	Card 3	K, constant elastic bulk modulus. If the bulk behavior is viscoelastic, then this modulus is used in determining the contact interface stiffnesses only.	E10.0
1-10	Card 4	G_1, Maxwell consant. If a relaxation curve is given on card 6 below, cards 4 and 5 may be blank.	E10.0
11-20		β_1, decay constant	E10.0
21-30		G_2 , Maxwell consant	E10.0
31-40		β_2, decay constant	E10.0
41-50		G_3, Maxwell consant	E10.0
51-60		β_3 , decay constant	E10.0
61-70		G_4, Maxwell consant	E10.0
71-80		β_4, decay constant	E10.0
1-10	Card 5	G_5, Maxwell consant	E10.0
11-20		β_5, decay constant	E10.0
21-30		G_6 , Maxwell consant	E10.0
31-40		β_6, decay constant	E10.0
1-10	Card 6	Load curve ID if constants, G_i, and β_i are determined via a least squares fit. This relaxation curve is shown in Figure 3.25.	E10.0
11-20		Number of terms in fit. If zero the default is 6. Currently, the maximum number is set to 6.	E10.0
21-30		BSTART. In the fit, β_1 is set to zero, β_2 is set to BSTART, β_3 is 10 times β_2 , β_4 is 100 times greater than β_3 , and so on. If zero, BSTART= .01.	E10.0
31-40		TRAMP, optional ramp time for loading.	E10.0

Material Type 76 (General Viscoelastic)

Columns		Quantity	Format

If there is no volumetric relaxation then insert two blank cards:

Card 7 Blank

Card 8 Blank

For volumetric relaxation define the following information:

41-50	Card 6	Load curve ID if constants, K_i, and $\beta\kappa_i$ are determined via a least squares fit. This relaxation curve is shown in Figure 3.25.	E10.0
51-60		Number of terms in fit. If zero the default is 6. Currently, the maximum number is set to 6.	E10.0
61-70		Number of terms in fit. If zero, the default is 6. Currently, the maximum number is 6. Values less than 6, possibly 3-5 are recommended, since each term used adds significantly to the cost. Caution should be exercised when taking the results from the fit. Always check the results of the fit in the output file. Preferably, all generated coefficients should be positive-they generally will be. Negative values may lead to unstable results. Once a satisfactory fit has been achieved it is recommended that the coefficients which are written into the output file be input in future runs.	E10.0
61-70		Blank	E10.0
71-80		TRAMP, optional ramp time for loading.	E10.0
1-10	Card 7	K_1, Maxwell consant. If a relaxation curve is given on card 6 above, cards 7 and 8 may be blank.	E10.0
11-20		$\beta\kappa_1$, decay constant	E10.0
21-30		K_2 , Maxwell consant	E10.0
31-40		$\beta\kappa_2$, decay constant	E10.0
41-50		K_3, Maxwell consant	E10.0
51-60		$\beta\kappa_3$, decay constant	E10.0
61-70		K_4, Maxwell consant	E10.0
71-80		$\beta\kappa_4$, decay constant	E10.0

Columns		Quantity	Format
1-10	Card 8	K_5, Maxwell consant	E10.0
11-20		$\beta\kappa_5$, decay constant	E10.0
21-30		K_6 , Maxwell consant	E10.0
31-40		$\beta\kappa_6$, decay constant	E10.0

Rate effects are taken into accounted through linear viscoelasticity by a convolution integral of the form:

$$\sigma_{ij} = \int_0^t g_{ijkl}(t-\tau)\frac{\partial \varepsilon_{kl}}{\partial \tau}\,d\tau$$

where $g_{ijkl}(t-\tau)$ is the relaxation function.

If we wish to include only simple rate effects for the deviatoric stresses, the relaxation function is represented by six terms from the Prony series:

$$g(t) = \sum_{m=1}^{N} G_m e^{-\beta_m t}$$

We characterize this in the input by shear modulii, G_i, and decay constants, β_i. An arbitrary number of terms, up to 6, may be used when applying the viscoelastic model.

For volumetric relaxation, the relaxation function is also represented by the Prony series in terms of bulk modulii:

$$k(t) = \sum_{m=1}^{N} K_m e^{-\beta_{k_m} t}$$

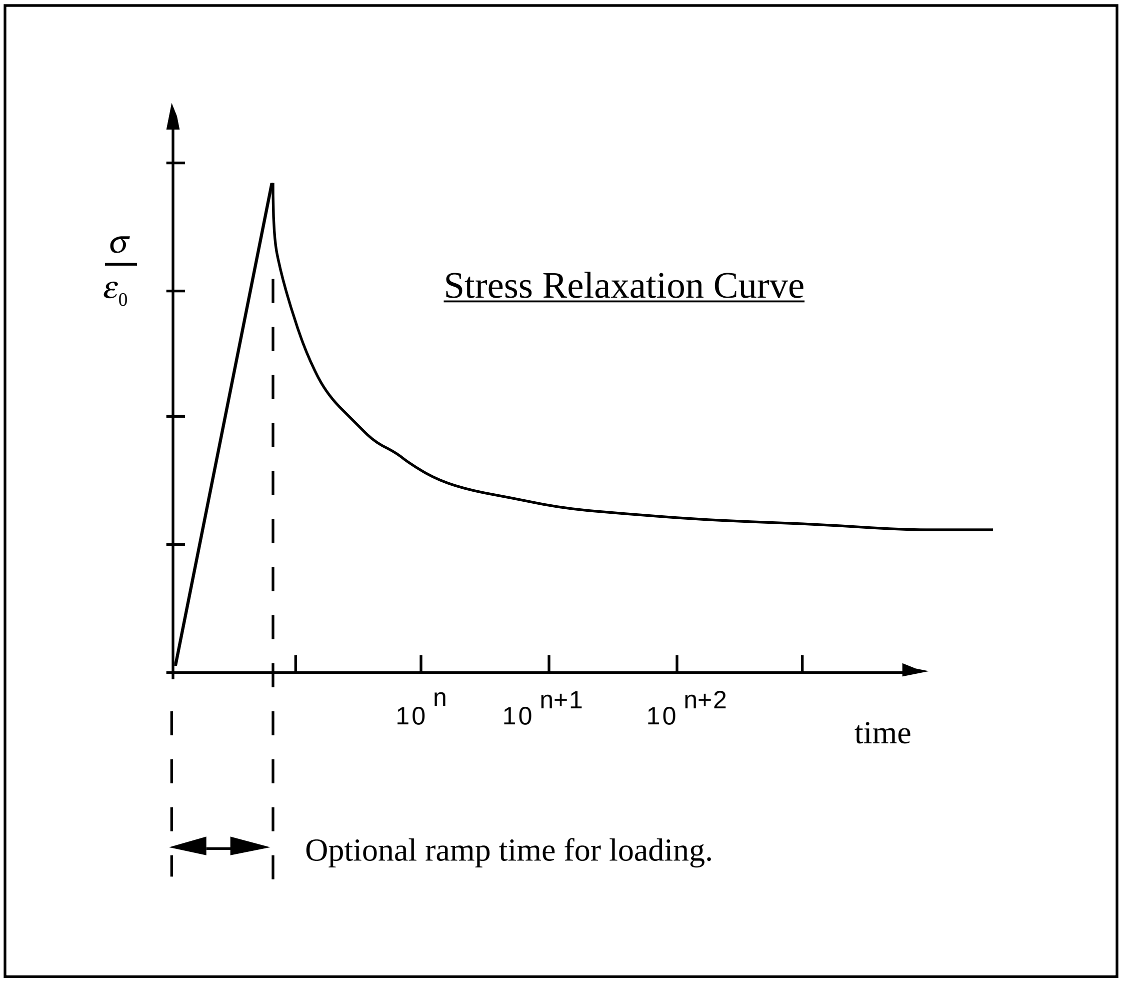


Figure 3.25. Relaxation curve. This curve defines stress versus time where time is defined on a logarithmic scale. For best results, the points defined in the load curve should be equally spaced on the logarithmic scale. Furthermore, the load curve should be smooth and defined in the positive quadrant. If nonphysical values are determined by least squares fit, LS-DYNA will terminate with an error message after the initialization phase is completed. If the ramp time for loading is included, then the relaxation which occurs during the loading phase is taken into account. This effect may or may not be important.

Material Type 77 (Hyperviscoelastic Rubber)

This material model provides a general hyperelastic or Ogden rubber model combined with linear viscoelasticity as outlined by Christensen [1980]

Columns		Quantity	Format
1-10	Card 3	υ, Poisson's ratio (> .49 is recommended-smaller values may not work.and should not be used)	E10.0
11-20		n, order of fit to experimental data defined as load curve. Currently, for Ogden n<5. For the strain enrgy functionalal, n=1 for C_{10} and C_{01}, n=2 for C_{10}, C_{01}, C_{11}, C_{20}, and C_{02}, and n=3 for C_{10}, C_{01}, C_{11}, C_{20}, C_{02}, and C_{30}.	E10.0
21-30		Formumlation EQ.0.0: strain energy functional EQ.1.0: Ogden model.	E10.0
41-50		Number of Prony series terms in fit. If zero, the default is 6. Currently, the maximum number is 6. Values less than 6, possibly 3-5 are recommended, since each term used adds significantly to the cost. Caution should be exercised when taking the results from the fit. Always check the results of the fit in the output file. Preferably, all generated coefficients should be positive. Negative values may lead to unstable results. Once a satisfactory fit has been achieved it is recommended that the coefficients which are written into the output file be input in future runs.	E10.0

If n>0 then a least square fit is computed with uniaxial data. Card 4 should contain the following information. Also see Figure 3.12.

1-10	Card 4	Specimen gauge length	E10.0
11-20		Specimen width	E10.0
21-30		Specimen thickness	E10.0
31-40		Load curve ID giving the force versus actual change in the gauge length.	E10.0
41-50		Type of experimental data. EQ.0.0: uniaxial data EQ.1.0: biaxial data (Ogden only.)	E10.0

Columns		Quantity	Format
51-60		Load curve ID if constants $\beta\iota$ are determined via a least squares fit. This relaxation curve is shown in Figure 3.25. This model ignores the constant stress	E10.0
61-70	Card 4	Blank	E10.0
71-80		TRAMP, optional ramp time for loading.	E10.0

For the general hyperelastic model define the following input on cards 5 and 6:

The input on card 5 is required if and only if n=0.

Columns		Quantity	Format
1-10	Card 5	C_{10}, see definition below	E10.0
11-20		C_{01}	E10.0
21-30		C_{11}	E10.0
31-40		C_{20}	E10.0
41-50		C_{02}	E10.0
51-60		C_{30}	E10.0
	Card 6	Blank	E10.0
	.	.	.
	.	.	.

For the Ogden model define the following input on cards 5 and 6. Up to eight pairs of constants may be defined (required if and only if n=0).

Columns		Quantity	Format
1-10	Card 5	μ_1, see definition below	E10.0
11-20		μ_2	E10.0
21-30		μ_3	E10.0
31-40		μ_4	E10.0
41-50		μ_5	E10.0
51-60		μ_6	E10.0
61-70		μ_7	E10.0
71-80		μ_8	E10.0
1-10	Card 6	α_1, see definition below	E10.0
11-20		α_2	E10.0

Columns		Quantity	Format
21-30		α_3	E10.0
31-40		α_4	E10.0
41-50		α_5	E10.0
51-60		α_6	E10.0
61-70		α_7	E10.0
71-80		α_8	E10.0

If the viscous effects are active and if a load curve ID is not defined on card 4 giving the relaxation data, then define the following input on cards 7 and 8. If not leave cards 7 and 8 blank. Up to 6 pairs may be defined.

1-10	Card 7	G_1, Maxwell consant, optional	E10.0
11-20		β_1, decay constant, optional	E10.0
21-30		G_2 , Maxwell consant	E10.0
31-40		β_2, decay constant	E10.0
41-50		G_3, Maxwell consant	E10.0
51-60		β_3 , decay constant	E10.0
61-70		G_4, Maxwell consant	E10.0
71-80		β_4, decay constant	E10.0
1-10	Card 8	G_5, Maxwell consant, optional	E10.0
11-20		β_5, decay constant, optional	E10.0
21-30		G_6 , Maxwell consant	E10.0
31-40		β_6, decay constant	E10.0

Rubber is generally considered to be fully incompressible since the bulk modulus greatly exceeds the shear modulus in magnitude. To model the rubber as an unconstrained material a hydrostatic work term, $W_H(J)$, is included in the strain energy functional which is function of the relative volume, J , [Ogden, 1984]:

$$W(J_1, J_2, J) = \sum_{p,q=0}^{n} C_{pq} (J_1 - 3)^p (J_2 - 3)^q + W_H(J)$$

$$J_1 = I_1 I_3^{-1/3}$$

$$J_2 = I_2 I_3^{-2/3}$$

In order to prevent volumetric work from contributing to the hydrostatic work the first and second invarients are modified as shown. This procedure is described in more detail by Sussman and Bathe [1987]. For the Ogden model the energy equation is given as:

$$W^* = \sum_{i=1}^{3} \sum_{j=1}^{n} \frac{\mu_j}{\alpha_j} \left(\lambda_i^{*\alpha_j} - 1 \right) + \frac{1}{2} K (J-1)^2$$

where the asterisk (*) indicates that the volumetric effects have be eliminated from the principal stretches, λ_j^*. See Ogden [1984] for more details.

Rate effects are taken into accounted through linear viscoelasticity by a convolution integral of the form:

$$\sigma_{ij} = \int_0^t g_{ijkl}(t - \tau) \frac{\partial \varepsilon_{kl}}{\partial \tau} d\tau$$

or in terms of the second Piola-Kirchhoff stress, S_{ij}, and Green's strain tensor, E_{ij},

$$S_{ij} = \int_0^t G_{ijkl}(t - \tau) \frac{\partial E_{kl}}{\partial \tau} d\tau$$

where $g_{ijkl}(t-\tau)$ and $G_{ijkl}(t-\tau)$ are the relaxation functions for the different stress measures. This stress is addedto the stress tensor determined from the strain energy functional.

If we wish to include only simple rate effects, the relaxation function is represented by six terms from the Prony series:

$$g(t) = \alpha_0 + \sum_{m=1}^{N} \alpha_m e^{-\beta t}$$

given by,

$$g(t) = \sum_{i=1}^{n} G_i e^{-\beta_i t}$$

This model is effectively a Maxwell fluid which consists of a dampers and springs in series. We characterize this in the input by shear modulii, G_i, and decay constants, β_i. The viscoelastic behavior is optional and an arbitrary number of terms may be used.

The Mooney-Rivlin rubber model is obtained by specifying n=1. In spite of the differences in formulations with Model 27, we find that the results obtained with this model are nearly identical with those of 27 as long as large values of Poisson's ratio are used.

Material Type 78 (Soil/Concrete)

Columns		Quantity	Format
1-10	Card 3	Shear modulus	E10.0
11-20		Bulk modulus	E10.0
21-30		Load curve ID for pressure versus volumetric strain The pressure versus volumetric strain curve is defined for compression only. The sign convention requires that both pressure and compressive strain be defined as positive values where the compressive strain is taken as the negative value of the natural logrithm of the relative volume.	E10.0
31-40		Load curve ID for deviatoric yield versus pressure GT.0: Von Mises stress versus pressure LT.0: second stress invarient, J_2, versus pressure This curve must be defined.	E10.0
41-50		Load curve ID for plastic strain at which fracture begins versus pressure. Define if b>0.	E10.0
51-60		Load curve ID for plastic strain at which residual strength is reached versus pressure. Define if b>0.	E10.0
61-70		Pressure cutoff for tensile fracture	E10.0
71-80		Output option for plastic strain EQ.0: volumetric plastic strain EQ.1: deviatoric plastic strain	E10.0
1-10	Card 4	Residual strength factor after cracking, b	E10.0
11-20		Flag to specify failure of element EQ.0: no failure EQ.1: after cut-off is reached tension is no longer carried.	E10.0
	Card 5	Blank	
	Card 6	Blank	
	Card 7	Blank	
	Card 8	Blank	

Pressure is positive in compression. Volumetric strain is defined as the natural log of the relative volume and is *positive* in compression where the relative volume, V, is the ratio of the current volume to the initial volume. The tabulated data should be given in order of increasing compression. If the pressure drops below the cutoff value specified, it is reset to that value and the deviatoric stress state is eliminated.

If the load curve ID is provided as a positive number, the deviatoric perfectly plastic pressure dependent yield function ϕ, is described in terms of the second invariant, J_2, the pressure, p, and the tabulated load curve, $F(p)$, as

$$\phi = \sqrt{3J_2} - F(p) = \sigma_y - F(p)$$

where J_2 is defined in terms of the deviatoric stress tensor as:

$$J_2 = \frac{1}{2} S_{ij} S_{ij}$$

assuming that. If the ID is given as negative then the yield function becomes:

$$\phi = J_2 - F(p)$$

being the deviatoric stress tensor.

If cracking is invoked by setting the residual strength factor on card 4 to a value between 0.0 and 1.0, the yield stress is multiplied by a factor f which reduces with plastic stain according to a trilinear law as shown in Figure 3.26.

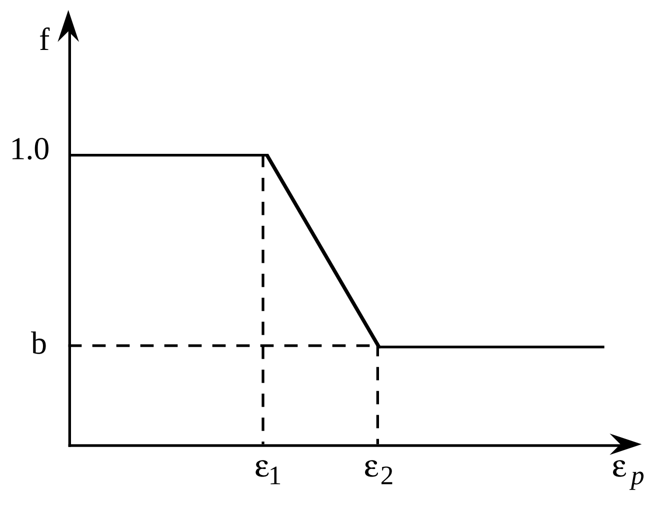


Figure 3.26. Strength reduction factor.

b = residual strength factor

ε_1 = plastic stain at which cracking begins.

ε_2 = plastic stain at which residual strength is reached.

ε_1 and ε_2 are tabulated function of pressure that are defined by load curves (see Figure 3.27). The values on the curves are pressure versus strain and should be entered in order of increasing pressure. The strain values should always increase monotonically with pressure.

By properly defining the load curves, it is possible to obtain the desired strength and ductility over a range of pressures. See Figure 3.26.

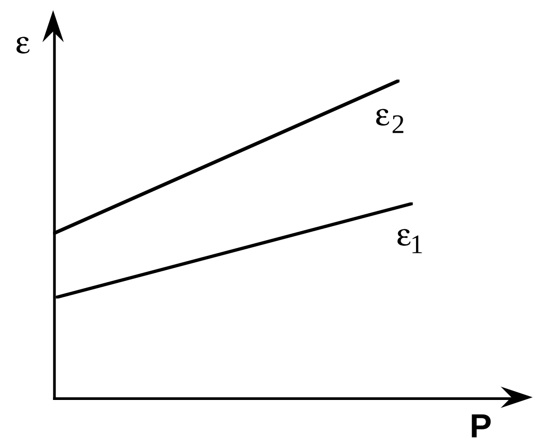


Figure 3.27. Cracking strain versus pressure.

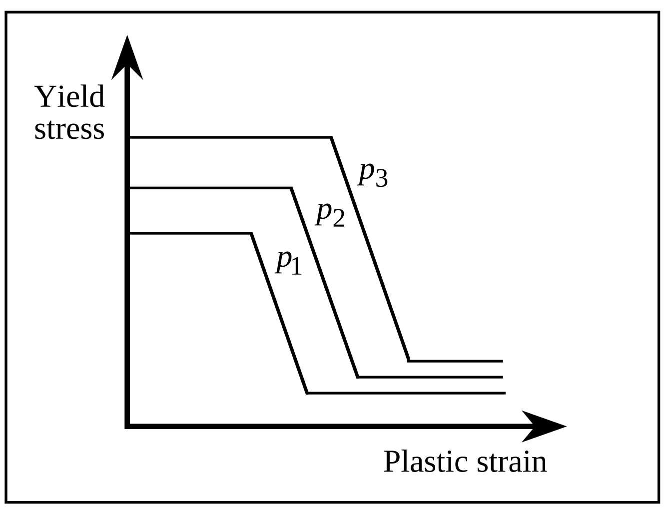


Figure 3.28.

Material Type 79 (Hysteretic Soil)

This model is a nested surface model with five superposed “layers” of elasto-perfectly plastic material, each with its own elastic modulii and yield values. Nested surface models give hysteretic behavior, as the different “layers” yield at different stresses.

Columns		Quantity	Format
1-10	Card 3	K_o, bulk modulus at the reference pressure (this should give a sensible Poisson’s ratio with G)	E10.0
11-20		P_o, cut-off/datum pressure (must be 0≤ i.e. tensile). Below this pressure, stiffness and strength disappears; this is also the “zero” pressure for pressure-varying properties.	E10.0
21-30		b, exponent for pressure-sensitive moduli: $G = G_0(p - p_o)^b$ $K = K_0(p - p_o)^b$ b, must lie in the range $0 \le b < 1$. Values close to 1 are not recommended because the pressure becomes indeterminate.	E10.0
31-40		Yield function constant a_0 (Default = 1.0)	E10.0
41-50		Yield function constant a_1 (Default = 0.0)	E10.0
51-60		Yield function constant a_2 (Default = 0.0)	E10.0
61-70		fd, damping factor. Must be in the range $0 \le fd \le 1$ EQ.0: no damping EQ.1: maximum damping	E10.0
71-80	Reference pressure for input data. Pref.		E10.0

Cards 4 and 5 define a shear strain, shear stress curve.

Columns		Quantity	Format
1-10	Card 4	γ_1, shear strain	E10.0
11-20		γ_2, shear strain	E10.0
21-30		γ_3, shear strain	E10.0
31-40		γ_4, shear strain	E10.0

Columns		Quantity	Format
41-50		γ_5, shear strain	E10.0
1-10	Card 5	τ_1, shear stress at γ_1	E10.0
11-20		τ_2, shear stress at γ_2	E10.0
21-30		τ_3, shear stress at γ_3	E10.0
31-40		τ_4, shear stress at γ_4	E10.0
41-50		τ_5, shear stress at γ_5	E10.0
	Card 6	Blank	
	Card 7	Blank	
	Card 8	Blank	

The constants a_0, a_1, a_2 govern the pressure sensitivity of the yield stress. Only the ratios between these values are important - the absolute stress values are take from the stress-strain curve.

The stress strain pairs (γ_1, τ_1), ... (γ_5, τ_5) define a shear stress versus shear strain curve. The first point on the curve is assumed by default to be (0,0) and does not need to be entered. The slope of the curve must decrease with increasing γ. Not all five points need be to be defined. This curves applies at the reference pressure; at other pressures the curve varies according to a_0, a_1, and a_2 as in the soil and crushable foam model, Material 5.

The elastic moduli G and K are pressure sensitive.

$$G = G_0 \left(p - p_o\right)^b$$
$$K = K_0 \left(p - p_o\right)^b$$

where G_0 and K_0 are the input values, p is the current pressure, p_0 the cut-off or reference pressure (must be zero or negative). If p attempts to fall below p_0 (i.e., more tensile) the shear stresses are set to zero and the pressure is set to p_0. Thus, the material has no stiffness or strength in tension. The pressure in compression is calculated as follows:

$$p = \left[-K_0 \ln(V)\right]^{1/1-b}$$

where V is the relative volume, i.e., the ratio between the original and current volume.

Material Type 80 (Ramberg-Osgood Plasticity)

Columns		Quantity	Format
1-10	Card 3	Reference shear strain (γ_y)	E10.0
1-10	Card 4	Reference shear stress (τ_y)	E10.0
1-10	Card 5	Stress coefficient (α)	E10.0
1-10	Card 6	Stress exponent (r)	E10.0
1-10	Card 7	Elastic bulk modulus	E10.0

The Ramberg-Osgood equation is an empirical constitutive relation to represent the one-dimensional elastic-plastic behavior of many materials, including soils. This model allows a simple rate independent representation of the hysteretic energy dissipation observed in soils subjected to cyclic shear deformation. For monotonic loading, the stress-strain relationship is given by:

$$\frac{\gamma}{\gamma_y} = \frac{\tau}{\tau_y} + \alpha \left| \frac{\tau}{\tau_y} \right|^r \quad if \quad \gamma \geq 0$$

$$\frac{\gamma}{\gamma_y} = \frac{\tau}{\tau_y} - \alpha \left| \frac{\tau}{\tau_y} \right|^r \quad if \quad \gamma < 0$$

where γ is the shear and τ is the stress. The model approaches perfect plasticity as the stress exponent $r \to \infty$. These equations must be augmented to correctly model unloading and reloading material behavior. The first load reversal is detected by $\gamma\dot{\gamma} < 0$. After the first reversal, the stress-strain relationship is modified to

$$\frac{(\gamma - \gamma_0)}{2\gamma_y} = \frac{(\tau - \tau_0)}{2\tau_y} + \alpha \left| \frac{(\tau - \tau_0)}{2\tau_y} \right|^r \quad if \quad \gamma \geq 0$$

$$\frac{(\gamma - \gamma_0)}{2\gamma_y} = \frac{(\tau - \tau_0)}{2\tau_y} - \alpha \left| \frac{(\tau - \tau_0)}{2\tau_y} \right|^r \quad if \quad \gamma < 0$$

where γ_0 and τ_0 represetn the values of strain and stress at the point of load reversal. Subsequent load reversals are detected by $(\gamma - \gamma_0)\dot{\gamma} < 0$.

Material Type 80 (Ramberg-Osgood)

The Ramberg-Osgood equations are inherently one-dimensional and are assumed to apply to shear components. To generalize this theory to the multidimensional case, it is assumed that each component of the deviatoric stress and deviatoric tensorial strain is independently related by the one-dimensional stress-strain equations. A projection is used to map the result back into deviatoric stress space if required. The volumetric behavior is elastic, and, therefore, the pressure p is found by

$$p = -K\varepsilon_v$$

where ε_v is the volumetric strain.

Material Type 81 (Plastic With Damage)

Columns		Quantity	Format
1-10	Card 3	Young's modulus	E10.0
11-20		Strain rate parameter, C	E10.0
21-30		Strain rate parameter, p	E10.0
31-40		Formulation for rate effects: EQ.0.0: Scale yield stress (default) EQ.1.0: Viscoplastic formulation	E10.0
1-10	Card 4	Poisson's ratio	E10.0
1-10	Card 5	Yield stress	E10.0
11-20		Load curve ID or Table ID. The load curve ID defines effective stress versus effective plastic strain. Cards 7 and 8 are ignored with this option. The table ID, see Figure 3.9, defines for each strain rate value a load curve ID giving the stress versus effectiveplastic strain for that rate. The stress versus effective plastic strain curve for the lowest value of strain rate is used if the strain rate falls below the minimum value. Likewise, the stress versus effective plastic strain curve for the highest value of strain rate is used if the strain rate exceeds the maximum value. The strain rate parameters on card 3, the curve ID on card 6, and cards 7 and 8 are ignored if a Table ID is defined.	E10.0
1-10	Card 6	Tangent modulus, ignored if the stress-strain curve is defined below	E10.0
11-20		Plastic strain at failure	E10.0
21-30		Time step size for automatic element deletion	E10.0
31-40		Load curve number to scale yield stress to account for strain rate effects.	E10.0
41-50		Plastic strain at rupture	E10.0
1-80	Card 7	Effective plastic strain values (define up to 8 points)	8E10.0
1-80	Card 8	Corresponding yield stress values	8E10.0

The stress strain behavior may be treated by a bilinear stress strain curve by defining the tangent modulus. Alternately, a stress versus effective plastic strain curve (Card 5, Columns 11-20) similar to that shown in Figure 3.8 can be used. If eight point are insufficient, a load curve may be used with an arbitrary number of points. The cost is roughly the same for either approach. The most general approach is to used the table definition, (Card 5, Columns 11-20) discussed below.

Three options to account for strain rate effects are possible.

I. Strain rate may be accounted for using the Cowper and Symonds model which scales the yield stress with the factor

$$1+\left(\frac{\dot{\varepsilon}}{C}\right)^{1/p}$$

where $\dot{\varepsilon}$ is the strain rate.

II. For complete generality a load curve (Card 5) to scale the yield stress may be input instead. In this curve the scale factor versus strain rate is defined.

III. If different stress versus strain curves can be provided for various strain rates, the option using the reference to a table (Card 5, Columns 11-20) can be used. Then the table input in Section 22 (Load Curve/Table Definitions) has to be used. See Figure 3.9.

A fully viscoplastic formulation is optional which incorporates the different options above within the yield surface. An additional cost is incurred over the simple scaling but the improvement is results can be dramatic.

Material Type 83 (Fu-Chang's Foam with Rate Effects)

This model allows rate effects to be modelled in low and medium density foams, see Figure 3.29. Hysteretic unloading behavior in this model is a function of the rate sensitivity with the most rate sensitive foams providing the largest hystersis and visa versa. The unified constitutive equations for foam materials by Fu-Chang [1995] provides the basis for this model. The mathematical description given below is excerpted from the reference. Further improvements have been incorporated based on work by Hirth, Du Bois, and Weimar [1998]. Their improvements permit: load curves generated by drop tower test to be directly input, a choice of principal or volumetric strain rates, load curves to be defined in tension, and the volumetric behavior to be specified by a load curve.

Columns		Quantity	Format
1-10	Card 3	E, Young's modulus for tensile strains	E10.0
11-20		Stiffness coefficient for contact interface stiffness. EQ.0.0: Maximum slope in stress vs. strain curve is used. When the maximum slope is taken for the contact, the time step size for this material is reduced for stability. In some cases Δt may be significantly smaller, and defining a reasonable stiffness is recommended.	E10.0
21-30		Tension cut-off stress	E10.0
31-40		Failure option after cutoff stress is reached. EQ.0: tensile stress remains at cut-off value EQ.1: tensile stress is reset to zero.	E10.0
41-50		Viscous coefficient (.05<recommended value<.50) for stress oscillations and shock waves.	E10.0
51-60		Optional table ID providing stress-strain data as a function of strain rate. If the table ID is provided, cards 5 and 6 may be left blank and the fit will be done internally.	E10.0
1-10	Card 4	Bulk viscosity activation flag. EQ.0.0: no bulk viscosity (recommended) EQ.1.0: bulk viscosity active	E10.0
11-20		Strain rate flag (see comment below): EQ.0.0: true constant strain rate, EQ.1.0: engineering strain rate.	E10.0

Columns		Quantity	Format
21-30		Strain rate evaluation flag: EQ.0.0: first principal direction, EQ.1.0: principal strain rates for each principal direction, EQ.2.0: volumetric strain rate.	E10.0
31-40		Tensile stress evaluation: EQ.0.0: linear in tension. EQ.1.0: input via load curves with the tensile response corresponds to negative values of stress and strain.	E10.0
41-50		Optional load curve ID defining pressure versus volumetric strain.	E10.0
1-10	Card 5	D_0, material constant, see equations below.	E10.0
11-20		n_0, material constant, see equations below.	E10.0
21-30		n_1, material constant, see equations below.	E10.0
31-40		n_2, material constant, see equations below.	E10.0
41-50		n_3, material constant, see equations below.	E10.0
51-60		c_0, material constant, see equations below.	E10.0
61-70		c_1, material constant, see equations below.	E10.0
71-80		c_2, material constant, see equations below.	E10.0
1-10	Card 6	c_3, material constant, see equations below.	E10.0
11-20		c_4, material constant, see equations below.	E10.0
21-30		c_5, material constant, see equations below.	E10.0
31-40		a_{ij}, material constant, see equations below.	E10.0
41-50		s_{ij}, material constant, see equations below.	E10.0
51-60		Ratemin, minimum strain rate of interest.	E10.0
61-70		Ratemax, maximum strain rate of interest.	E10.0
	Card 7	Blank	
	.	.	
	.	.	
	.	.	
	Card 8	Blank	

Dynamic compression tests at the strain rates of interest in vehicle crash are usually performed with a drop tower. In this test the loading velocity is nearly constant but the true strain rate, which depends on the instantaneous specimen thickness, is not. Therefore, the engineering strain rate input is optional so that the stress strain curves obtained at constant velocity loading can be used directly.

Correlation under triaxial loading is achieved by directly inputting the results of hydrostatic testing in addition to the uniaxial data. Without this additional information which is fully optional, triaxial response tends to be underestimated. To further improve the response under multiaxial loading, the strain rate parameter can either be based on the principal strain rates or the volumetric strain rate.

The strain is divided into two parts: a linear part and a non-linear part of the strain

$$E(t) = E^L(t) + E^N(t)$$

and the strain rate become

$$\dot{E}(t) = \dot{E}^L(t) + \dot{E}^N(t)$$

$\dot{E}^N$ is an expression for the past history of E^N. A postulated constitutive equation may be written as:

$$\sigma(t) = \int_{\tau=0}^{\infty} \left[E_t^N(\tau), S(t) \right] d\tau$$

where $S(t)$ is the state variable and $\int_{.\tau=0}^{\infty}$ is a functional of all values of τ in $T_\tau : 0 \le \tau \le \infty$ and

$$E_t^N(\tau) = E^N(t-\tau)$$

where τ is the history parameter:

$$E_t^N(\tau = \infty) \Leftrightarrow \textit{the virgin material}$$

It is assumed that the material remembers only its immediate past, i.e., a neighborhood about $\tau = 0$. Therefore, an expansion of $E_t^N(\tau)$ in a Taylor series about $\tau = 0$ yields:

$$E_t^N(\tau) = E^N(0) + \frac{\partial E_t^N}{\partial t}(0) dt$$

Hence, the postulated constitutive equation becomes:

$$\sigma(t)=\sigma^*\left(E^N(t),\dot{E}^N(t),S(t)\right)$$

where we have replaced $\dfrac{\partial E_t^N}{\partial t}$ by $\dot{E}^N$, and σ^* is a function of its arguments.

For a special case,

$$\sigma(t)=\sigma^*\left(\dot{E}^N(t),S(t)\right)$$

we may write

$$\dot{E}_t^N = f\left(S(t),s(t)\right)$$

which states that the nonlinear strain rate is the function of stress and a state variable which represents the history of loading. Therefore, the proposed kinetic equation for foam materials is:

$$\dot{E}^N=\frac{\sigma}{\|\sigma\|}D_0\exp\left[-c_0\left(\frac{tr(\sigma S)}{(\|\sigma\|)^2}\right)^{2n_0}\right]$$

where D_0, c_0, and n_0 are material constants, and S is the overall state variable. If either $D_0=0$ or $c_0\rightarrow\infty$ then the nonlinear strain rate vanishes.

$$\dot{S}_{ij}=\left[c_1\left(a_{ij}R-c_2S_{ij}\right)P+c_3W^{n_1}\left(\left\|\dot{E}^N\right\|\right)^{n_2}I_{ij}\right]R$$

$$R=1+c_4\left(\frac{\left\|\dot{E}^N\right\|}{c_5}-1\right)^{n_3}$$

$$P=tr\left(\sigma\dot{E}^N\right)$$

$$W=\int tr(\sigma\, dE)$$

where c_1, c_2, c_3, c_4, c_5, n_1, n_2, n_3, and a_{ij} are material constants and:

$$\|\sigma\|=\left(\sigma_{ij}\sigma_{ij}\right)^{\frac{1}{2}}$$

$$\left\|\dot{E}\right\|=\left(\dot{E}_{ij}\dot{E}_{ij}\right)^{\frac{1}{2}}$$

$$\left\|\dot{E}^N\right\|=\left(\dot{E}_{ij}^N\dot{E}_{ij}^N\right)^{\frac{1}{2}}$$

In the implementation by Fu Chang the model was simplified such that the input constants a_{ij} and the state variables S_{ij} are scalars.

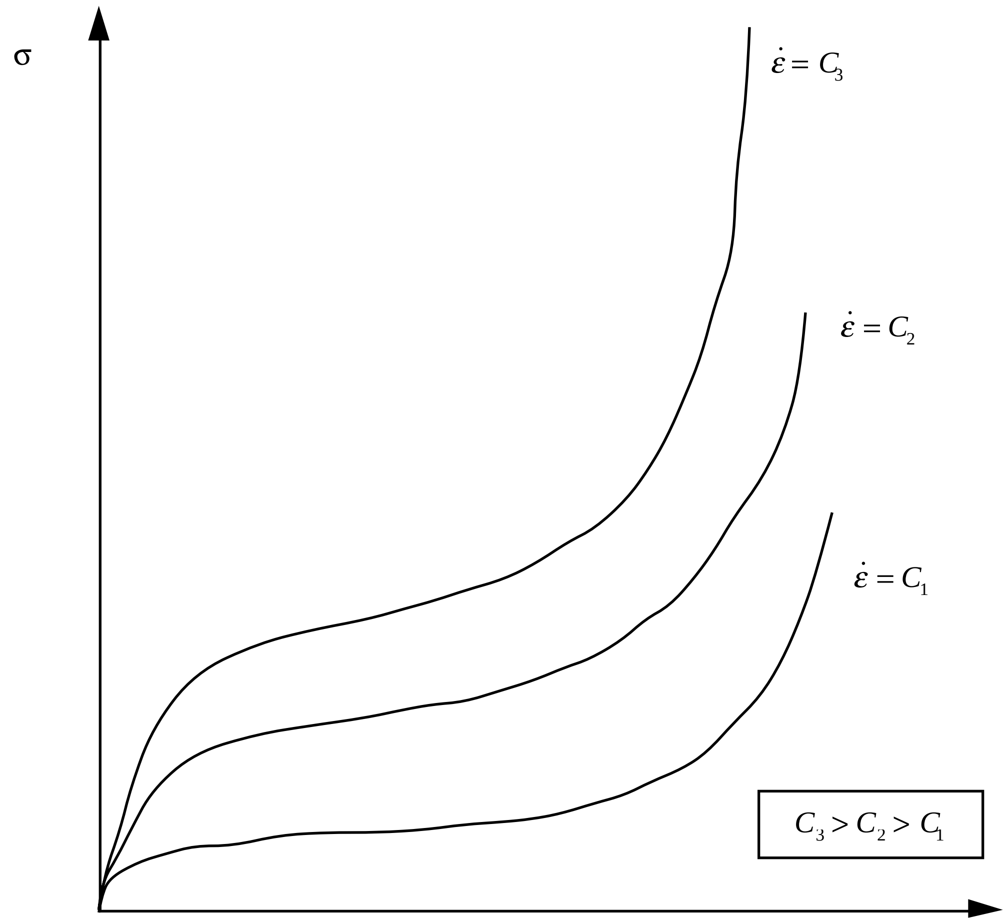


Figure 3.29. Rate effects in Fu Chang's foam model.

Material Type 86 (Orthotropic-Viscoelastic)

Columns		Quantity	Format
1-10	Card 3	E_a (see Figure 3.5)	E10.0
11-20		E_b	E10.0
21-30		E_c	E10.0
31-40		Volume fraction of viscoelastic material	E10.0
41-50		K, elastic bulk modulus.	E10.0
51-60		G_0, short-time shear modulus.	E10.0
61-70		G_∞, long-time shear modulus.	E10.0
71-80		β, decay constant.	E10.0
1-10	Card 4	ν_{ba}	E10.0
11-20		ν_{ca}	E10.0
21-30		ν_{cb}	E10.0
1-10	Card 5	G_{ab}	E10.0
11-20		G_{bc}	E10.0
21-30		G_{ca}	E10.0
1-10	Card 6	Material axes option, AOPT EQ.0.0: locally orthotropic with material axes determined by element nodes as shown in Figure 3.5. Cards 7 and 8 are blank with this option. EQ.1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center. Card 8 below is blank. EQ.2.0: globally orthotropic with material axes determined by vectors defined on Cards 7 and 8	E10.0

Columns		Quantity	Format
		EQ.3.0: This option determines locally orthotropic material axes by offsetting the material axes by an angle (Card 8) from a line in the plane of the shell determined by taking the cross product of the vector defined on Card 7 with the shell normal vector. In solid elements the normal vector is normal to the plane of the midsurface between the inner surface and outer surface defined by the first four nodes and the last four nodes of the connectivity of the element, respectively.	
		EQ.4.0: locally orthotropic in cylindrical coordinate system with material axes determined by the vector defined on Card 7 and the originating point, P, on Card 8.	
1-10	Card 7	x_p, define for AOPT = 1.0	E10.0
11-20		y_p, define for AOPT = 1.0	E10.0
21-30		z_p, define for AOPT = 1.0	E10.0
1-10	Card 7	a_1, define for AOPT = 2.0	E10.0
11-20		a_2, define for AOPT = 2.0	E10.0
21-30		a_3, define for AOPT = 2.0	E10.0
1-10	Card 7	v_1, define for AOPT = 3.0 & 4.0	E10.0
11-20		v_2, define for AOPT = 3.0 & 4.0	E10.0
21-30		v_3, define for AOPT = 3.0 & 4.0	E10.0
1-10	Card 8	d_1, define for AOPT = 2.0	E10.0
11-20		d_2, define for AOPT = 2.0	E10.0
21-30		d_3, define for AOPT = 2.0	E10.0
1-10	Card 8	Material angle beta, may be overridden on the element card	E10.0
1-10	Card 8	P_1, define for AOPT = 4.0	E10.0
11-20		P_2, define for AOPT = 4.0	E10.0
21-30		P_3, define for AOPT = 4.0	E10.0

Material Type 87 (Cellular Rubber)

This material model provides a cellular rubber model combined with linear viscoelasticity as outlined by Christensen [1980]. See Figure 3.30.

Columns		Quantity	Format
1-10	Card 3	υ, Poisson's ratio, typical values are between .0 to .2. Due to the large compressibility of air, large values of Poisson's ratio generates physically meaningless results.	E10.0
11-20		n, order of fit, (currently <3)	

If $n>0$ then a least square fit is computed with uniaxial data. Card 4 should contain the following information. Also see Figure 3.12. A Poisson's ratio of .5 is assumed for the void free rubber during the fit. The Poisson's ratio defined on Card 3 is for the cellular rubber. A void fraction formulation is used.

Columns		Quantity	Format
1-10	Card 4	Specimen gauge length	E10.0
11-20		Specimen width	E10.0
21-30		Specimen thickness	E10.0
31-40		Load curve ID giving the force versus actual change in the gauge length.	E10.0

The input on card 5 is required if and only if $n=0$

Columns		Quantity	Format
1-10	Card 5	C_{10}, see definition below	E10.0
11-20		C_{01}	E10.0
21-30		C_{11}	E10.0
31-40		C_{20}	E10.0
41-50		C_{02}	E10.0
	Card 6	Blank	

Material Type 87 (Cellular Rubber)

Columns		Quantity	Format
1-10	Card 7	p_0, initial air pressure	E10.0
11-20		ϕ, ratio of cellular rubber to rubber density	E10.0
21-30		γ_0, initial volumetric strain	E10.0
1-10	Card 8	Optional shear relaxation modulus, G, for rate effects.	E10.0
11-20		Optional decay constant, β_1.	E10.0

Rubber is generally considered to be fully incompressible since the bulk modulus greatly exceeds the shear modulus in magnitude. To model the rubber as an unconstrained material a hydrostatic work term, $W_H(J)$, is included in the strain energy functional which is function of the relative volume, J, [Ogden, 1984]:

$$W(J_1, J_2, J) = \sum_{p,q=0}^{n} C_{pq}\left(J_1 - 3\right)^p \left(J_2 - 3\right)^q + W_H(J)$$

$$J_1 = I_1 I_3^{-1/3}$$

$$J_2 = I_2 I_3^{-2/3}$$

In order to prevent volumetric work from contributing to the hydrostatic work the first and second invarients are modified as shown. This procedure is described in more detail by Sussman and Bathe [1987].

The effects of confined air pressure in its overall response characteristics is included by augmenting the stress state within the element by the air pressure.

$$\sigma_{ij} = \sigma_{ij}^{sk} - \delta_{ij}\sigma^{air}$$

where σ_{ij}^{sk} is the bulk skeletal stress and σ^{air} is the air pressure computed from the equation:

$$\sigma^{air} = -\frac{p_0 \gamma}{1 + \gamma - \phi}$$

where p_0 is the initial foam pressure usually taken as the atmospheric pressure and γ defines the volumetric strain

$$\gamma = V - 1 + \gamma_0$$

where V is the relative volume of the voids and γ_0 is the initial volumetric strain which is typically zero. The rubber skeletal material is assumed to be incompressible.

Rate effects are taken into accounted through linear viscoelasticity by a convolution integral of the form:

$$\sigma_{ij} = \int_0^t g_{ijkl}(t-\tau)\frac{\partial \varepsilon_{kl}}{\partial \tau} d\tau$$

or in terms of the second Piola-Kirchhoff stress, S_{ij}, and Green's strain tensor, E_{ij},

$$S_{ij} = \int_0^t G_{ijkl}(t-\tau)\frac{\partial E_{kl}}{\partial \tau} d\tau$$

where $g_{ijkl}(t-\tau)$ and $G_{ijkl}(t-\tau)$ are the relaxation functions for the different stress measures. This stress is addedto the stress tensor determined from the strain energy functional.

Since we wish to include only simple rate effects, the relaxation function is represented by one term from the Prony series:

$$g(t) = \alpha_0 + \sum_{m=1}^{N} \alpha_m e^{-\beta t}$$

given by,

$$g(t) = E_d e^{-\beta_1 t}$$

This model is effectively a Maxwell fluid which consists of a damper and spring in series. We characterize this in the input by a shear modulus, G, and decay constant, β_1.

The Mooney-Rivlin rubber model is obtained by specifying *n*=1. In spite of the differences in formulations with Model 27, we find that the results obtained with this model are nearly identical with those of 27 as long as large values of Poisson's ratio are used.

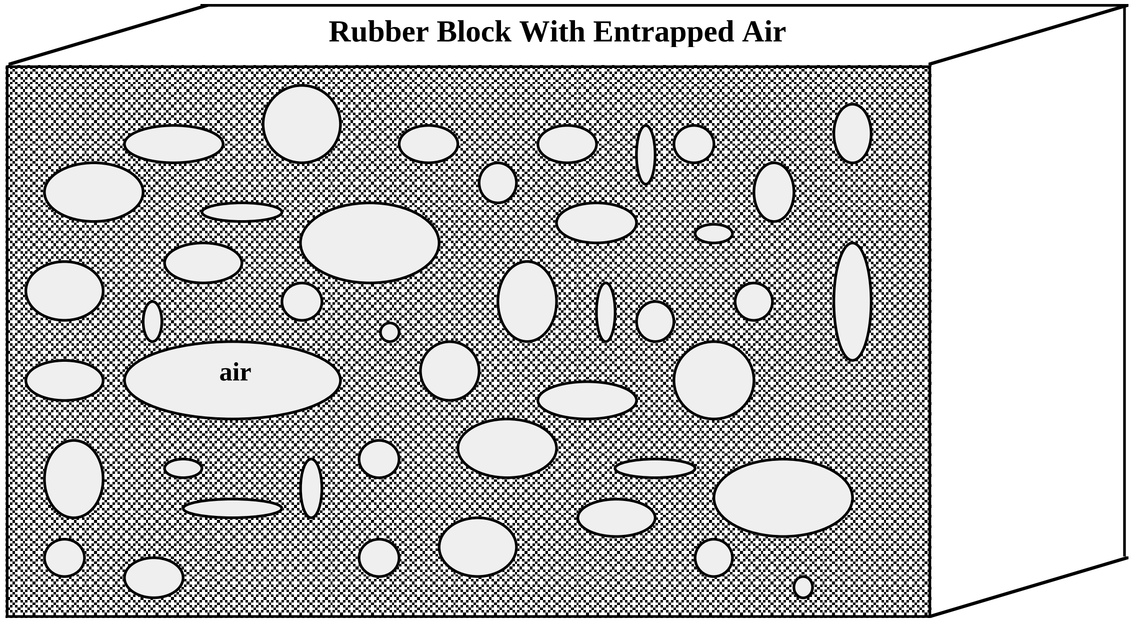


Figure 3.30. Cellular rubber with entrapped air. By setting the initial air pressure to zero, an open cell, cellular rubber can be simulated.

Material Type 88 (MTS Model)

The MTS model is due to Maudlin, Davidson, and Henninger [1990] and is available for applications involving large strains, high pressures and strain rates. As described in the foregoing reference, this model is based on dislocation mechanics and provides a better understanding of the plastic deformation process for ductile materials by using an internal state variable call the mechanical threshold stress. This kinematic quantity tracks the evolution of the material's microstructure along some arbitrary strain, strain rate, and temperature-dependent path using a differential form that balances dislocation generation and recovery processes. Given a valure for the mechanical threshold stress, the flow stress is determined using either a thermal-activation-controlled or a drag-controlled kinetics relationship.. An equation-of-state is required for solid elements and a bulk modulus must be defined below for shell elements.

Columns		Quantity	Format
1-10	Card 3	SIGA, $\hat{\sigma}_a$, dislocation interactions with long-range barriers (force/area).	E10.0
11-20		SIGI, $\hat{\sigma}_i$, dislocation interactions with interstitial atoms (force/area).	E10.0
21-30		SIGS, $\hat{\sigma}_s$, dislocation interactions with solute atoms (force/area).	E10.0
31-40		SIG0, $\hat{\sigma}_0$, initial value of $\hat{\sigma}$ at zero plastic strain (force/area). NOT USED.	E10.0
41-50		HF0 a_0, dislocation generation material constant (force/area).	E10.0
51-60		HF1, a_1, dislocation generation material constant (force/area).	E10.0
61-70		HF2, a_2, dislocation generation material constant (force/area).	E10.0
71-80		SIGS0, $\hat{\sigma}_{\varepsilon so}$, saturation threshold stress at 0^{o} K (force/area).	E10.0
1-10	Card 4	EDOTS0, $\dot{\varepsilon}_{\varepsilon so}$, reference strain-rate (time $^{-1}$).	E10.0
11-20		BURG, b, magnitude of Burgers vector (interatomic slip distance), (distance)	E10.0

Material Type 88 (Mechanical Threshold Stress)

Columns		Quantity	Format
21-30		CAPA, A, material constant.	E10.0
31-40		BOLTZ, k, Boltzmann's constant (energy/degree).	E10.0
41-50		SM0, G_0, shear modulus at zero degrees Kelvin (force/area).	E10.0
51-60		SM1, b_1, shear modulus constant (force/area).	E10.0
61-70		SM2, b_2, shear modulus constant (degree).	E10.0
71-80		EDOT0, $\dot{\varepsilon}_o$, reference strain-rate (time^{-1}).	E10.0
1-10	Card 5	G0, g_0 , normalized activation energy for a . dislocation/dislocation interaction.	E10.0
11-20		PINV, $\frac{1}{p}$, material constant.	E10.0
21-30		QINV, $\frac{1}{q}$, material constant.	E10.0
31-40		EDOTI, $\dot{\varepsilon}_{o,i}$, reference strain-rate (time^{-1}).	E10.0
41-50		G0I, $g_{0,i}$, normalized activation energy for a . dislocation/interstitial interaction.	E10.0
51-60		PINVI, $\frac{1}{p_i}$, material constant.	E10.0
61-70		QINVI, $\frac{1}{q_i}$, material constant.	E10.0
71-80		EDOTS, $\dot{\varepsilon}_{o,s}$, reference strain-rate (time^{-1}).	E10.0
1-10	Card 6	G0S, $g_{0,s}$, normalized activation energy for a dislocation/solute interaction.	E10.0
11-20		PINVS, $\frac{1}{p_s}$, material constant.	E10.0
21-30		QINVS, $\frac{1}{q_s}$, material constant.	E10.0
31-40		RHOCPR, ρc_p, product of density and specific heat	E10.0
41-50		TEMPRF, T_{ref}, initial element temperature in degrees K.	E10.0

Columns		Quantity	Format
51-60		BULK, bulk modulus defined for shell elements only. Do not input for solid elements.	E10.0
1-10	Card 7	ALPHA, α, material constant (typical value is between 0 and 2).	E10.0
11-20		EPS0, ε_o, factor to normalize strain rate in the calculation of Θ_o. (Use 1., 10^{-3}, or 10^{-6} for time units of seconds, milliseconds, or microseconds, respectively.)	E10.0

The flow stress σ is given by:

$$\sigma = \hat{\sigma}_a + \frac{G}{G_0}\left[s_{th}\hat{\sigma} + s_{th,i}\hat{\sigma}_i + s_{th,s}\hat{\sigma}_s\right]$$

The first product in the equation for σ contains a micro-structure evolution variable, i.e,. $\hat{\sigma}$, called the *Mechanical Threshold Stress* (MTS), that is multiplied by a constant-structure deformation variable s_{th}: s_{th} is a function of absolute temperature T and the plastic strain-rates $\dot{\varepsilon}^P$. The evolution equation for $\hat{\sigma}$ is a differential hardening law representing dislocation-dislocation interactions:

$$\frac{\partial \hat{\sigma}}{\partial \varepsilon^p} \equiv \Theta_o \left[1 - \frac{\tanh\left(\alpha \dfrac{\hat{\sigma}}{\hat{\sigma}_{\varepsilon s}}\right)}{\tanh(\alpha)}\right]$$

The term, $\frac{\partial \hat{\sigma}}{\partial \varepsilon^p}$, represents the hardening due to dislocation generation and the stress ratio, $\frac{\hat{\sigma}}{\hat{\sigma}_{\varepsilon s}}$, represents softening due to dislocation recovery. The threshold stress at zero strain-hardening $\hat{\sigma}_{\varepsilon s}$ is called the saturation threshold stress. Relationships for Θ_o, $\hat{\sigma}_{\varepsilon s}$ are:

$$\Theta_o = a_o + a_1 \ln\left(\frac{\dot{\varepsilon}^p}{\varepsilon_0}\right) + a_2\sqrt{\frac{\dot{\varepsilon}^p}{\varepsilon_0}}$$

which contains the material constants a_o, a_1, and a_2. The constant, $\hat{\sigma}_{\varepsilon s}$, is given as:

$$\hat{\sigma}_{\varepsilon s} = \hat{\sigma}_{\varepsilon so}\left(\frac{\dot{\varepsilon}^{p}}{\dot{\varepsilon}_{\varepsilon so}}\right)^{kT/Gb^3A}$$

which contains the input constants: $\hat{\sigma}_{\varepsilon so}$, $\dot{\varepsilon}_{\varepsilon so}$, b, A, and k. The shear modulus G appearing in these equations is assumed to be a function of temperature and is given by the correlation.

$$G = G_0 - b_1 / (e^{b_2/T} - 1)$$

which contains the constants: G_0, b_1, and b_2. For thermal-activation controlled deformation s_{th} is evaluated via an Arrhenius rate equation of the form:

$$s_{th} = \left[1 - \left(\frac{kT1n\left(\frac{\dot{\varepsilon}_0}{\dot{\varepsilon}^p}\right)}{Gb^3 g_0}\right)^{\frac{1}{q}}\right]^{\frac{1}{p}}$$

The absolute temperature is given as:

$$T = T_{ref} + \rho c_p E$$

where E in the internal energy density per unit initial volume.

Material Type 90 (Acoustic)

This model is appropiate for tracking low pressure stress waves in an acoustic media such as air or water and can be used only with the acoustic pressure element formulation. The acoustic pressure element requires only one unknown per node. This element is very cost effective.

Columns		Quantity	Format
1-10	Card 3	Sound speed	E10.0
11-20		β, damping factor. Recommend values are between 0.1 and 1.0.	E10.0
21-30		Cavitation flag EQ.0.0: off EQ.1.0: on	E10.0
31-40		Atmospheric pressure (optional)	E10.0
1-10	Card 4	Gravitational acceleration constant (optional)	E10.0
11-20		x-coordinate of free surface point	E10.0
21-30		y-coordinate of free surface point	E10.0
31-40		z-coordinate of free surface point	E10.0
41-50		x-direction cosine of free surface normal vector	E10.0
51-60		y-direction cosine of free surface normal vector	E10.0
61-70		z-direction cosine of free surface normal vector	E10.0
	Card 5	Blank	
	Card 6	Blank	
	Card 7	Blank	
	Card 8	Blank	

Material Type 96 (Brittle Damage Model)

This model, implemented into LS-DYNA by Govindjee simulates the cracking behavior of concrete. Rebar effects may be included.

Columns		Quantity	Format
1-10	Card 3	Young's modulus, E.	E10.0
11-20		Fraction of reinforcement in section.	E10.0
21-30		Young's modulus of reinforcement.	E10.0
31-40		Yield stress of reinforcement.	E10.0
41-50		Hardening modulus of reinforcement.	E10.0
51-60		True failure strain of reinforcement.	E10.0
1-10	Card 4	Poisson's ratio υ	E10.0
1-10	Card 5	Tensile limit, f_{t0}	E10.0
11-20		Shear limit, f_{s0}	E10.0
21-30		Compressive yield stress EQ.0: no compressive yield	E10.0
1-10	Card 6	Fracture toughness, g_c	E10.0
1-10	Card 7	Shear retention,β	E10.0
1-10	Card 8	Viscosity,η	E10.0

A full description of the tensile and shear damage parts of this material model is given in Govindjee, Kay and Simo[1994,1995]. It is an anisotropic brittle damage model designed primarily for concrete though it can be applied to a wide variety of brittle materials. It admits progressive degradation of tensile and shear strengths across smeared cracks that are initiated under tensile loadings. Compressive failure is governed by a simplistic J2 flow correction that can be disabled if not desired. Damage is handled by treating the rank 4 elastic stiffness tensor as an evolving internal variable for the material. Softening induced mesh dependencies are handled by a characteristic length method (Oliver [1989]).

Material Type 96 (Brittle Damage Model)

Description of properties:

1. E is the Young's modulus of the undamaged material also known as the virgin modulus.

2. υ is the Poisson's ratio of the undamaged material also known as the virgin Poisson's ratio.

3. f_n is the initial principal tensile strength (stress) of the material. Once this stress has been reached at a point in the body a smeared crack is initiated there with a normal that is co-linear with the 1st principal direction. Once initiated, the crack is fixed at that location, though it will convect with the motion of the body. As the loading progresses the allowed tensile traction normal to the crack plane is progressively degraded to a small machine dependent constant.

   The degradation is implemented by reducing the material's modulus normal to the smeared crack plane according to a maximum dissipation law that incorporates exponential softening. The restriction on the normal tractions is given by

$$\phi_t = (\mathbf{n} \otimes \mathbf{n}):\sigma - f_n + (1-\varepsilon) f_n \left(1 - \exp[-H\alpha]\right) \leq 0$$

   where $\mathbf{n}$ is the smeared crack normal, ε is the small constant, H is the softening modulus, and α is an internal variable. H is set automatically by the program; see g_c below. α measures the crack field intensity and is output in the equivalent plastic strain field, $\bar{\varepsilon}^p$, in a normalized fashion.

   The evolution of alpha is governed by a maximum dissipation argument. When the normalized value reaches unity it means that the material's strength has been reduced to 2% of its original value in the normal and parallel directions to the smeared crack. Note that for plotting purposes it is never output greater than 5.

4. f_s is the initial shear traction that may be transmitted across a smeared crack plane. The shear traction is limited to be less than or equal to $f_s(1-\beta)\left(1-\exp[-H\alpha]\right)$, through the use of two orthogonal shear damage surfaces. Note that the shear degradation is coupled to the tensile degradation through the internal variable alpha which measures the intensity of the crack field. β is the shear retention factor

defined below. The shear degradation is taken care of by reducing the material's shear stiffness parallel to the smeared crack plane.

5. g_c is the fracture toughness of the material. It should be entered as fracture energy per unit area crack advance. Once entered the softening modulus is automatically calculated based on element and crack geometries.

6. β is the shear retention factor. As the damage progresses the shear tractions allowed across the smeared crack plane asymptote to the product βf_s.

7. η represents the viscosity of the material. Viscous behavior is implemented as a simple Perzyna regularization method. This allows for the inclusion of first order rate effects. The use of some viscosity is recommend as it serves as regularizing parameter that increases the stability of calculations.

8. σ_y is a uniaxial compressive yield stress. A check on compressive stresses is made using the J2 yield function $\mathbf{s}:\mathbf{s} - \sqrt{\frac{2}{3}}\sigma_y \le 0$, where $\mathbf{s}$ is the stress deviator. If violated, a J2 return mapping correction is executed. This check is executed when (1) no damage has taken place at an integration point yet, (2) when damage has taken place at a point but the crack is currently closed, and (3) during active damage after the damage integration (ie. as an operator split). Note that if the crack is open the plasticity correction is done in the plane-stress subspace of the crack plane.

Remark: A variety of experimental data has been replicated using this model from quasi-static to explosive situations. Reasonable properties for a standard grade concrete would be E=3.15x10^6 psi, f_n=450 psi, f_s=2100 psi, ν = 0.2, g_c = 0.8 lbs/in, β =0.03, η = 0.0 psi-sec, σ_y = 4200 psi. For stability, values of η between 104 to 106 psi/sec are recommended. Our limited experience thus far has shown that many problems require nonzero valuies of η to run to avoid error terminations.

Remark: Various other internal variables such as crack orientations and degraded stiffness tensors are internally calculated but currently not available for output.

Material Type 100 (Spot weld)

This material model applies to beam element type 9 for spot welds. These beam elements may be placed between any two deformable shell surfaces and tied with type 7 constraint contact which eliminates the need to have adjacent nodes at spotweld locations. Beam spot welds may be placed between rigid bodies and rigid/deformable bodies by making the node on one end of the spot weld a rigid body node which can be an extra node for the rigid body. In the same way, rigid bodies may also be tied together with this spotweld option.

It is advisable to include all spotwelds, which provide the slave nodes, and spot welded materials, which define the master segments, within a single type 7 tied interface. As a constraint method, multiple type 7 interfaces are treated independently which can lead to significant problems if such interfaces share common nodal points. The offset option, “o 7”, should not be used with spotwelds.

Columns		Quantity	Format
1-10	Card 3	Young’s modulus	E10.0
1-10	Card 4	Poisson’s ratio	E10.0
1-10	Card 5	Yield stress	E10.0
1-10	Card 6	Hardening modulus, E_t	E10.0
1-10	Card 7	Time step size for mass scaling, Δt	E10.0
11-20		Failure time if nonzero. If zero this option is ignored.	E10.0
1-10	Card 8	Failure strain for eroding elements.	E10.0
11-20		Force resultant N_{rr_F} at failure	E10.0
21-30		Force resultant N_{rs_F} at failure	E10.0
31-40		Force resultant N_{rt_F} at failure	E10.0
41-50		Moment resultant M_{rr_F} at failure	E10.0
51-60		Moment resultant M_{ss_F} at failure	E10.0
61-70		Moment resultant T_{rr_F} at failure	E10.0

Columns	Quantity	Format
71-80	Number of force vectors stored for filtering. The default value is set to zero which is generally recommended unless oscillatory resultant forces are observed in the time history databases. Even though these welds should not oscillate significantly, this option was added for consistency with the other spot weld options. NF affects the storage since it is necessary to store the resultant forces as history variables. When NF is nonzero, the resultants in the output databases are filtered.	E10.0

The weld material is modeled with isotropic hardening plasticity coupled coupled to two failure models. The first model specifies a failure strain which fails each integration point in the spot weld indepedently. The second model fails the entire weld if the resultants are outside of the failure surface defined by:

$$\left(\frac{N_{rr}}{N_{rr_F}}\right)^2+\left(\frac{N_{rs}}{N_{rs_F}}\right)^2+\left(\frac{N_{rt}}{N_{rt_F}}\right)^2+\left(\frac{M_{rr}}{M_{rr_F}}\right)^2+\left(\frac{M_{ss}}{M_{ss_F}}\right)^2+\left(\frac{T_{rr}}{T_{rr_F}}\right)^2-1=0$$

where the *numerators* in the equation are the resultants calculated in the local coordinates of the cross section, and the **denominators** are the values specified in the input.

If the failure strain is set to zero, the failure strain model is not used. In a similar manner, when the value of a resultant at failure is set to zero, the corresponding term in the failure surface is ignored. For example, if only N_{rr_F} is nonzero, the failure surface is reduced to $|N_{rr}|=N_{rr_F}$. None, either, or both of the failure models may be active depending on the specified input values.

The inertias of the spot welds are scaled during the first time step so that their stable time step size is Δt. A strong compressive load on the spot weld at a later time may reduce the length of the spot weld so that stable time step size drops below Δt. If the value of Δt is zero, mass scaling is not performed, and the spot welds will probably limit the time step size. Under most circumstances, the inertias of the spot welds are small enough that scaling them will have a negligible effect on the structural response and the use of this option is encouraged.

Material Type 103 (Anisotropic Viscoplastic)

This anisotropic-viscoplastic material model applies to shell and brick elements. The material constants may be fit directly or, if desired, stress versus strain data may be input and a least squares fit will be performed by LS-DYNA to determine the constants. Kinematic or isotopic or a combination of kinematic and isotropic hardening may be used.. A detailed describtion of this model can be found in the following references: Berstad, Langseth, and Hopperstad [1994]; Hopperstad and Remseth [1995]; and Berstad [1996].

Columns		Quantity	Format
1-10	Card 3	Young’s modulus	E10.0
11-20		Poissons's ratio	E10.0
21-30		Initial yield stress	E10.0
31-40		Flag EQ.0 Give all material parameters EQ.1 Material parameters are fit in LS-DYNA to Load curve or Table given below. The parameters Q_{r1}, C_{r1}, Q_{r2}, and C_{r2} for isotropic hardening are determined by the fit and those for kinematic hardening are found by scaling those for isotropic hardening by $(1-\alpha)$ where α is defined below in columns 51-60. CHECK THE FIT IN THE PRINTED OUTPUT FILE BEFORE RUNNING THE CALCULATION.	E10.0
41-50		Load curve ID or Table ID. The load curve ID defines effective stress versus effective plastic strain. Card 4 is ignored with this option. The table ID, see Figure 3.9, defines for each strain rate value a load curve ID giving the stress versus effectiveplastic strain for that rate. If the load curve only is used, then the coefficients V_k and V_m must be given if viscoplastice behavior is desired. If a Table ID is given these coefficients are determined internally during initialization.	E10.0
51-60		α distribution of hardening used in the curve-fitting $\alpha=0$ pure kinematic hardening $\alpha=1$ pure isotropic hardening	E10.0
1-10	Card 4	Isotropic hardening parameter Q_{r1}	E10.0
11-20		Isotropic hardening parameter C_{r1}	E10.0

Material Type 103 (Anisotropic Viscoplastic)

Columns		Quantity	Format
21-30		Isotropic hardening parameter Q_{r2}	E10.0
31-40		Isotropic hardening parameter C_{r2}	E10.0
41-50		Kinematic hardening parameter $Q_{\chi 1}$	E10.0
51-60		Kinematic hardening parameter $C_{\chi 1}$	E10.0
61-70		Kinematic hardening parameter $Q_{\chi 2}$	E10.0
71-80		Kinematic hardening parameter $C_{\chi 2}$	E10.0
1-10	Card 5	Viscous material parameter V_k	E10.0
11-20		Viscous material parameter V_m	E10.0
21-30		R_{00} for shell (Default=1.0)	E10.0
31-40		R_{45} for shell (Default=1.0)	E10.0
41-50		R_{90} for shell (Default=1.0)	E10.0
21-30		F for brick (Default =1/2)	E10.0
31-40		G for brick (Default =1/2)	E10.0
41-50		H for brick (Default =1/2)	E10.0
51-60		L for brick (Default =3/2)	E10.0
61-70		M for brick (Default =3/2)	E10.0
71-80		N for brick (Default =3/2)	E10.0
1-10	Card 6	Material axes option, AOPT EQ.0.0: locally orthotropic with material axes determined by element nodes as shown in Figure 3.5. Cards 7 and 8 are blank with this option. EQ.1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center. Card 8 below is blank. EQ.2.0: globally orthotropic with material axes determined by vectors defined on Cards 7 and 8	E10.0

Columns		Quantity	Format
		EQ.3.0: applicable to shell elements only. This option determines locally orthotropic material axes by offsetting the material axes by an angle (Card 8) from a line in the plane of the shell determined by taking the cross product of the vector defined on Card 7 with the shell normal vector.	
1-10	Card 7	x_p, define for AOPT = 1.0	E10.0
11-20		y_p, define for AOPT = 1.0	E10.0
21-30		z_p, define for AOPT = 1.0	E10.0
1-10	Card 7	a_1, define for AOPT = 2.0	E10.0
11-20		a_2, define for AOPT = 2.0	E10.0
21-30		a_3, define for AOPT = 2.0	E10.0
1-10	Card 7	v_1, define for AOPT = 3.0	E10.0
11-20		v_2, define for AOPT = 3.0	E10.0
21-30		v_3, define for AOPT = 3.0	E10.0
1-10	Card 8	d_1, define for AOPT = 2.0	E10.0
11-20		d_2, define for AOPT = 2.0	E10.0
21-30		d_3, define for AOPT = 2.0	E10.0

The uniaxial stress-strain curve is given on the following form

$$\begin{aligned}\sigma(p,\dot{p}) = \sigma_0 &+ Q_{r1}(1-\exp(-C_{r1}p)) + Q_{r2}(1-\exp(-C_{r2}p)) \\ &+ Q_{\chi 1}(1-\exp(-C_{\chi 1}p)) + Q_{\chi 2}(1-\exp(-C_{\chi 2}p)) \\ &+ V_k \dot{p}^{V_m}\end{aligned}$$

For bricks the following yield criteria is used

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

where p is the effective plastic strain and $\dot{p}$ is the effectiv plastic strain rate$_e$. For shells the anisotropic behavior is given by R_{00}, R_{45} and R_{90}. The model will work when the three first parameters in card 3 are given values. When $V_k = 0$ the material will behave elasto-plastically. Default values are given by:

$$F = G = H = \frac{1}{2}$$

$$L = M = N = \frac{3}{2}$$

$$R_{00} = R_{45} = R_{90} = 1$$

Strain rate of accounted for using the Cowper and Symonds model which, e.g., model 3, scales the yield stress with the factor:

$$1 + \left(\frac{\dot{\varepsilon}}{C}\right)^{1/p}$$

To convert these constants set the viscoelastic constants, V_k and V_m, to the following values:

$$V_k = \sigma\left(\frac{1}{C}\right)^{\frac{1}{p}}$$

$$V_m = \frac{1}{p}$$

This model properly treats rate effects and should provide superior results to models 3 and 24.

Material Type 104 (Anisotropic Viscoplastic Damage)

This anisotropic-viscoplastic-damage material model applies to shell and brick elements. The material constants may be fit directly or, if desired, stress versus strain data may be input and a least squares fit will be performed by LS-DYNA to determine the constants.

Columns		Quantity	Format
1-10	Card 3	Young’s modulus	E10.0
11-20		Poissons's ratio	E10.0
21-30		Initial yield stress	E10.0
31-40		Flag EQ.0 Give all material parameters EQ.1 Material parameters are fit in LS-DYNA to Load curve or Table given below. The parameters Q_1, C_1, Q_2, and C_2 for isotropicp isotropic hardening are determined by the fit CHECK THE FIT IN THE PRINTED OUTPUT FILE BEFORE RUNNING THE CALCULATION.	E10.0
41-50		Load curve ID or Table ID. The load curve ID defines effective stress versus effective plastic strain. Card 4 is ignored with this option. The table ID, see Figure 3.9, defines for each strain rate value a load curve ID giving the stress versus effectiveplastic strain for that rate. If the load curve only is used, then the coefficients V_k and V_m must be given if viscoplastice behavior is desired. If a Table ID is given these coefficients are determined internally during initialization.	E10.0
1-10	Card 4	Isotropic hardening parameter Q_1	E10.0
11-20		Isotropic hardening parameter C_1	E10.0

Material Type 103 (Anisotropic Viscoplastic)

Columns		Quantity	Format
21-30		Isotropic hardening parameter Q_2	E10.0
31-40		Isotropic hardening parameter C_2	E10.0
41-50		Effective plastic strain when damage start p_D	E10.0
51-60		Resistance against damage s	E10.0
61-70		Critical damage D_C	E10.0
1-10	Card 5	Viscous material parameter V_k	E10.0
11-20		Viscous material parameter V_m	E10.0
21-30		R_{00} for shell (Default=1.0)	E10.0
31-40		R_{45} for shell (Default=1.0)	E10.0
41-50		R_{90} for shell (Default=1.0)	E10.0
21-30		F for brick (Default =1/2)	E10.0
31-40		G for brick (Default =1/2)	E10.0
41-50		H for brick (Default =1/2)	E10.0
51-60		L for brick (Default =3/2)	E10.0
61-70		M for brick (Default =3/2)	E10.0
71-80		N for brick (Default =3/2)	E10.0
1-10	Card 6	Material axes option, AOPT EQ.0.0: locally orthotropic with material axes determined by element nodes as shown in Figure 3.5. Cards 7 and 8 are blank with this option. EQ.1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center. Card 8 below is blank. EQ.2.0: globally orthotropic with material axes determined by vectors defined on Cards 7 and 8	E10.0

Columns		Quantity	Format
		EQ.3.0: applicable to shell elements only. This option determines locally orthotropic material axes by offsetting the material axes by an angle (Card 8) from a line in the plane of the shell determined by taking the cross product of the vector defined on Card 7 with the shell normal vector.	
1-10	Card 7	x_p, define for AOPT = 1.0	E10.0
11-20		y_p, define for AOPT = 1.0	E10.0
21-30		z_p, define for AOPT = 1.0	E10.0
1-10	Card 7	a_1, define for AOPT = 2.0	E10.0
11-20		a_2, define for AOPT = 2.0	E10.0
21-30		a_3, define for AOPT = 2.0	E10.0
1-10	Card 7	v_1, define for AOPT = 3.0	E10.0
11-20		v_2, define for AOPT = 3.0	E10.0
21-30		v_3, define for AOPT = 3.0	E10.0
1-10	Card 8	d_1, define for AOPT = 2.0	E10.0
11-20		d_2, define for AOPT = 2.0	E10.0
21-30		d_3, define for AOPT = 2.0	E10.0

The uniaxial stress-strain curve is given on the following form

$$\sigma(p,\dot{p}) = \sigma_0 + Q_1(1-\exp(-C_1 p)) + Q_2(1-\exp(-C_2 p)) + V_k \dot{p}^{V_m}$$

For bricks the following yield criteria is used

$$F(\sigma_{22}-\sigma_{33})^2 + G(\sigma_{33}-\sigma_{11})^2 + H(\sigma_{11}-\sigma_{22})^2 + 2L\sigma_{23}^2 + 2M\sigma_{31}^2 + 2N\sigma_{12}^2 = \sigma(p,p)$$

where p is the effective plastic strain and $\dot{p}$ is the effectiv plastic strain rate$_e$. For shells the anisotropic behavior is given by R_{00}, R_{45} and R_{90}. The model will work when the three first parameters in card 3 are given values. When $V_k = 0$ the material will behave elasto-plastically. Default values are given by:

$$F = G = H = \frac{1}{2}$$

$$L = M = N = \frac{3}{2}$$

$$R_{00} = R_{45} = R_{90} = 1$$

Strain rate of accounted for using the Cowper and Symonds model which, e.g., model 3, scales the yield stress with the factor:

$$1 + \left(\frac{\dot{\varepsilon}}{C}\right)^{1/p}$$

To convert these constants set the viscoelastic constants, V_k and V_m, to the following values:

$$V_k = \sigma\left(\frac{1}{C}\right)^{\frac{1}{p}}$$

$$V_m = \frac{1}{p}$$

Evolution of damage

$$\dot{D} = \begin{cases} 0, p < p_D \\ \dfrac{\sigma_{eq}^2 r_v}{2Es}\dot{p}, p \geq p_D \end{cases} .$$

where

$$r_v = \frac{2}{3}(1+\nu) + 3(1-2\nu)\left(\frac{\sigma_m}{\sigma_{eq}}\right)^2, \sigma_H = (\sigma_1 + \sigma_2 + \sigma_3)/3$$

and where σ_{eq} is the effective stress. When critical damage D_C the element will fail and be eroded.

Material Type 105 (Piecewise Linear Isotropic Plasticity Damage)

Columns		Quantity	Format
1-10	Card 3	Young's modulus	E10.0
11-20		Strain rate parameter, C	E10.0
21-30		Strain rate parameter, p	E10.0
1-10	Card 4	Poisson's ratio	E10.0
11-20		Effective plastic strain when damage start p_D	E10.0
21-30		Resistance against damage s	E10.0
31-40		Critical damage D_C	E10.0
1-10	Card 5	Yield stress	E10.0
11-20		Load curve ID or Table ID. The load curve ID defines effective stress versus effective plastic strain. Cards 7 and 8 are ignored with this option. The table ID, see Figure 3.9, defines for each strain rate value a load curve ID giving the stress versus effectiveplastic strain for that rate. The stress versus effective plastic strain curve for the lowest value of strain rate is used if the strain rate falls below the minimum value. Likewise, the stress versus effective plastic strain curve for the highest value of strain rate is used if the strain rate exceeds the maximum value. The strain rate parameters on card 3, the curve ID on card 6, and cards 7 and 8 are ignored if a Table ID is defined.	E10.0
1-10	Card 6	Tangent modulus, ignored if the stress-strain curve is defined below	E10.0
11-20		Failure flag: EQ.0.0: Failure is not considered. This option is recommended if failure is not of interest since many caluculations will be saved. GT.0.0: Plastic strain to failure. When the plastic strain reaches this value, the element is deleted from the calculation.	E10.0
21-30		Time step size for automatic element deletion	E10.0
31-40		Load curve number to scale yield stress to account for strain rate effects.	E10.0

Material Type 105 **(Piecewise Linear Isotropic Plasticity Damage)**

Columns		Quantity	Format
1-80	Card 7	Effective plastic strain values (define up to 8 points)	E10.0
1-80	Card 8	Corresponding yield stress values	E10.0

The stress strain behavior may be treated by a bilinear stress strain curve by defining the tangent modulus. Alternately, a stress versus effective plastic strain curve (Card 5, Columns 11-20) similar to that shown in Figure 3.8 can be used. If eight point are insufficient, a load curve may be used with an arbitrary number of points. The cost is roughly the same for either approach. The most general approach is to used the table definition, (Card 5, Columns 11-20) discussed below.

Three options to account for strain rate effects are possible.

I. Strain rate may be accounted for using the Cowper and Symonds model which scales the yield stress with the factor

$$1+\left(\frac{\dot{p}}{C}\right)^{1/p}$$

where $\dot{p}$ is the strain rate.

II. For complete generality a load curve (Card 5) to scale the yield stress may be input instead. In this curve the scale factor versus strain rate is defined.

III. If different stress versus strain curves can be provided for various strain rates, the option using the reference to a table (Card 5, Columns 11-20) can be used. Then the table input in Section 22 (Load Curve/Table Definitions) has to be used. See Figure 3.9.

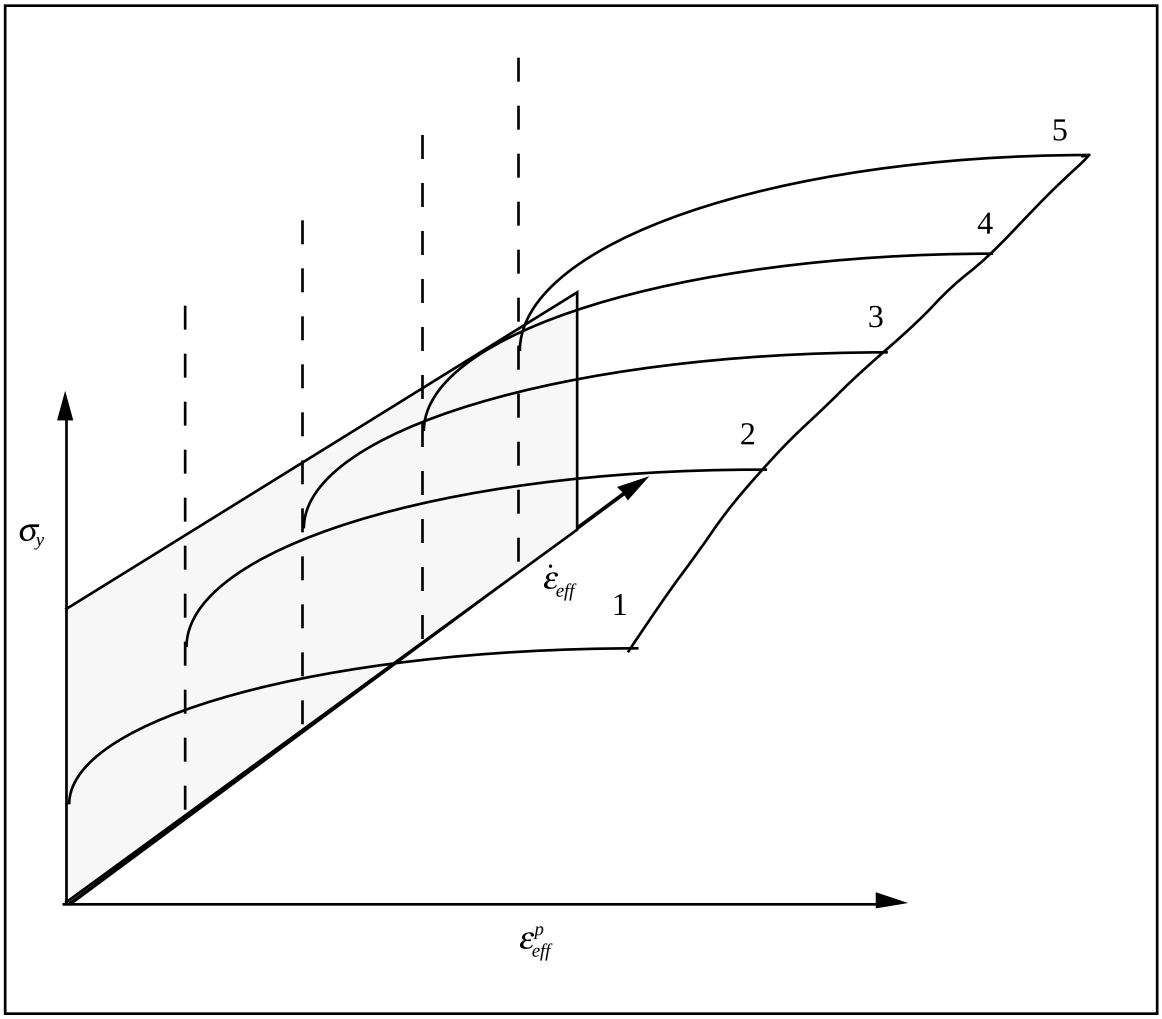


Figure 3.9. Rate effects may be accounted for by defining a table of curves. If a table ID is specified a curve ID is given for each strain rate, see Section 22. Intermediate values are found by interpolating between curves. Effective plastic strain versus yield stress is expected.

Evolution of damage

$$\dot{D} = \begin{cases} 0, p < p_D \\ \dfrac{\sigma_{eq}^2 r_v}{2Es} \dot{p}, p \geq p_D \end{cases} .$$

where

$$r_v = \frac{2}{3}(1+\nu)+3(1-2\nu)\left(\frac{\sigma_m}{\sigma_{eq}}\right)^2, \sigma_H = \left(\sigma_1+\sigma_2+\sigma_3\right)/3$$

and where σ_{eq} is the effective stress. When critical damage D_C the element will fail and be eroded.

Material Type 123 (Modified Piecewise Linear Isotropic Plasticity)

An elasto-plastic material with an arbitrary stress versus strain curve and arbitrary strain rate dependency can be defined. This model is currently available for shell elements only. Another model, material type 24, is similar but lacks the enhanced failure criteria. Failure is based on effecitve plastic strain, plastic thinning, the major principal in plane strain component, or a minimum time step size. See the discussion under the model description for material type 124 if more information is desired.

Columns		Quantity	Format
1-10	Card 3	Young's modulus	E10.0
11-20		Strain rate parameter, C	E10.0
21-30		Strain rate parameter, p	E10.0
31-40		Formulation for rate effects: EQ.0.0: Scale yield stress (default) EQ.1.0: Viscoplastic formulation	E10.0
1-10	Card 4	Poisson's ratio	E10.0
1-10	Card 5	Yield stress	E10.0
11-20		Load curve ID or Table ID. The load curve ID defines effective stress versus effective plastic strain. Cards 7 and 8 are ignored with this option. The table ID, see Figure 3.9, defines for each strain rate value a load curve ID giving the stress versus effectiveplastic strain for that rate. The stress versus effective plastic strain curve for the lowest value of strain rate is used if the strain rate falls below the minimum value. Likewise, the stress versus effective plastic strain curve for the highest value of strain rate is used if the strain rate exceeds the maximum value. The strain rate parameters on card 3, the curve ID on card 6, and cards 7 and 8 are ignored if a Table ID is defined.	E10.0
1-10	Card 6	Tangent modulus, ignored if the stress-strain curve is defined below	E10.0
11-20		Failure flag: LT.0.0: User defined failure subroutine is called to determine failure EQ.0.0: Failure is not considered. This option is recommended if failure is not of interest since many caluculations will be saved.	E10.0

Columns		Quantity	Format
		GT.0.0: Plastic strain to failure. When the plastic strain reaches this value, the element is deleted from the calculation.	
21-30		Time step size for automatic element deletion	E10.0
31-40		Load curve number to scale yield stress to account for strain rate effects.	E10.0
41-50		Thinning plastic strain at failure. This number should be given as a positive number.	E10.0
51-60		Major in plane strain at failure.	E10.0
61-70		Number of through thickness integration points which must fail before the element is deleted. (If zero, all points must fail.)	E10.0
1-80	Card 7	Effective plastic strain values (define up to 8 points)	E10.0
1-80	Card 8	Corresponding yield stress values	E10.0

Material Type 124 (Plasticity Compression and Tension)

Columns		Quantity	Format
1-10	Card 3	Young's modulus	E10.0
11-20		Strain rate parameter, C	E10.0
21-30		Strain rate parameter, p	E10.0
1-10	Card 4	Poisson's ratio	E10.0
	Card 5	Blank	
	Card 6	Blank	
11-20		Failure flag: LT.0.0: User defined failure subroutine is called to determine failure EQ.0.0: Failure is not considered. This option is recommended if failure is not of interest since many caluculations will be saved. GT.0.0: Plastic strain to failure. When the plastic strain reaches this value, the element is deleted from the calculation.	E10.0
21-30		Time step size for automatic element deletion	E10.0
1-10	Card 7	Load curve ID in Compression	E10.0
11-20		Load curve ID in Tension	E10.0
1-10	Card 8	Compressive mean stress (pressure) at which the yield stress follows load curve ID. If the pressure falls between PC and PT a weighted average of the two load curves is used.	E10.0
11-20		Tensile mean stress at which the yield stress follows load curve ID.	E10.0

The stress strain behavior follows a different curve in compression than it does in tension. Compression and tension is determined by the sign of the mean stress. Two curves must be defined giving the yield stress versus effective plastic strain for both the tension and compression regimes.

Strain rate may be accounted for using the Cowper and Symonds model which scales the yield stress with the factor

$$1+\left(\frac{\dot{\varepsilon}}{C}\right)^{1/p}$$

where $\dot{\varepsilon}$ is the strain rate.

Material Type 126 (Modified Honeycomb)

Columns		Quantity	Format
1-10	Card 3	Young's modulus (for honeycomb material), E	E10.0
11-20		Poisson's ratio (for honeycomb material), ν	E10.0
21-30		Yield stress for fully compacted honeycomb material, σ_y	E10.0
31-40		LCA, load curve number for sigma-aa versus normal strain-aa **(See Figure 3.31.).** For the corotational solid element, type 9, engineering strain is expected, but for all other solid element formulations logarithmic strain is used. <u>When switching between element formulations to type 9, the curves must be redefined to account for the change in strain measue.</u>	E10.0
41-50		LCB, load curve number for sigma-bb versus normal strain-bb. (default: LCB=LCA). For the corotational solid element, type 9, engineering strain is expected, but for all other solid element formulations logarithmic strain is used.	E10.0
51-60		LCC, load curve number for sigma-cc versus normal strain-cc. (default: LCC=LCA). For the corotational solid element, type 9, engineering strain is expected, but for all other solid element formulations logarithmic strain is used.	E10.0
61-70		LCS, load curve number for shear stress versus either relative volume or volumetric strain. (default LCS=LCA). Each component of shear stress may have its own load curve via Card 5 input.	E10.0
71-80		Relative volume at which the honeycomb is fully compacted, V_f	E10.0
	Card 4	The following honeycomb parameters must be defined for there are no defaults.	
1-10		Elastic modulus E_{aau} in uncompressed configuration	E10.0
11-20		Elastic modulus E_{bbu} in uncompressed configuration	E10.0
21-30		Elastic modulus E_{ccu} in uncompressed configuration	E10.0
31-40		Elastic shear modulus G_{abu} in uncompressed configuration	E10.0

Columns		Quantity	Format
41-50		Elastic shear modulus G_{bcu} in uncompressed configuration	E10.0
51-60		Elastic shear modulus G_{cau} in uncompressed configuration	E10.0
61-70		μ, material viscosity coefficient. (default=.05)	E10.0
71-80		Bulk viscosity flag. EQ.0.0: bulk viscosity is not used. This is recommended. EQ.1.0: bulk viscosity is active and μ=0 This will give results identical to previous versions of LS-DYNA .	E10.0
1-10	Card 5	LCAB, load curve number for sigma-ab versus shear strain-ab (default: LCAB=LCS). For the corotational solid element, type 9, engineering strain is expected, but for all other solid element formulations a shear strain based on the deformed configuration is used.	E10.0
11-20		LCBC, load curve number for sigma-bc versus shear strain-bc (default: LCBC=LCS). For the corotational solid element, type 9, engineering strain is expected, but for all other solid element formulations a shear strain based on the deformed configuration is used.	E10.0
21-30		LCCA, load curve number for sigma-ca versus shear strain-ca (default: LCCA=LCS). For the corotational solid element, type 9, engineering strain is expected, but for all other solid element formulations a shear strain based on the deformed configuration is used.	E10.0
31-40		LCSR, optional load curve number for strain-rate effects	E10.0
41-50		Tensile strain at element failure (element will erode)	E10.0
51-60		Shear strain at element failure (element will erode)	E10.0
1-10	Card 6	Material axes option, AOPT EQ.0.0: locally orthotropic with material axes determined by element nodes as shown in Figure 3.5. Cards 7 and 8 are blank with this option. EQ.1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center. Card 8 below is blank.	E10.0

Columns		Quantity	Format
		EQ.2.0: globally orthotropic with material axes determined by vectors defined on Cards 7 and 8	
1-10	Card 7	x_p, define for AOPT = 1.0	E10.0
11-20		y_p, define for AOPT = 1.0	E10.0
21-30		z_p, define for AOPT = 1.0	E10.0
1-10	Card 7	a_1, define for AOPT = 2.0	E10.0
11-20		a_2, define for AOPT = 2.0	E10.0
21-30		a_3, define for AOPT = 2.0	E10.0
1-10	Card 8	d_1, define for AOPT = 2.0	E10.0
11-20		d_2, define for AOPT = 2.0	E10.0
21-30		d_3, define for AOPT = 2.0	E10.0

For efficiency it is strongly recommended that the load curve ID’s: LCA, LCB, LCC, LCS, LCAB, LCBC, and LCCA, contain exactly the same number of points with corresponding strain values on the abcissa. If this recommendation is followed the cost of the table lookup is insignificant. Conversely, the cost increases significantly if the abcissa strain values are not consistent between load curves.

The behavior before compaction is orthotropic where the components of the stress tensor are uncoupled, i.e., an a component of strain will generate resistance in the local *a*-direction with no coupling to the local *b* and *c* directions. The elastic modulii vary from their initial values to the fully compacted values linearly with the relative volume:

$$E_{aa} = E_{aau} + \beta_{aa}(E - E_{aau}) \qquad G_{ab} = G_{abu} + \beta(G - G_{abu})$$

$$E_{bb} = E_{bbu} + \beta_{bb}(E - E_{bbu}) \qquad G_{bc} = G_{bcu} + \beta(G - G_{bcu})$$

$$E_{cc} = E_{ccu} + \beta_{cc}(E - E_{ccu}) \qquad G_{ca} = G_{cau} + \beta(G - G_{cau})$$

where

$$\beta = \max\left[\min\left(\tfrac{1-V}{1-V_f}, 1\right), 0\right]$$

and G is the elastic shear modulus for the fully compacted honeycomb material

$$G = \frac{E}{2(1+\nu)} .$$

The relative volume, V, is defined as the ratio of the current volume over the initial volume, and typically, V=1 at the beginning of a calculation.

The load curves define the magnitude of the stress as the material undergoes deformation. The first value in the curve should be less than or equal to zero corresponding to tension and increase to full compaction. **Care should be taken when defining the curves so the extrapolated values do not lead to negative yield stresses.**

At the beginning of the stress update we transform each element's stresses and strain rates into the local element coordinate system. For the uncompacted material, the trial stress components are updated using the elastic interpolated modulii according to:

$$\sigma_{aa}^{n+1^{trial}} = \sigma_{aa}^{n} + E_{aa}\Delta\varepsilon_{aa} \qquad \sigma_{ab}^{n+1^{trial}} = \sigma_{ab}^{n} + 2G_{ab}\Delta\varepsilon_{ab}$$

$$\sigma_{bb}^{n+1^{trial}} = \sigma_{bb}^{n} + E_{bb}\Delta\varepsilon_{bb} \qquad \sigma_{bc}^{n+1^{trial}} = \sigma_{bc}^{n} + 2G_{bc}\Delta\varepsilon_{bc}$$

$$\sigma_{cc}^{n+1^{trial}} = \sigma_{cc}^{n} + E_{cc}\Delta\varepsilon_{cc} \qquad \sigma_{ca}^{n+1^{trial}} = \sigma_{ca}^{n} + 2G_{ca}\Delta\varepsilon_{ca}$$

We then independently check each component of the updated stresses to ensure that they do not exceed the permissible values determined from the load curves, e.g., if

$$\left|\sigma_{ij}^{n+1^{trial}}\right| > \lambda\sigma_{ij}(\varepsilon_{ij})$$

then

$$\sigma_{ij}^{n+1} = \sigma_{ij}(\varepsilon_{ij})\frac{\lambda\sigma_{ij}^{n+1^{trial}}}{\left|\sigma_{ij}^{n+1^{trial}}\right|}$$

On Card 3 $\sigma_{ij}(\varepsilon_{ij})$ is defined in the load curve specified in columns 31-40 for the aa stress component, 41-50 for the bb component, 51-60 for the cc component, and 61-70 for the ab, bc, cb shear stress components. The parameter λ is either unity or a value taken from the load curve number, LCSR, that defines λ as a function of strain-rate. Strain-rate is defined here as the Euclidean norm of the deviatoric strain-rate tensor.

For fully compacted material we assume that the material behavior is elastic-perfectly plastic and updated the stress components according to:

$$s_{ij}^{trial} = s_{ij}^{n} + 2G\Delta\varepsilon_{ij}^{dev^{n+1/2}}$$

where the deviatoric strain increment is defined as

$$\Delta\varepsilon_{ij}^{dev} = \Delta\varepsilon_{ij} - \tfrac{1}{3}\Delta\varepsilon_{kk}\delta_{ij} \ .$$

We now check to see if the yield stress for the fully compacted material is exceeded by comparing

$$s_{eff}^{trial} = \left(\tfrac{3}{2} s_{ij}^{trial} s_{ij}^{trial}\right)^{1/2}$$

the effective trial stress to the yield stress, σ_y (Card 3, field 21-30). If the effective trial stress exceeds the yield stress we simply scale back the stress components to the yield surface

$$s_{ij}^{n+1} = \frac{\sigma_y}{s_{eff}^{trial}} s_{ij}^{trial} \ .$$

We can now update the pressure using the elastic bulk modulus, K

$$p^{n+1} = p^{n} - K\Delta\varepsilon_{kk}^{n+1/2}$$

$$K = \frac{E}{3(1-2\nu)}$$

and obtain the final value for the Cauchy stress

$$\sigma_{ij}^{n+1} = s_{ij}^{n+1} - p^{n+1}\delta_{ij} \ .$$

After completing the stress update we transform the stresses back to the global configuration.

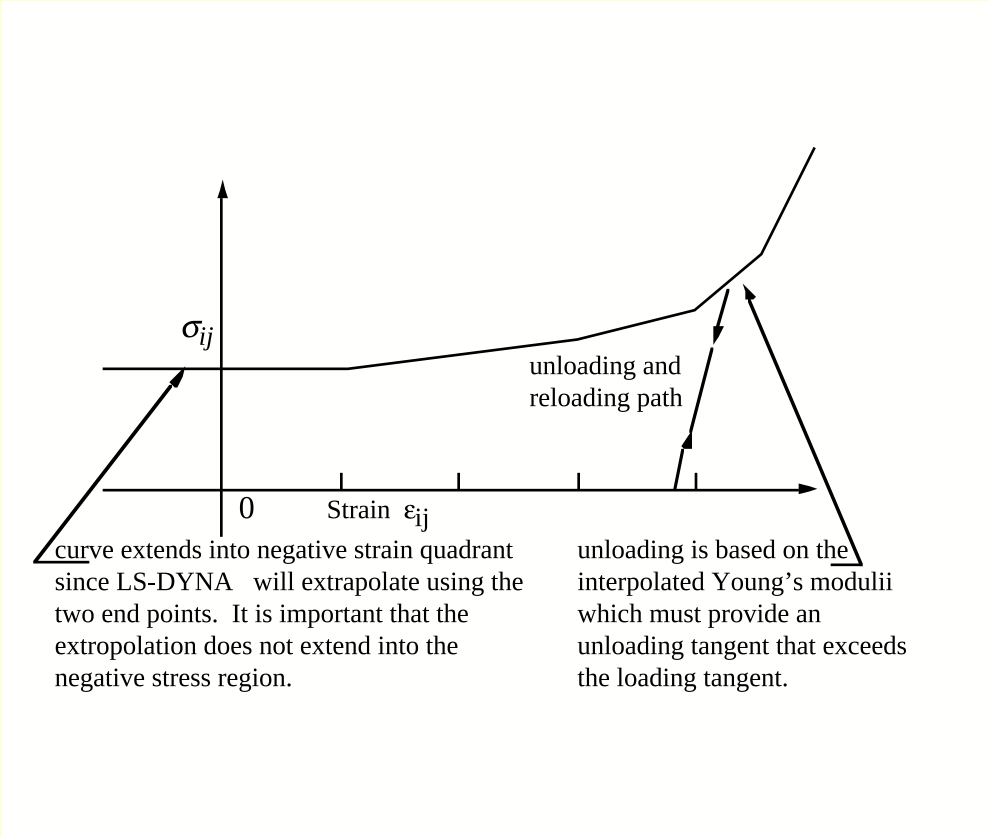


Figure 3.31. Stress quantity versus strain. Note that the "yield stress" at a strain of zero is nonzero. In the load curve definition the "time" value is the directional strain and the "function" value is the yield stress.

Material Type 127 (Arruda-Boyce Hyperviscoelastic Rubber)

This material model provides a rubber model optionally combined with linear viscoelasticity. The rubber model is described in the paper by Arruda and Boyce[1993].

Columns		Quantity	Format
1-10	Card 1	Bulk modulus	E10.0
11-20		Shear modulus, $nk\theta$	E10.0
21-30		N, number of statistical links.	E10.0
31-40		Blank	E10.0
	Card 4	Blank	
1-10	Card 5	Load curve ID if constants $\beta\iota$ are determined via a least squares fit. This relaxation curve is shown in Figure 3.25. This model ignores the constant stress	E10.0
11-20		TRAMP, optional ramp time for loading.	E10.0
21-30		Number of Prony series terms in fit. If zero, the default is 6. Currently, the maximum number is 6. Values less than 6, possibly 3-5 are recommended, since each term used adds significantly to the cost. Caution should be exercised when taking the results from the fit. Always check the results of the fit in the output file. Preferably, all generated coefficients should be positive. Negative values may lead to unstable results. Once a satisfactory fit has been achieved it is recommended that the coefficients which are written into the output file be input in future runs.	E10.0
	Card 6	Blank	

Columns		Quantity	Format

If the viscous effects are active and if a load curve ID is not defined on card 5 giving the relaxation data, then define the following input on cards 7 and 8. If not leave cards 7 and 8 blank. Up to 6 pairs may be defined.

1-10	Card 7	G_1, Maxwell consant, optional	E10.0
11-20		β_1, decay constant, optional	E10.0
21-30		G_2 , Maxwell consant	E10.0
31-40		β_2, decay constant	E10.0
41-50		G_3, Maxwell consant	E10.0
51-60		β_3 , decay constant	E10.0
61-70		G_4, Maxwell consant	E10.0
71-80		β_4, decay constant	E10.0
1-10	Card 8	G_5, Maxwell consant, optional	E10.0
11-20		β_5, decay constant, optional	E10.0
21-30		G_6 , Maxwell consant	E10.0
31-40		β_6, decay constant	E10.0

Rubber is generally considered to be fully incompressible since the bulk modulus greatly exceeds the shear modulus in magnitude. To model the rubber as an unconstrained material a hydrostatic work term, $W_H(J)$, is included in the strain energy functional which is function of the relative volume, J, [Ogden, 1984]:

$$W(J_1,J_2,J) = nk\theta\left[\tfrac{1}{2}(J_1-3)+\tfrac{1}{20N}(J_1^2-9)+\tfrac{11}{1050N^2}(J_1^3-27)\right]$$

$$+nk\theta\left[\tfrac{19}{7000N^3}(J_1^4-81)+\tfrac{519}{673750N^4}(J_1^5-243)\right]+W_H(J)$$

$$J_1 = I_1 J^{-1/3}$$

$$J_2 = I_2 J$$

where the hydrostatic work term is in terms of the bulk modulus, K, and the third invarient J, as:

$$W_H(J) = \frac{K}{2}(J-1)^2$$

Rate effects are taken into accounted through linear viscoelasticity by a convolution integral of the form:

$$\sigma_{ij} = \int_0^t g_{ijkl}(t-\tau)\frac{\partial \varepsilon_{kl}}{\partial \tau}d\tau$$

or in terms of the second Piola-Kirchhoff stress, S_{ij}, and Green's strain tensor, E_{ij},

$$S_{ij} = \int_0^t G_{ijkl}(t-\tau)\frac{\partial E_{kl}}{\partial \tau}d\tau$$

where $g_{ijkl}(t-\tau)$ and $G_{ijkl}(t-\tau)$ are the relaxation functions for the different stress measures. This stress is addedto the stress tensor determined from the strain energy functional.

If we wish to include only simple rate effects, the relaxation function is represented by six terms from the Prony series:

$$g(t) = \alpha_0 + \sum_{m=1}^{N} \alpha_m e^{-\beta t}$$

given by,

$$g(t) = \sum_{i=1}^{n} G_i e^{-\beta_i t}$$

This model is effectively a Maxwell fluid which consists of a dampers and springs in series. We characterize this in the input by shear modulii, G_i, and decay constants, β_i. The viscoelastic behavior is optional and an arbitrary number of terms may be used.

Material Type 128 (Heart Tissue)

This material model provides a tissue model described in the paper by Guccione, McCulloch, and Waldman[1991].

Columns	Quantity		Format
1-10	Card 1	C, material coefficient	E10.0
11-20		b_1 ,material coefficient	E10.0
21-30		b_2 ,material coefficient	E10.0
31-40		b_3 ,material coefficient	E10.0
41-50		K, bulk modulus.	E10.0
31-40		κ, maximum shear/bulk modulus ratio The acceptable range is [0,1], and the default is 0.1.	E10.0
	Card 4	Blank	
	Card 5	Blank	
1-10	Card 6	Material axes option, AOPT EQ.0.0: locally orthotropic with material axes determined by element nodes as shown in Figure 3.5. Cards 7 and 8 are blank with this option. EQ.1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center. Card 8 below is blank. EQ.2.0: globally orthotropic with material axes determined by vectors defined on Cards 7 and 8 EQ.3.0: This option determines locally orthotropic material axes by offsetting the material axes by an angle (Card 8) from a line in the plane of the shell determined by taking the cross product of the vector defined on Card 7 with the shell normal vector. In solid elements the normal vector is normal to the plane of the midsurface between the inner surface and outer surface defined by the first four nodes and the last four nodes of the connectivity of the element, respectively.	E10.0

Columns		Quantity	Format
		EQ.4.0: locally orthotropic in cylindrical coordinate system with material axes determined by the vector defined on Card 7 and the originating point, P, on Card 8.	
1-10	Card 7	x_p, define for AOPT = 1.0	E10.0
11-20		y_p, define for AOPT = 1.0	E10.0
21-30		z_p, define for AOPT = 1.0	E10.0
1-10	Card 7	a_1, define for AOPT = 2.0	E10.0
11-20		a_2, define for AOPT = 2.0	E10.0
21-30		a_3, define for AOPT = 2.0	E10.0
1-10	Card 7	v_1, define for AOPT = 3.0 & 4.0	E10.0
11-20		v_2, define for AOPT = 3.0 & 4.0	E10.0
21-30		v_3, define for AOPT = 3.0 & 4.0	E10.0
1-10	Card 8	d_1, define for AOPT = 2.0	E10.0
11-20		d_2, define for AOPT = 2.0	E10.0
21-30		d_3, define for AOPT = 2.0	E10.0
1-10	Card 8	Material angle beta, may be overridden on the element card (degrees)	E10.0
1-10	Card 8	P_1, define for AOPT = 4.0	E10.0
11-20		P_2, define for AOPT = 4.0	E10.0
21-30		P_3, define for AOPT = 4.0	E10.0

The tissue model is described in terms of the energy functional in terms of the Green strain components, E_{ij},

$$W(E,J) = \tfrac{C}{2}(e^Q - 1)$$

$$Q = b_1 E_{11}^2 + b_2(E_{22}^2 + E_{33}^2 + E_{23}^2 + E_{32}^2) + b_3(E_{12}^2 + E_{21}^2 + E_{13}^2 + E_{31}^2)$$

The Second Piola-Kirchhoff stress is evaluated from the energy functional by

$$S = \frac{\partial W}{\partial E}$$

and transformed to the Cauchy stress by

$$\sigma^* = \tfrac{1}{J} FSF^T .$$

The pressure acts as a penalty term to enforce incompressibility.

$$\begin{aligned} &\sigma^* = \sigma^* - \left(p + \mathrm{tr}(\sigma^*)\right) I \\ &p = -K(J-1) \end{aligned}$$

and to ensure the penalty term is effective, the ratio of the effective shear modulus, $\tfrac{C}{2} e^Q$, to the bulk modulus is limited by

$$\tfrac{C}{2} e^Q \le \kappa K$$

. Material Type 129 (Isotropic Lung Tissue)

This material model provides a lung tissue model described in the paper by D. Vawter [1980].

Columns		Quantity	Format
1-10	Card 1	K, bulk modulus.	E10.0
11-20	Card 1	C, material coefficient	E10.0
21-30		Δ , material coefficient	E10.0
31-40		α , material coefficient	E10.0
41-50		β , material coefficient	E10.0
51-60		C1, material coefficient	E10.0
61-70		C2, material coefficient	E10.0
	Card 4	Blank	
1-10	Card 5	Load curve ID if constants $\beta\iota$ are determined via a least squares fit. This relaxation curve is shown in Figure 3.25. This model ignores the constant stress	E10.0
11-20		TRAMP, optional ramp time for loading.	E10.0
21-30		Number of Prony series terms in fit. If zero, the default is 6. Currently, the maximum number is 6. Values less than 6, possibly 3-5 are recommended, since each term used adds significantly to the cost. Caution should be exercised when taking the results from the fit. Always check the results of the fit in the output file. Preferably, all generated coefficients should be positive. Negative values may lead to unstable results. Once a satisfactory fit has been achieved it is recommended that the coefficients which are written into the output file be input in future runs.	E10.0
	Card 6	Blank	

Columns		Quantity	Format

If the viscous effects are active and if a load curve ID is not defined on card 5 giving the relaxation data, then define the following input on cards 7 and 8. If not leave cards 7 and 8 blank. Up to 6 pairs may be defined.

1-10	Card 7	G_1, Maxwell consant, optional	E10.0
11-20		β_1, decay constant, optional	E10.0
21-30		G_2 , Maxwell consant	E10.0
31-40		β_2, decay constant	E10.0
41-50		G_3, Maxwell consant	E10.0
51-60		β_3 , decay constant	E10.0
61-70		G_4, Maxwell consant	E10.0
71-80		β_4, decay constant	E10.0
1-10	Card 8	G_5, Maxwell consant, optional	E10.0
11-20		β_5, decay constant, optional	E10.0
21-30		G_6 , Maxwell consant	E10.0
31-40		β_6, decay constant	E10.0

The material is described by a strain energy functional expressed in terms of the invarients of the Green Strain:

$$W(I_1, I_2) = \frac{C}{2\Delta} e^{(\alpha I_1^2 + \beta I_2)} + \frac{12C_1}{\Delta(1+C_2)} \left[A^{(1+C_2)} - 1 \right]$$

$$A^2 = \tfrac{4}{3}\left(I_1 + I_2\right) - 1$$

where the hydrostatic work term is in terms of the bulk modulus, K, and the third invarient J, as:

$$W_H(J) = \frac{K}{2}(J-1)^2$$

Rate effects are taken into accounted through linear viscoelasticity by a convolution integral of the form:

$$\sigma_{ij} = \int_0^t g_{ijkl}(t-\tau)\frac{\partial \varepsilon_{kl}}{\partial \tau}d\tau$$

or in terms of the second Piola-Kirchhoff stress, S_{ij}, and Green's strain tensor, E_{ij},

$$S_{ij} = \int_0^t G_{ijkl}(t-\tau)\frac{\partial E_{kl}}{\partial \tau}d\tau$$

where $g_{ijkl}(t-\tau)$ and $G_{ijkl}(t-\tau)$ are the relaxation functions for the different stress measures. This stress is addedto the stress tensor determined from the strain energy functional.

If we wish to include only simple rate effects, the relaxation function is represented by six terms from the Prony series:

$$g(t) = \alpha_0 + \sum_{m=1}^{N} \alpha_m e^{-\beta t}$$

given by,

$$g(t) = \sum_{i=1}^{n} G_i e^{-\beta_i t}$$

This model is effectively a Maxwell fluid which consists of a dampers and springs in series. We characterize this in the input by shear modulii, G_i, and decay constants, β_i. The viscoelastic behavior is optional and an arbitrary number of terms may be used.

Material Type 130 (Special Orthotropic Model)

This model is available the Belytschko-Tsay and the C0 triangular shell elements and is based on a resultant stress formulation. In-plane behavior is treated separately from bending in order to model perforated materials such as television shadow masks. If other shell formulations are specified, the formulation will be automatically switched to Belyschko-Tsay.

Columns		Quantity	Format
1-10	Card 3	E_{11p}, for in plane behavior	E10.0
11-20		E_{22p}, for in plane behavior	E10.0
21-30		ν_{12p}, for in plane behavior	E10.0
31-40		ν_{21p}, for in plane behavior	E10.0
41-50		G_{12p}, for in plane behavior	E10.0
51-60		G_{23p}, for in plane behavior	E10.0
61-70		G_{31p}, for in plane behavior	E10.0
1-10	Card 4	E_{11b}, for bending behavior	E10.0
11-20		E_{22b}, for bending behavior	E10.0
21-30		ν_{12b}, for bending behavior	E10.0
31-40		ν_{21b}, for bending behavior	E10.0
41-50		G_{12b}, for bending behavior	E10.0
	Card 5	Blank	
1-10	Card 6	Material axes option, AOPT EQ.0.0: locally orthotropic with material axes determined by element nodes as shown in Figure 3.5. Cards 7 and 8 are blank with this option. EQ.1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center. Card 8 below is blank. EQ.2.0: globally orthotropic with material axes determined by vectors defined on Cards 7 and 8	E10.0

Material Type 130 **(Special Orthotropic)**

Columns		Quantity	Format
		EQ.3.0: This option determines locally orthotropic material axes by offsetting the material axes by an angle (Card 8) from a line in the plane of the shell determined by taking the cross product of the vector defined on Card 7 with the shell normal vector. In solid elements the normal vector is normal to the plane of the midsurface between the inner surface and outer surface defined by the first four nodes and the last four nodes of the connectivity of the element, respectively.	
1-10	Card 7	x_p, define for AOPT = 1.0	E10.0
11-20		y_p, define for AOPT = 1.0	E10.0
21-30		z_p, define for AOPT = 1.0	E10.0
1-10	Card 7	a_1, define for AOPT = 2.0	E10.0
11-20		a_2, define for AOPT = 2.0	E10.0
21-30		a_3, define for AOPT = 2.0	E10.0
1-10	Card 7	v_1, define for AOPT = 3.0 & 4.0	E10.0
11-20		v_2, define for AOPT = 3.0 & 4.0	E10.0
21-30		v_3, define for AOPT = 3.0 & 4.0	E10.0
1-10	Card 8	d_1, define for AOPT = 2.0	E10.0
11-20		d_2, define for AOPT = 2.0	E10.0
21-30		d_3, define for AOPT = 2.0	E10.0
1-10	Card 8	Material angle beta, may be overridden on the element card (degrees)	E10.0

The in-plane elastic matrix for in-plane, plane stress behavior is given by:

$$C_{in\,plane} = \begin{bmatrix} Q_{11p} & Q_{12p} & 0 & 0 & 0 \\ Q_{12p} & Q_{22p} & 0 & 0 & 0 \\ 0 & 0 & Q_{44p} & 0 & 0 \\ 0 & 0 & 0 & Q_{55p} & 0 \\ 0 & 0 & 0 & 0 & Q_{66p} \end{bmatrix}$$

The terms Q_{ijp} are defined as:

$$
\begin{aligned}
Q_{11p} &= \frac{E_{11p}}{1-\nu_{12p}\nu_{21p}} \\
Q_{22p} &= \frac{E_{22p}}{1-\nu_{12p}\nu_{21p}} \\
Q_{12p} &= \frac{\nu_{12p}E_{11p}}{1-\nu_{12p}\nu_{21p}} \\
Q_{44p} &= G_{12p} \\
Q_{55p} &= G_{23p} \\
Q_{66p} &= G_{31p}
\end{aligned}
$$

The in plane elastic matrix for bending behavior is given by:

$$
C_{bending} = \begin{bmatrix} Q_{11b} & Q_{12b} & 0 \\ Q_{12b} & Q_{22b} & 0 \\ 0 & 0 & Q_{44b} \end{bmatrix}
$$

The terms Q_{ijp} are similarly defined.

Material Type 134 (Viscoelastic Fabric)

The viscoelastic fabric model is a variation on the general viscoelastic model of material 76. This model is valid for 3 and 4 node membrane elements only and is strongly recommended for modeling isotropic viscoelastic fabrics where wrinkling may be a problem. For thin fabrics, buckling can result in an inability to support compressive stresses; thus, a flag is included for this option. If bending stresses are important use a shell formulation with model 76.

Columns		Quantity	Format
1-10	Card 3	K, constant elastic bulk modulus. If the bulk behavior is viscoelastic, then this modulus is used in determining the contact interface stiffnesses only.	E10.0
11-20		Compressive stress flag (default = 0.0). This option does not apply to the liner. EQ.0.0: don't eliminate compressive stresses EQ.1.0: eliminate compressive stresses	E10.0
1-10	Card 4	G_1, Maxwell consant. If a relaxation curve is given on card 6 below, cards 4 and 5 may be blank.	E10.0
11-20		β_1, decay constant	E10.0
21-30		G_2 , Maxwell consant	E10.0
31-40		β_2, decay constant	E10.0
41-50		G_3, Maxwell consant	E10.0
51-60		β_3 , decay constant	E10.0
61-70		G_4, Maxwell consant	E10.0
71-80		β_4, decay constant	E10.0
1-10	Card 5	G_5, Maxwell consant	E10.0
11-20		β_5, decay constant	E10.0
21-30		G_6 , Maxwell consant	E10.0
31-40		β_6, decay constant	E10.0

Columns		Quantity	Format
1-10	Card 6	Load curve ID if constants, G_i, and β_i are determined via a least squares fit. This relaxation curve is shown in Figure 3.25.	E10.0
11-20		Number of terms in fit. If zero the default is 6. Currently, the maximum number is set to 6.	E10.0
21-30		BSTART. In the fit, β_1 is set to zero, β_2 is set to BSTART, β_3 is 10 times β_2 , β_4 is 100 times greater than β_3 , and so on. If zero, BSTART= .01.	E10.0
31-40		TRAMP, optional ramp time for loading.	E10.0

If there is no volumetric relaxation then insert two blank cards:

	Card 7	Blank
	Card 8	Blank

For volumetric relaxation define the following information:

41-50	Card 6	Load curve ID if constants, K_i, and $\beta\kappa_i$ are determined via a least squares fit. This relaxation curve is shown in Figure 3.25.	E10.0
51-60		Number of terms in fit. If zero the default is 6. Currently, the maximum number is set to 6.	E10.0
61-70		Number of terms in fit. If zero, the default is 6. Currently, the maximum number is 6. Values less than 6, possibly 3-5 are recommended, since each term used adds significantly to the cost. Caution should be exercised when taking the results from the fit. Always check the results of the fit in the output file. Preferably, all generated coefficients should be positive-they generally will be. Negative values may lead to unstable results. Once a satisfactory fit has been achieved it is recommended that the coefficients which are written into the output file be input in future runs.	E10.0
61-70		Blank	E10.0
71-80		TRAMP, optional ramp time for loading.	E10.0

Columns		Quantity	Format
1-10	Card 7	K_1, Maxwell consant. If a relaxation curve is given on card 6 above, cards 7 and 8 may be blank.	E10.0
11-20		$\beta\kappa_1$, decay constant	E10.0
21-30		K_2 , Maxwell consant	E10.0
31-40		$\beta\kappa_2$, decay constant	E10.0
41-50		K_3, Maxwell consant	E10.0
51-60		$\beta\kappa_3$, decay constant	E10.0
61-70		K_4, Maxwell consant	E10.0
71-80		$\beta\kappa_4$, decay constant	E10.0
1-10	Card 8	K_5, Maxwell consant	E10.0
11-20		$\beta\kappa_5$, decay constant	E10.0
21-30		K_6 , Maxwell consant	E10.0
31-40		$\beta\kappa_6$, decay constant	E10.0

Rate effects are taken into accounted through linear viscoelasticity by a convolution integral of the form:

$$\sigma_{ij} = \int_0^t g_{ijkl}(t-\tau)\frac{\partial \varepsilon_{kl}}{\partial \tau} d\tau$$

where $g_{ijkl}(t-\tau)$ is the relaxation function.

If we wish to include only simple rate effects for the deviatoric stresses, the relaxation function is represented by six terms from the Prony series:

$$g(t) = \sum_{m=1}^{N} G_m e^{-\beta_m t}$$

We characterize this in the input by shear modulii, G_i, and decay constants, β_i. An arbitrary number of terms, up to 6, may be used when applying the viscoelastic model.

For volumetric relaxation, the relaxation function is also represented by the Prony series in terms of bulk modulii:

$$k(t) = \sum_{m=1}^{N} K_m e^{-\beta_{k_m} t}$$

Equations of State

Equation-of-State Form 1 (Linear Polynomial)

Columns		Quantity	Format
1-10	Card 10	C_0	E10.0
11-20		C_1	E10.0
21-30		C_2	E10.0
31-40		C_3	E10.0
41-50		C_4	E10.0
51-60		C_5	E10.0
61-70		C_6	E10.0
71-80		E_0, initial internal energy	E10.0
1-10	Card 11	V_0, initial relative volume	E10.0

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.

Equation-of-State Form 2 (JWL)

Columns		Quantity	Format
1-10	Card	A	E10.0
11-20		B	E10.0
21-30		R_1	E10.0
31-40		R_2	E10.0
41-50		ω	E10.0
51-60		E_0	E10.0
61-70		V_0	E10.0

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.

Equation-of-State Form 3 (Sack "Tuesday")

Columns	Quantity		Format
1-10	Card 10	A_1	E10.0
11-20		A_2	E10.0
21-30		A_3	E10.0
31-40		B_1	E10.0
41-50		B_2	E10.0
51-60		E_0, initial internal energy	E10.0
61-70		V_0, initial relative volume	E10.0

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.

Equation-of-State Form 4 (Gruneisen)

Columns		Quantity	Format
1-10	Card 10	C	E10.0
11-20		S_1	E10.0
21-30		S_2	E10.0
31-40		S_3	E10.0
41-50		γ_0	E10.0
51-60		a	E10.0
61-70		E_0, initial internal energy	E10.0
71-80		V_0, initial relative volume	E10.0

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 u_s-u_p curve; S_1, S_2, and S_3 are the coefficients of the slope of the u_s-u_p curve; γ_0 is the Gruneisen gamma; a is the first order volume correction to γ_0 and $\mu = \frac{\rho}{\rho_0} - 1.$

Equation-of-State Form 5 (Ratio of Polynomials)

Columns		Quantity	Format
1-16	Card 10	A_{10}	E16.0
17-32		A_{11}	E16.0
33-48		A_{12}	E16.0
49-64		A_{13}	E16.0
1-16	Card 11	A_{20}	E16.0
17-32		A_{21}	E16.0
33-48		A_{22}	E16.0
49-64		A_{23}	E16.0
1-16	Card 12	A_{30}	E16.0
17-32		A_{31}	E16.0
33-48		A_{32}	E16.0
49-64		A_{33}	E16.0
1-16	Card 13	A_{40}	E16.0
17-32		A_{41}	E16.0
33-48		A_{42}	E16.0
49-64		A_{43}	E16.0
1-16	Card 14	A_{50}	E16.0
17-32		A_{51}	E16.0
33-48		A_{52}	E16.0
49-64		A_{53}	E16.0
1-16	Card 15	A_{60}	E16.0
17-32		A_{61}	E16.0
33-48		A_{62}	E16.0
49-64		A_{63}	E16.0

Equation-of-State Form 5 (Ratio of Polynomials)

Columns		Quantity	Format
1-16	Card 16	A_{70}	E16.0
17-32		A_{71}	E16.0
33-48		A_{72}	E16.0
49-64		A_{73}	E16.0
1-16	Card 17	α	E16.0
17-32		β	E16.0
33-48		A_{14}	E16.0
49-64		A_{24}	E16.0
1-16	Card 18	E_0, initial internal energy	E16.0
17-32		V_0, initial relative volume	E16.0

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

$$F_i = \sum_{j=0}^{n} A_{ij}\,\mu^j \qquad \text{n} = 4 \text{ if } i < 3$$

$$\mu = \frac{\rho}{\rho_0} - 1 \qquad \text{n} = 3 \text{ if } i \geq 3$$

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.

Equation-of-State Form 6 (Linear Polynomial with Energy Leak)

Columns		Quantity	Format
1-10	Card 10	C_0	E10.0
11-20		C_1	E10.0
21-30		C_2	E10.0
31-40		C_3	E10.0
41-50		C_4	E10.0
51-60		C_5	E10.0
61-70		C_6	E10.0
71-80		E_0, initial internal energy	E10.0
1-10	Card 11	V_0, initial relative volume	E10.0
11-20		*CN*, number of time history curve that gives energy deposition rate.	E10.0

This polynomial equation of state, linear in the internal energy per initial volume, E, is given by

$$p = C_0 + C_1\mu + C_2\mu^2 + C_3\mu^3 + \left(C_4 + C_5\mu + C_6\mu^2\right)E \tag{17.1.1}$$

in which C_0, C_1, C_2, C_3, C_4, C_5, and C_6 are user defined constants and

$$\mu = \frac{1}{V} - 1 . \tag{17.1.2}$$

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.

Equation-of-State Form 7 (Ignition and Growth of Reaction in HE)

Columns		Quantity	Format
1-10	Card 10	a	E10.0
11-20		b	E10.0
21-30		xp_1	E10.0
31-40		xp_2	E10.0
41-50		frer	E10.0
1-10	Card 11	g	E10.0
11-20		r_1	E10.0
21-30		r_2	E10.0
31-40		r_3	E10.0
41-50		r_5	E10.0
1-10	Card 12	r_6	E10.0
11-20		fmxig	E10.0
21-30		freq	E10.0
31-40		growl	E10.0
41-50		em	E10.0
1-10	Card 13	ar_1	E10.0
11-20		es_1	E10.0
21-30		cvp (heat capacity of products)	E10.0
31-40		cvr (heat capacity of explosive)	E10.0
41-50		eetal	

Equation-of-State Form 7 (Ignition and Growth of Reaction in HE)

Columns		Quantity	Format
1-10	Card 14	ccrit	E10.0
11-20		Enq (heat of reaction)	E10.0
21-30		Tmp0 (initial temperature generally 298°K)	E10.0
31-40		Blank	
41-50		Blank	
1-10	Card 15	grow2	E10.0
11-20		ar_2	E10.0
21-30		es_2	E10.0
31-40		en	E10.0
41-50		fmxgr	E10.0
51-60		fmngr	E10.0

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

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

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 ($T_e=T_p$, $p_e=p_p$) and the relative volumes are additive, i.e.,

$$V = (1\text{-}F)\ V_e + V_p$$

The chemical reaction rate for conversion of unreacted explosive to reaction products consists of three physically realistic terms: an ignition term in which a small amount of explosive reacts soon after the shock wave compresses it; a slow growth of reaction as this initial reaction spreads; and a rapid completion of reaction at high pressure and temperature. The form of the reaction rate equation is

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

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

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

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. The standard inputs for PBX-9504 and LX-17 are given below.

Equation-of-State Form 7 (Ignition and Growth of Reaction in HE)

1. PBX-9504

Card 1	Material No	Material Type=10	Density = 1.842	EOS form = 7		
Card 2	PBX-9504 input					
Card 3	Shear modulus = 0.0454	Yield Strength = 0.002				
Card 4						
Card 5						
Card 6	Blank					
Card 7						
Card 8						
Card 9	PBX-9504 Ignition and Growth					
Columns	1-10	11-20	21-30	31-40	41-50	51-60
Card 10	8.524	0.1802	4.6	1.3	0.667	
Card 11	3.8e-6	9522.	-0.05944	2.4656e-5	14.1	
Card 12	1.41	0.3	7.43e+11	3.1	1.0	
Card 13	0.111	0.667	1.0e-5	2.7813e-5	20.0	
Card 14	0.0	0.102	298.0	-	-	
Card 15	400.0	1.0	0.333	2.0	0.5	0.0

2. LX -17

Card 1	Material No	Material type=10	Density=1.90	EOS Form=7		
Card 2	LX-17 input					
Card 3	Shear Modulus=0.0354	Yield Strength=0.002				
Card 4						
Card 5						
Card 6	Blank					
Card 7						
Card 8						
Card 9	LX-17 Ignition and Growth					
Columns	1-10	11-20	21-30	31-40	41-50	51-60
Card 10	5.31396	0.0270309	4.1	1.1	0.667	
Card 11	4.60e-6	778.1	-0.05031	2.2229e-5	11.3	
Card 12	l.13	0.5	4.0e+6	0.6	1.0	
Card 13	0.111	0.667	1.0e-5	2.487e-5	7.0	
Card 14	0.22	0.069	298.0	-	-	
Card 15	400.0	1.0	0.333	3.0	0.5	0.0

Equation-of-State Form 8 (Tabulated-Compaction)

Columns		Quantity	Format
1-16	Card 10	ε_{V1} (ln V)	E16.0
17-32		ε_{V2}	E16.0
33-48		ε_{V3}	E16.0
49-64		ε_{V4}	E16.0
65-80		ε_{V5}	E16.0
1-16	Card 11	ε_{V6}	E16.0
17-32		ε_{V7}	E16.0
33-48		ε_{V8}	E16.0
49-64		ε_{V9}	E16.0
65-80		ε_{V10}	E16.0
1-16	Card 12	C_1	E16.0
17-32		C_2	E16.0
33-48		C_3	E16.0
49-64		C_4	E16.0
65-80		C_5	E16.0
1-16	Card 13	C_6	E16.0
17-32		C_7	E16.0
33-48		C_8	E16.0
49-64		C_9	E16.0
65-80		C_{10}	E16.0

Equation-of-State Form 8 (Tabulated-Compaction)

Columns		Quantity	Format
1-16	Card 14	T_1	E16.0
17-32		T_2	E16.0
33-48		T_3	E16.0
49-64		T_4	E16.0
65-80		T_5	E16.0
1-16	Card 15	T_6	E16.0
17-32		T_7	E16.0
33-48		T_8	E16.0
49-64		T_9	E16.0
65-80		T_{10}	E16.0
1-16	Card 16	K_1	E16.0
17-32		K_2	E16.0
33-48		K_3	E16.0
49-64		K_4	E16.0
65-80		K_5	E16.0
1-16	Card 17	K_6	E16.0
17-32		K_7	E16.0
33-48		K_8	E16.0
49-64		K_9	E16.0
65-80		K_{10}	E16.0
1-16	Card 18	γ	E16.0
17-32		E_0, initial internal energy	E16.0
33-48		V_0, initial relative volume	E16.0

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 3.32. 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.

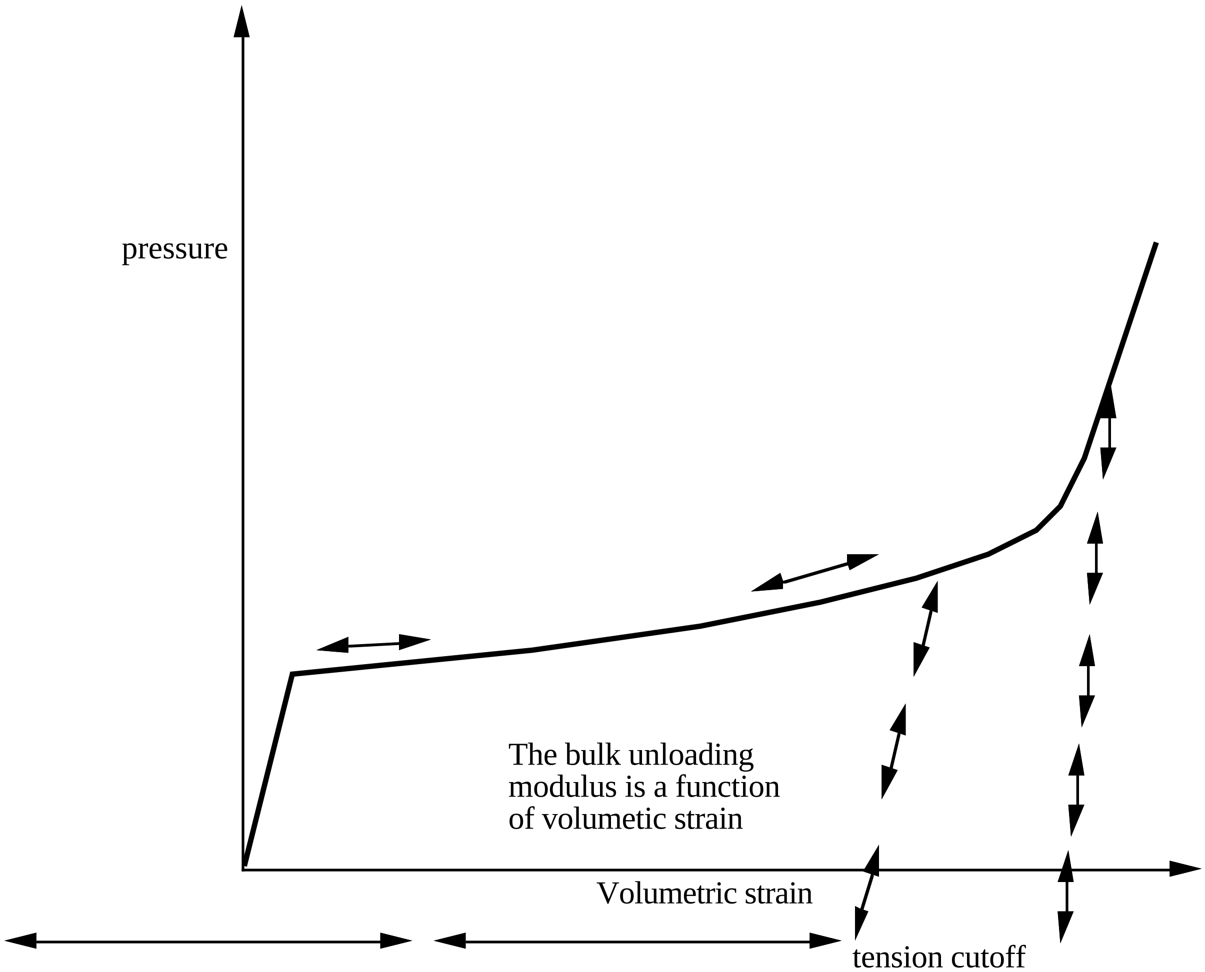


Figure 3.32. Pressure versus volumetric strain curve for equation-of-state Form 8 with compaction. In the compacted states the bulk unloading modulus depends on the peak volumetric strain.

Equation-of-State Form 9 (Tabulated)

Columns		Quantity	Format
1-16	Card 10	ε_{V1} (ln V)	E16.0
17-32		ε_{V3}	E16.0
33-48		ε_{V3}	E16.0
49-64		ε_{V4}	E16.0
65-80		ε_{V5}	E16.0
1-16	Card 11	ε_{V6}	E16.0
17-32		ε_{V7}	E16.0
33-48		ε_{V8}	E16.0
49-64		ε_{V9}	E16.0
65-80		ε_{V10}	E16.0
1-16	Card 12	C_1	E16.0
17-32		C_2	E16.0
33-48		C_3	E16.0
49-64		C_4	E16.0
65-80		C_5	E16.0
1-16	Card 13	C_6	E16.0
17-32		C_7	E16.0
33-48		C_8	E16.0
49-64		C_9	E16.0
65-80		C_{10}	E16.0

Equation-of-State Form 9 (Tabulated)

Columns		Quantity	Format
1-16	Card 14	T_1	E16.0
17-32		T_2	E16.0
33-48		T_3	E16.0
49-64		T_4	E16.0
65-80		T_5	E16.0
1-16	Card 15	T_6	E16.0
17-32		T_7	E16.0
33-48		T_8	E16.0
49-64		T_9	E16.0
65-80		T_{10}	E16.0
1-16	Card 16	γ	E16.0
17-32		E_0	E16.0
33-48		V_0	E16.0

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.

Equation-of-State Form 10 (Propellant-Deflagration)

This equation-of-state has been added to model airbag propellants.

Columns		Quantity	Format
1-10	Card 10	*a*, product JWL coefficient	E10.0
11-20		*b*, product JWL coefficient	E10.0
21-30		*xp*1, product JWL coefficient	E10.0
31-40		*xp*2, product JWL coefficient	E10.0
41-50		frer, unreacted Co-volume	E10.0
1-10	Card 11	*g*, product wC_V	E10.0
11-20		*r*1, unreacted JWL coefficient	E10.0
21-30		*r*2, unreacted JWL coefficient	E10.0
31-40		*r*3, unreacted $\mathrm{wC_V}$	E10.0
41-50		*r*5, unreacted JWL coefficient	E10.0
1-10	Card 12	*r*6, unreacted JWL coefficient	E10.0
11-20		*fmxig*, initial Fraction Reacted F_o	E10.0
21-30		*freq*, initial Pressure P_o	E10.0
31-40		*grow*1, first burn rate coefficient	E10.0
41-50		*em*, pressure Exponent (1^{st} term)	E10.0
1-10	Card 13	*ar*1, exponent on *F* (1^{st} term)	E10.0
11-20		*es*1, exponent on (1-*F*) (1^{st} term)	E10.0
21-30		*cvp*, heat capacity products	E10.0
31-40		*cvr*, heat capacity unreacted	E10.0
41-50		*eetal*, *extra*, not presently used	E10.0
1-10	Card 14	*ccrit*, product co-volume	E10.0
11-20		*enq*, heat of Reaction	E10.0
21-30		*tmp*0, initial Temperature (298°K)	E10.0

Equation-of-State Form 10 (Propellant)

Columns		Quantity	Format
1-10	Card 15	*grow*2, second burn rate coefficient	E10.0
11-20		*ar*2, exponent on *F* (2^{nd} term)	E10.0
21-30		*es*2, exponent on (1-*F*) (2^{nd} term)	E10.0
31-40		*en*, pressure Exponent (2^{nd} term)	E10.0
41-50		*fmxgr*, maximum *F* for 1^{st} term	E10.0
51-60		*fmngr*, minimum *F* for 2^{nd} term	E10.0

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 the are described by [Hallquist, et.al., 1990].

The unreacted propellant and the reaction product equations-of-state are both of the form:

$$p = Ae^{-R_1 V} + Be^{-R_2 V} + \frac{\omega C_v T}{V - d}$$

where p is pressure (in Mbars), V is the relative specific volume (inverse of relative density), ω is the Gruneisen coefficient, C_v is heat capacity (in Mbars -cc/cc°K), T is temperature in °K, d is the co-volume, and A, B, R_1 and R_2 are constants. Setting A=B=0. yields the van der Waal's co-volume equation-of-state. The JWL equation-of-state is generally useful at pressures above several kilobars, while the van der Waal's is useful at pressures below that range and above the range for which the perfect gas law holds. Of course, setting A=B=d=0 yields the perfect gas law. If accurate values of ω and C_v plus the correct distribution between "cold" compression and internal energies are used, the calculated temperatures are very reasonable and thus can be used to check propellant performance.

The reaction rate used for the propellant deflagration process is of the form:

$$\frac{\partial F}{\partial t} = \underset{\text{for } 0 < F < F_{limit1}}{Z(1-F)^y F^x p^w} + \underset{\text{for } F_{limit2} < F < 1}{V(1-F)^u Frp^s}$$

where F is the fraction reacted (F = 0 implies no reaction, F = 1 is complete reaction), t is time, and p is pressure (in Mbars), r,s,u,w,x,y, F_{limit1} and F_{limit2} are constants used to describe the pressure dependance and surface area dependence of the reaction rates. Two (or more) pressure dependent 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 P_p 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:

$$\frac{\partial F}{\partial t} = GROW1(p + freq)^{em}(F + fmxig)^{ar1}(1 - F + fmxig)^{es1}$$
$$+GROW2(p + freq)^{en}(F + fmxig$$

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.

Equation-of-State Form 11 (TENSOR Pore Collapse)
Card 1 (2I5,4E10.0)

Columns	Quantity	Format
1-5	Number of Virgin Loading Curve points (NLD)	I5
6-10	Number of Completely Crushed Curve points (NCR)	I5
11-20	Excess Compression required before any pores can collapse (μ_1)	E10.0
21-30	Excess Compression point where the Virgin Loading Curve and the Completely Crushed Curve intersect (μ_2)	E10.0
31-40	Initial Internal Energy	E10.0
41-50	Initial Excess Compression	E10.0

Virgin Loading Curve Definition
Card 2 through NLD+1 (2E15.0)

Columns	Quantity	Format
1-15	Excess Compression	E15.0
16-30	Pressure	E15.0

Completely Crushed Curve Definition
Card NLD+2 through NLD+NCR+1 (2E15.0)

Columns	Quantity	Format
1-15	Excess Compression	E15.0
16-30	Pressure	E15.0

The pore collapse model described in the TENSOR manual [Burton et al. 1982] 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 high quality link between them. The TENSOR model defines two curves, the virgin loading curve and the completely crushed curve as shown in Figure 3.33. 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

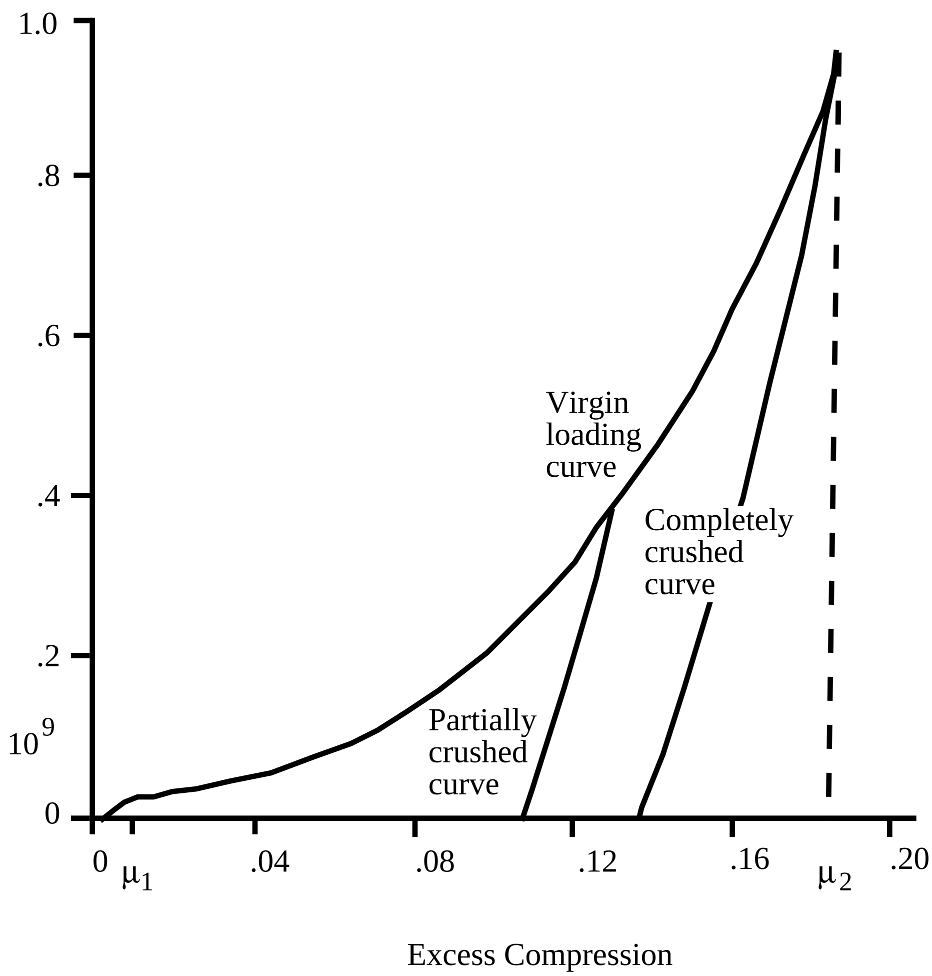


Figure 3.33. Pressure versus compaction curve.

determined. 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 *p*c 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 $V_{\min} = \frac{1}{1+\mu_{\max}}$ 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:

$$\frac{\partial P}{\partial \mu} = \frac{\partial P_{cc}\left(\frac{(1+\mu_B)(1+\mu)}{(1\mu_{\max})}\right)(1+\mu_B)}{\partial\mu\,(1+\mu_{\max})}$$

Simplifying,

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

where

$$\mu_a = \frac{\left(1+\mu_B\right)\left(1+\mu\right)}{\left(1+\mu_{\max}\right)} - 1 .$$

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.

Equation-of-State Form 14 (JWLB)

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.

Columns	Quantity		Format
1-10	Card 1	A_1	E10.0
11-20		A_2	E10.0
21-30		A_3	E10.0
31-40		A_4	E10.0
41-50		A_5	E10.0
1-10	Card 2	R_1	E10.0
11-20		R_2	E10.0
21-30		R_3	E10.0
31-40		R_4	E10.0
41-50		R_5	E10.0
1-10	Card 3	$A_{\lambda 1}$	E10.0
11-20		$A_{\lambda 2}$	E10.0
21-30		$A_{\lambda 3}$	E10.0
31-40		$A_{\lambda 4}$	E10.0
41-50		$A_{\lambda 5}$	E10.0

Equation-of-State Form 14 (JWLB)

Columns		Quantity	Format
1-10	Card 4	$B_{\lambda 1}$	E10.0
11-20		$B_{\lambda 2}$	E10.0
21-30		$B_{\lambda 3}$	E10.0
31-40		$B_{\lambda 4}$	E10.0
41-50		$B_{\lambda 5}$	E10.0
1-10	Card 5	$R_{\lambda 1}$	E10.0
11-20		$R_{\lambda 2}$	E10.0
21-30		$R_{\lambda 3}$	E10.0
31-40		$R_{\lambda 4}$	E10.0
41-50		$R_{\lambda 5}$	E10.0
1-10	Card 6	C	E10.0
11-20		ω	E10.0
21-30		E, energy density per unit initial volume	E10.0
31-40		V0, initial relative volume	E10.0

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

Material Property Data Cards (Thermal) - define 2 control cards

Define 2 control cards followed by material property definition cards for each material.

Card 1

Columns	Quantity	Format
1-10	Material type EQ.1: isotropic - define only k_1 EQ.2: orthotropic - define k_1, k_2, and k_3 EQ.3: isotropic temperature dependent EQ.4: orthotropic temperature dependent EQ.10: isotropic temperature dependent, properties defined by load curves	I10
11-20	Density	E10.0
21-30	Temperature at which latent heat is absorbed or released	E10.0
31-40	Latent heat	E10.0
41-45	Thermal generation rate curve number	I5
46-55	Thermal generation rate multiplier	E10.0
56-60	For orthotropic material define material axes option, AOPT EQ.0: locally orthotropic with material axes by element nodes N_1, N_2 and N_4. EQ.1: locally orthotropic with material axes deteramind by a point in space and global location of element center. EQ.2: globally orthrotorpic with material axes determined by vectors	I5

Card 2

Columns	Quantity	Format
1-80	Material identification name	10A8

Thermal Constitutive Models

Material Type 1 - Isotropic - define 1 card

Columns	Quantity	Format
1-10	Heat capacity	E10.0
11-20	Thermal conductivity k_1	E10.0

Material Type 2 - Orthotropic - define 3 cards

Columns		Quantity	Format
1-10	card 1	heat capacity	E10.0
11-20		thermal conductivity k_1	E10.0
21-30		thermal conductivity k_2	E10.0
31-40		thermal conductivity k_3	E10.0

if aopt=0, then include 2 blank cards

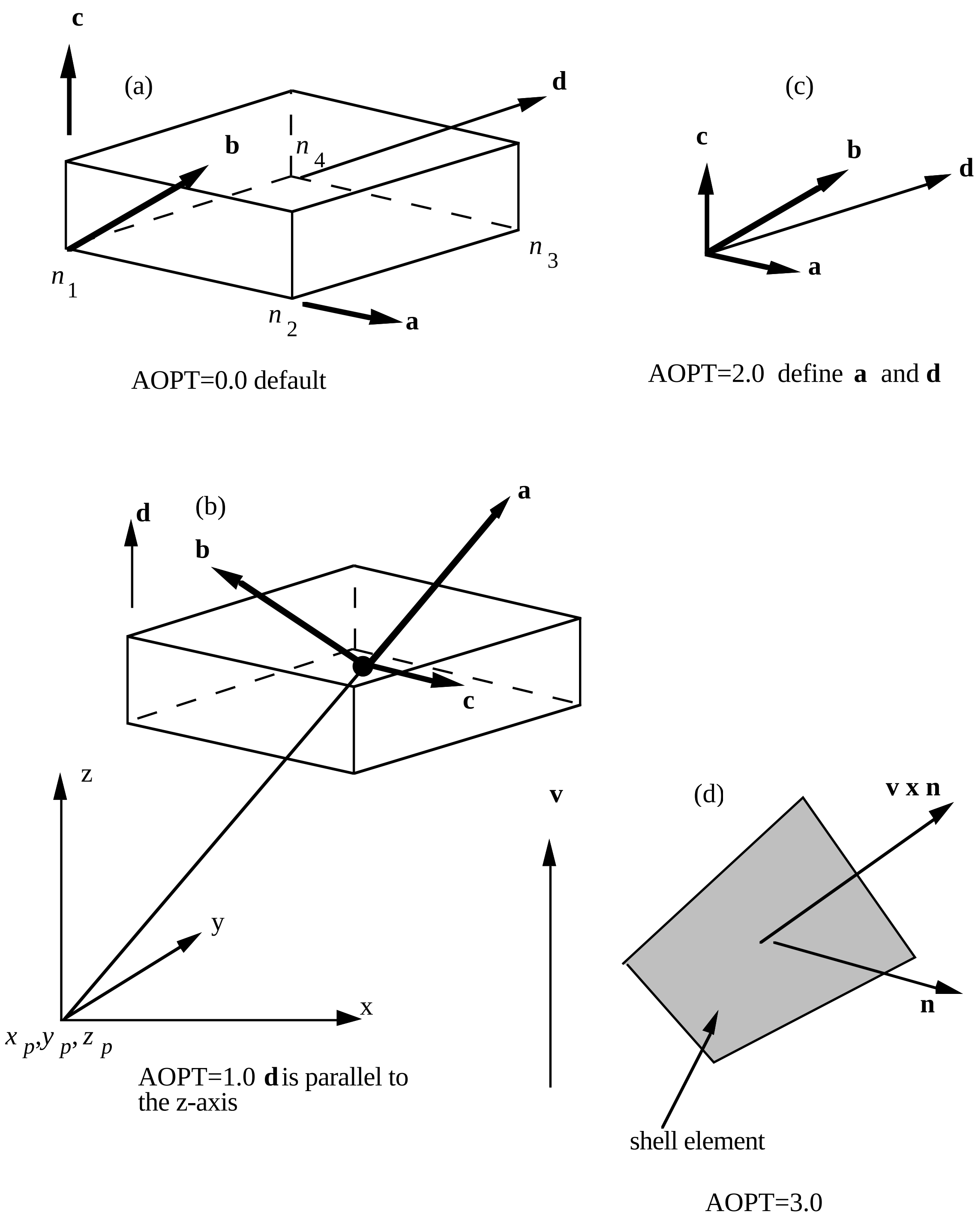
1-80	card 2	blank	80x
1-80	card 3	blank	80x

if aopt=1, then include the following 2 cards

1-10	card 2	x_p, define for aopt=1	E10.0
11-20		y_p, define for aopt=1	E10.0
21-30		z_p, define for aopt=1	E10.0
1-80	card 3	blank	80x

if aopt=2, then include the following 2 cards

1-10	card 2	a_1, define for aopt=2	E10.0
11-20		a_2, define for aopt=2	E10.0
21-30		a_3, define for aopt=2	E10.0
1-10	card 3	d_1, define for aopt=2	E10.0
11-20		d_2, define for aopt=2	E10.0
21-30		d_3, define for aopt=2	E10.0



Options for determining principal material axes: (a) AOPT = 0.0, (b) AOPT = 1.0, (c) AOPT = 2.0. Note that $\underset{\sim}{c} = \underset{\sim}{a} \times \underset{\sim}{d}$ and that $\underset{\sim}{b} = \underset{\sim}{c} \times \underset{\sim}{a}$.

Material Type 3 - Isotropic Temperature Dependent - define 3 cards

Columns	Quantity	Format
1-80	Card 1 Temperature $T_1, T_2, \ldots\ldots, T_8$	8E10.0
1-80	Card 2 Heat capacity $C_1, C_2, \ldots\ldots, C_8$	8E10.0
1-80	Card 3 Thermal conductivity $k_1, k_2, \ldots\ldots, k_8$	8E10.0

Material Type 4 - Orthotropic Temperature Dependent - define 7 cards

Columns		Quantity	Format
1-80	card 1	temperature $T_1, T_2, \ldots, T_8$	8E10.0
1-80	card 2	heat capacity $C_1, C_2, \ldots, C_8$	8E10.0
1-80	card 3	thermal conductivity $(k_1)_1, (k_1)_2, \ldots, (k_1)_8$	8E10.0
1-80	card 4	thermal conductivity $(k_2)_1, (k_2)_2, \ldots, (k_2)_8$	8E10.0
1-80	card 5	thermal conductivity $(k_3)_1, (k_3)_2, \ldots, (k_3)_8$	8E10.0

if aopt=0, then include 2 blank cards

1-80	card 6	blank	80x
1-80	card 7	blank	80x

if aopt=1, then include the following 2 cards

1-10	card 6	x_p, define for aopt=1	E10.0
11-20		y_p, define for aopt=1	E10.0
21-30		z_p, define for aopt=1	E10.0
1-80	card 7	blank	80x

if aopt=2, then include the following 2 cards

1-10	card 6	a_1, define for aopt=2	E10.0
11-20		a_2, define for aopt=2	E10.0
21-30		a_3, define for aopt=2	E10.0
1-10	card 7	d_1, define for aopt=2	E10.0
11-20		d_2, define for aopt=2	E10.0
21-30		d_3, define for aopt=2	E10.0

Material Type 10 - Isotropic temperature dependent - define 1 card

Columns	Quantity	Format
1-10	Heat capacity load curve number	I5
6-10	Thermal conductivity load curve number	I5

4. User Defined Integration Rules for Beams with Arbitrary Cross Sections

Define NUBIR (see Control Card 12, columns 36-40) card sets in this section.

Card 1 (I5,E10.0, I5)

Columns	Quantity	Format
1-5	Number of integration points, NIP, (If defined, ICST=0)	I5
6-15	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 Figure 4.1.	E10.0
16-20	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. 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	I5

Define the following card if ICST is nonzero, i.e., NIP=0.

Cards 2 (6E10.0)

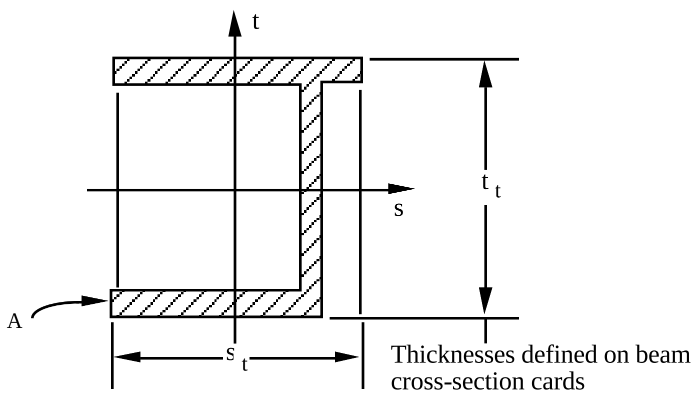
Columns	Quantity	Format
1-10	w, flange width	E10.0
11-20	t_f, flange thickness	E10.0
21-30	d, depth	E10.0
31-40	t_w, web thickness	E10.0
41-50	s_{ref}, location of reference surface normal to *s*, for the Hughes-Liu beam only.	E10.0
51-60	t_{ref}, location of reference surface normal to *t*, for the Hughes-Liu beam only.	E10.0

Integration Rules for Beams

Define the following cards if NIP is nonzero, i.e., ICST=0.

Cards 2,3,...,NIP+1 (3E10.0)

Columns	Quantity	Format
1-10	*s* coordinate of integration point	E10.0
11-20	*t* coordinate of integration point	E10.0
21-30	Weighting factor, A_{ri}, i.e., the area associated with the integration point divided by actual cross sectional area $A_{ri} = \frac{A_i}{A}$. See Figure 4.2.	E10.0



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

Figure 4.1. Definition of relative area for user defined integration rule.

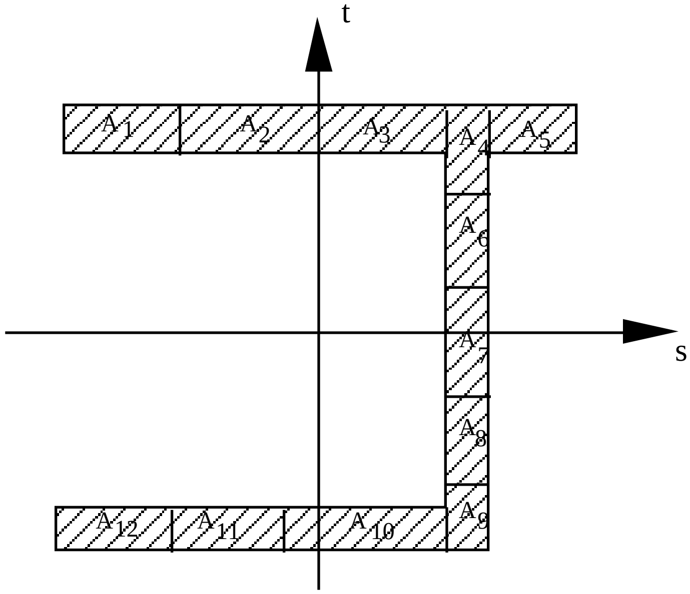


Figure 4.2. Definition of integration points for user defined integration rule.

The input for standard beam section types are defined below. In Figure 4.3 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.

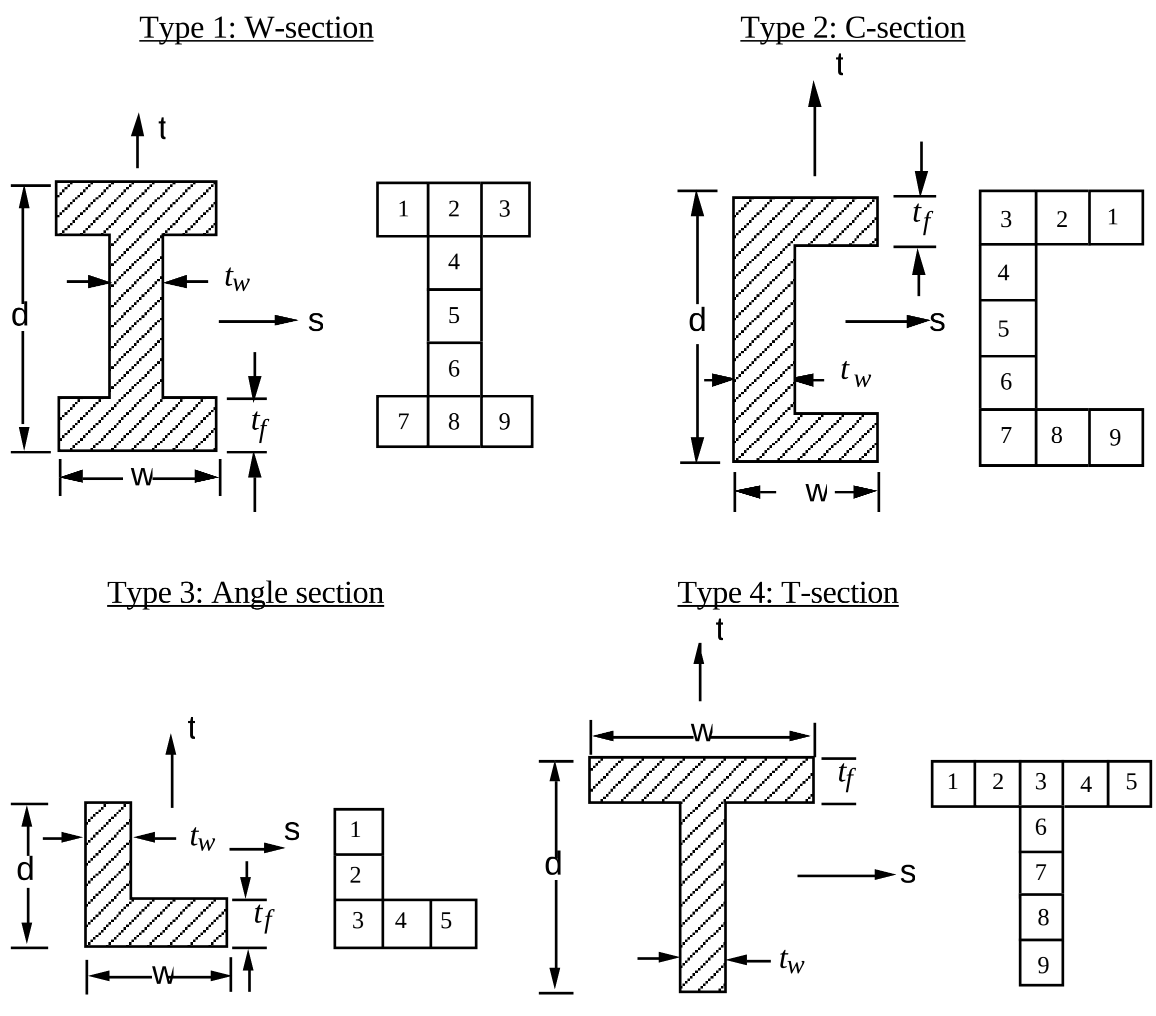
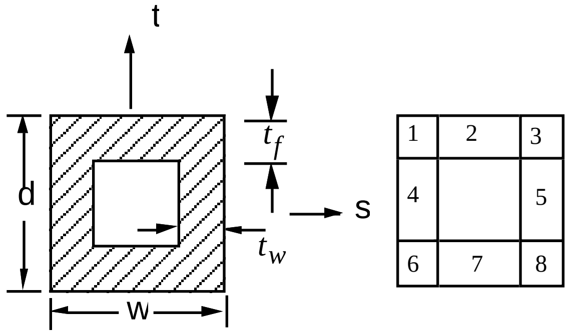
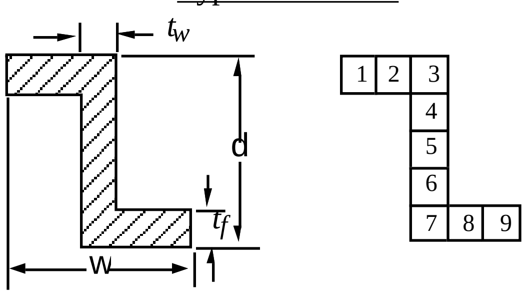


Figure 4.3a. Standard beam cross sections.

Type 5: Rectangular tubing



Type 6: Z-section



Type 7: Trapezoidal section.

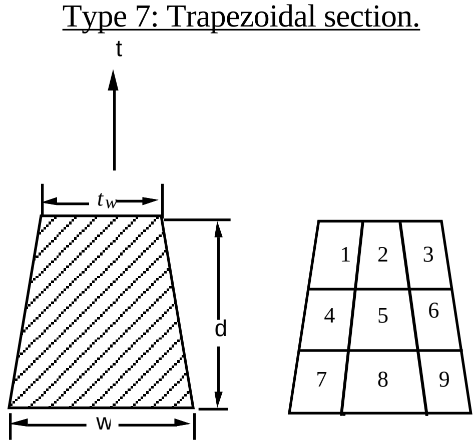


Figure 4.3b. Standard beam cross sections.

5. User Defined Integration Rules for Shells

Define NUSIR (see Control Card 12, columns 46-50) card sets in this section.

Card 1 (2I5)

Columns	Quantity	Format
1-15	Number of integration points, NIP	15
6-10	IESOP, 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	15

Card 2,3,...,NIP+1 (2E10.0,I5) or (2E10.0,I10) for MLARG

Define these cards if and only if IESOP equals 0.

Columns		Quantity	Format	
1-10	(1-10)	Coordinate of integration point in range -1 to 1	E10.0	(E10.0)
11-20	(11-20)	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 piont = $\frac{\Delta t_i}{t}$ as seen in Figure 5.1	E10.0	(E10.0)
21-25	(21-30)	Optional part identification number if different from the number specified on the element card. The material type is not allowed to change. If IARB=1, (Control Card 11, cols 11-15), specify the property set card label corres-ponding to the material to be used. <u>Do not specify the constitutive model ID if IARB=1</u>.	I5	(I10)

Note: The density, hourglass type, element type, element formulation, and section data is always taken from the data for the part ID specified on the element card.

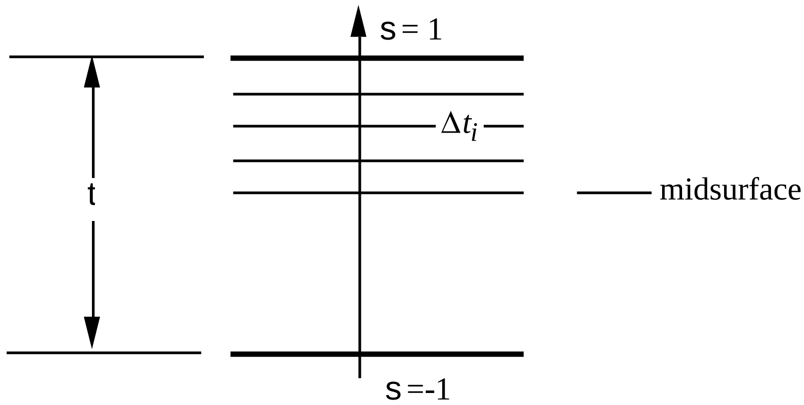


Figure 5.1. In the user defined shell integration rule the ordering of the integration points is arbitrary.

6. Nodal Point Cards

(2I5,3'//NIF//',2I5), or (I8,I5,3'//NIF//',I5) for LARGE option

(NIF is defined on Control Card 11 Cols. 16-20)

Define NUMNP (see Control Card 1, columns 11-20) nodal point cards in this section.

Columns		Quantity	Format	
1-5	(1-8)	Node number	I5	(I8)
6-10	(9-13)	Displacement boundary condition code 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 EQ.8: constrained in direction specified on the first boundary condition card read in Section 15 below EQ.9: constrained in direction specified on the second boundary condition card read in Section 15 below	I5	(I5)

<u>If NIF = 'E20.0', otherwise subtract 10 spaces from the next 3 fields.</u>

11-30	(14-33)	x-coordinate	E20.0	(E20.0)
31-50	(34-53)	y-coordinate	E20.0	(E20.0)
51-70	(54-73)	z-coordinate	E20.0	(E20.0)
71-75	(-)	Nodal increment k (consecutive ordering only)	I5	(omit)
76-80	(74-78)	Rotational boundary condition code 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	I5	(I5)

Nodal Point Cards

If the arbitrary flag is set to 1 on the eighth control card, simply define NUMNP nodal points. The nodal point numbers are labels and no ordering is assumed and no nodes are generated.

On the other hand, if the arbitrary flag is 0 the nodal point numbers must be between 1 and NUMNP inclusive, and the largest nodal point number must terminate the nodal data. Whenever nodal data are missing, node numbers are generated according to the sequence

$$n_i, n_i + k, n_i + 2k, \ldots, n_j$$

where n_i and n_j are the nodal numbers defined on two consecutive cards. Linear interpolation is used to obtain the coordinates of the generated nodes. The boundary condition code is set to zero whenever the boundary condition code of n_i differs from that of n_j. Unconstrained nodes can be generated between constrained nodes that have the same boundary condition by making the code on one of the two cards negative. After the nodal data is generated, the signs of all negative boundary condition codes are reset.

7. Element Cards for Solid Elements

(11I5,24x,A1), or (I8, I5, 8I8,2x,A1) for LARGE option
or (2I10/8I8,15x,A1) for MLARG option

Define NUMELH (see Control Card 1, columns 21-30) element cards in this section. For the default and **LARGE** option the following card is defined.

Columns		Quantity	Format	
1-5	(1-8)	Element number	I5	(I8)
6-10	(9-13)	Material or property set label	I5	(I5)
11-15	(-)	Increment *k*	I5	(omit)
15-20	(14-21)	Nodal point n_1	I5	(I8)
21-25	(22-29)	Nodal point n_2	I5	I8)
26-30	(30-37)	Nodal point n_3	I5	(I8)
.	.	.		.
.	.	.		.
.	.	.		.
51-55	(70-77)	Nodal point n_8	I5	(I8)
80-80	(80-80)	Optional continuation flag for material orientation	A1	(A1)
********		**second card here if C appears in column 80**	********	
1-10	(1-10)	a_1	E10.0	(E10.0)
11-20	(11-20)	a_2	E10.0	(E10.0)
21-30	(21-30)	a_3	E10.0	(E10.0)
31-40	(31-40)	d_1	E10.0	(E10.0)
41-50	(41-50)	d_2	E10.0	(E10.0)
51-60	(51-60)	d_3	E10.0	(E10.0)

For the **MLARG** option define the following 2 cards.

Card 1 (2I10)

Columns	Quantity	Format
1-10	Element ID	I10
11-20	Material or property set label	I10

Card 2 (8I8)

Columns	Quantity	Format
1-8	Nodal point n_1	I8
9-16	Nodal point n_2	I8
17-24	Nodal point n_3	I8
.	. . .	
.	. . .	
.	. . .	
57-64	Nodal point n_8	I8
********	**third card here if C appears in column 80**	********
1-10	a_1	E10.0
11-20	a_2	E10.0
21-30	a_3	E10.0
31-40	d_1	E10.0
41-50	d_2	E10.0
51-60	d_3	E10.0

If the arbitrary flag is set to 1 in column 20 of the eleventh control card, simply define NUMELH elements. The element numbers are unique labels, no ordering is assumed, and no elements are generated.

However, if the arbitrary flag is set to 0, element cards are assumed to be in element number sequence. Omitted data are automatically generated with respect to the first card prior to the omitted data as follows:

$$n_j^{i+1} = n_j^i + k$$

The material properties for the generated elements and the mesh generation parameter *k* are taken from the first card. The default value of *k* is l.

Nodal points n_1 - n_8 define the corner nodes of the 8-node solid elements. Elements having fewer than 8 nodes are obtained by repeating one or more nodes. Four, six, and eight node elements are shown in Figure 7.1. Input of nodes on the element cards for the former two elements would be in the form

<u>4-node</u>	$n_1\ n_2\ n_3\ n_4\ n_4\ n_4\ n_4\ n_4$
<u>6-node</u>	$n_1\ n_2\ n_3\ n_4\ n_5\ n_5\ n_6\ n_6$

Note: In all cases the first four node numbers must be unique.

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 the computed from the two vectors such that (also see Figure 7.2):

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

These vectors are internally normalized within LS-DYNA.

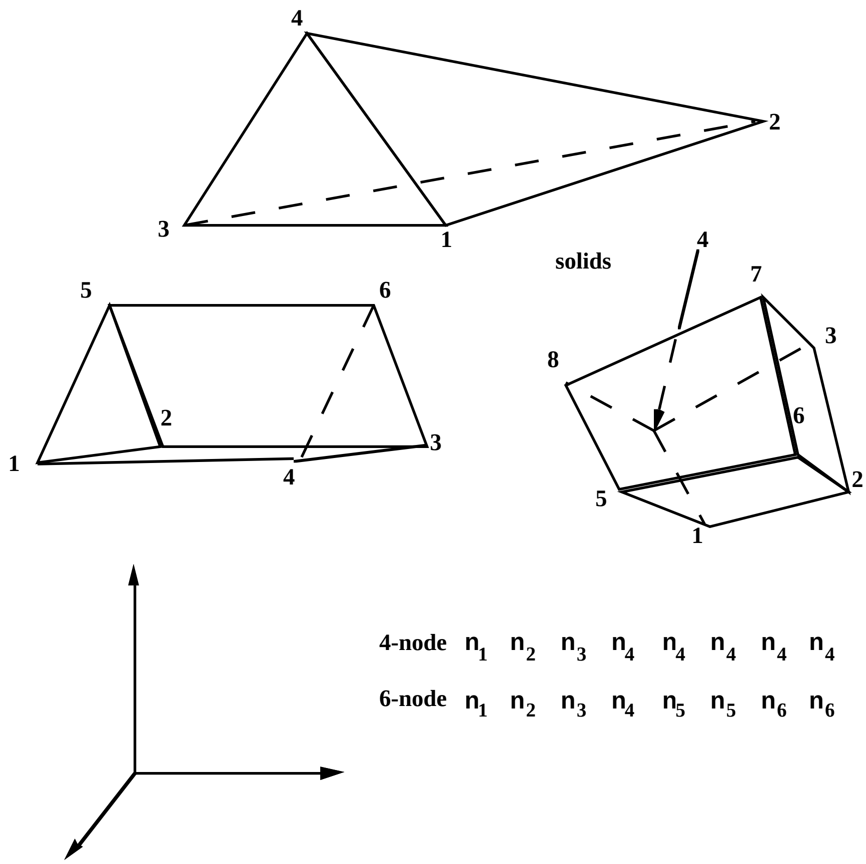


Figure 7.1. Four, six, and eight node solid elements.

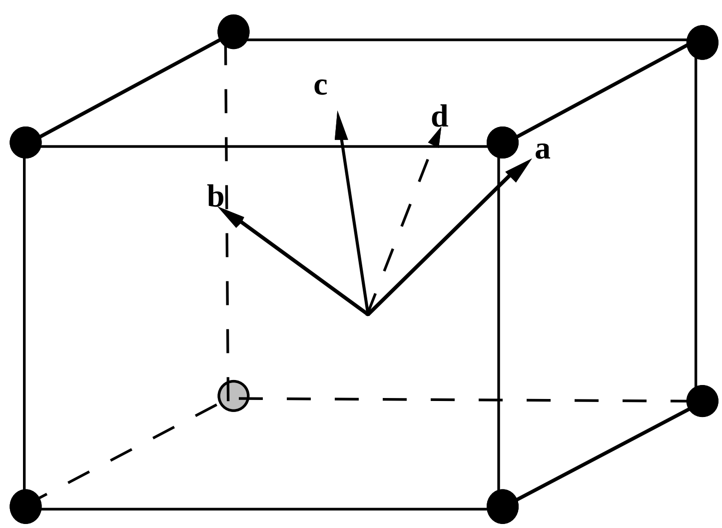


Figure 7.2. Two vectors **a** and **d** are defined and the triad is computed and stored. Vectors **b** and **d** lie in the same plane.

8. Element Cards for Beam Elements
(6I5,5E10.0), or (I8,I5,3I8) for LARGE option
or (5I10/E510.0) for MLARG option

Define NUMELB (see Control Card 1, columns 31-40) element cards in this section.

For the default and **LARGE** option the following card format is defined.

Columns		Quantity	Format	
1-5	(1-8)	Element number	15	(I8)
6-10	(9-13)	Material or property set label	15	(I5)
11-15	(-)	Increment *k* (consecutive ordering only)	I5	(omit)
16-20	(14-21)	Nodal point n_1	I5	(I8)
21-25	(22-29)	Nodal point n_2	I5	(I8)
26-30	(30-37)	Nodal point n_3 (see Figure 3.2) The third node is optional for beam types 3, 6, 7, 8, and 9 if the latter has a circular cross section.	I5	(I8)
31-40	(38-45)	Optional beam parameter 1. 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	E10.0	(E8.0)
41-50	(46-53)	Optional beam parameter 2. 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	E10.0	(E8.0)

Columns	Quantity	Format
51-60 (54-61)	Optional beam parameter 3. 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	E10.0 (E8.0)
61-70 (62-69)	Optional beam parameter 4. 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	E10.0 (E8.0)
71-80 (70-77)	Optional beam parameter 5. 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	E10.0 (E8.0)

Define the second card if a $ is in column **79** of card 1. Note: Optional for LARGE format only.

Card 2 (2I10.0) (Large option only)

Columns	Quantity	Format
1-10	Optional welded part ID (spotweld beam type 9) for node n_1 If not specified, the nearest master segment is used in the type 7 contact definition is used.	I10
11-20	Optional welded part ID (spotweld beam type 9) for node n_2 If not specified, the nearest master segment in the type 7 contact definition is used, excluding the segment to which node n_1 is tied.	I10

For the **MLARG** option the following card format is used (see above for detailed description):

Card 1 (5I10)

Columns	Quantity	Format
1-10	Element number	I10
11-20	Part ID	I10
21-30)	Nodal point n_1	I10
31-40	Nodal point n_2	I10
41-50	Nodal point n_3 (see Figure 3.11)	I10
51-60	Optional welded part ID (spotweld beam type 9) for node n_1 If not specified nearest master segment is used in the type 7 contact definition is used.	I10
61-70	Optional welded part ID (spotweld beam type 9) for node n_2 If not specified, the nearest master segment in the type 7 contact definition is used, excluding the segment to which node n_1 is tied.	I10

Define the second card if a $ is in column 80 of card 1

Card 2 (5E10.0)

Columns	Quantity	Format
1-10	Optional beam parameter 1. Based on beam type.	E16.0
11-20	Optional beam parameter 2. Based on beam type.	E16.0
21-30	Optional beam parameter 3. Based on beam type.	E16.0
31-40	Optional beam parameter 4. Based on beam type.	E16.0
41-50	Optional beam parameter 5. Based on beam type.	E16.0

If the arbitrary flag is set to 1 in column 10 of the eleventh control card, simply define NUMELB elements. The element numbers are unique labels, no ordering is assumed, and no elements are generated. However, if the arbitrary flag is set to 0, element cards are assumed to be in element number sequence. Omitted data are automatically generated with respect to the first card prior to the omitted data as follows:

$$n_j^{i+1} = n_j^i + k \qquad (j = 1,2)$$

The material properties, cross-sectional properties, and orientation node, $n_{3,}$ for the generated elements and the mesh generation parameter k are taken from the card preceding the generated data. The default value of k is l. If the thickness or cross-sectional properties are undefined, they are taken from the material cards.

9. Element Cards for Shell Elements

(7I5,4E10.0,E5.0), (I8,I5 4I8/5E10.0) for LARGE option or (6I10/5E10.0) for MLARG option

Define NUMELS (see Control Card 1, columns 41-50) element cards in this section.

Columns		Quantity	Format	
For the default and **LARGE** option the following card format is defined.				
1-5	(1-8)	Element number	15	(I8)
6-10	(9-13)	Material or property set label	15	(I5)
11-15	(-)	Increment k	I5	(omit)
16-20	(14-21)	Nodal point n_1 (see Figure 3.13)	I5	(I8)
21-25	(22-29)	Nodal point n_2	I5	(I8)
26-30	(30-37)	Nodal point n_3	I5	(I8)
31-35	(38-45)	Nodal point n_4	I5	(I8)
For the **MLARG** option the following card format is defined.				
1-10		Element number	I10	
11-20		Material or property set label	I10	
21-30		Nodal point n_1 (see Figure 3.13)	I10	
31-40		Nodal point n_2	I10	
41-50		Nodal point n_3	I10	
51-60		Nodal point n_4	I10	
****		**second card here if LARGE or MLARGE option is active and there is no dollar sign $ in column 80**	****	
36-45	(1-10)	Shell thickness at node 1 (optional)	E10.0	(E10.0)
46-55	(11-20)	Shell thickness at node 2 (optional)	E10.0	(E10.0)
56-65	(21-30)	Shell thickness at node 3 (optional)	E10.0	(E10.0)
66-75	(31-40)	Shell thickness at node 4 (optional)	E10.0	(E10.0)
76-80	(41-50)	Orthotropic material angle (optional)	E5.00	(E10.0)

If the arbitrary flag is set to 1 in column 10 of the eleventh control card, simply define NUMELS elements. The element numbers are unique labels, no ordering is assumed, and no elements are generated. However, if the arbitrary flag is set to 0, element cards are assumed to be in element number sequence. Omitted data are automatically generated with respect to the first card prior to the omitted data as follows:

$$n_j^{i+1} = n_j^i + k$$

The material and cross-sectional properties for the generated elements and the mesh generation parameter *k* are taken from the card preceding the generated data. The default value of *k* is l. If the thicknesses are undefined, they are taken from the material cards. Triangular elements are defined by repeating the third node, i.e., by setting $n_4 = n_3$.

10. Element Cards for 8-Node Solid Shell
(11I5), or (I8,I5,8I8) for LARGE option
or (2I10/8I8) for MLARG option

Define NUMELT (see Control Card 1, columns 51-60) element cards in this section. For the default and **LARGE** option the following card is defined.

Columns		Quantity	Format	
1-5	(1-8)	Element number	15	(I8)
6-10	(9-13)	Material or property set label	I5	(I5)
11-15	(-)	Increment *k*	I5	(omit)
15-20	(14-21)	Nodal point n_1	I5	(I8)
21-25	(22-29)	Nodal point n_2	I5	I8)
26-30	(30-37)	Nodal point n_3	I5	(I8)
.	.	.	.	
.	.	.	.	
.	.	.	.	
51-55	(70-77)	Nodal point n_8	I5	(I8)

For the **MLARG** option define the following 2 cards.

Card 1 (2I10)

Columns	Quantity	Format
1-10	Element ID	I10
11-20	Material or property set label	I10

Card 2 (8I8)

Columns	Quantity	Format
1-8	Nodal point n_1	I8
9-16	Nodal point n_2	I8
17-24	Nodal point n_3	I8
.	. . .	
.	. . .	
.	. . .	
57-64	Nodal point n_8	I8

If the arbitrary flag is set to 1 in column 10 of the eleventh control card, simply define NUMELT elements. The element numbers are unique labels, no ordering is assumed, and no elements are generated. However, if the arbitrary flag is set to 0, element cards are assumed to be in element number sequence. Omitted data are automatically generated with respect to the first card prior to the omitted data as follows:

$$n_j^{i+1} = n_j^i + k$$

The material properties for the generated elements and the mesh generation parameter k are taken from the first card. The default value of k is l.

Nodes n_1 to n_4 define the lower surface, and nodes n_5 to n_8 define the upper surface. The integration points lie along the t-axis as depicted in Figure 10.1. Extreme care must be used in defining the connectivity to insure proper orientation.

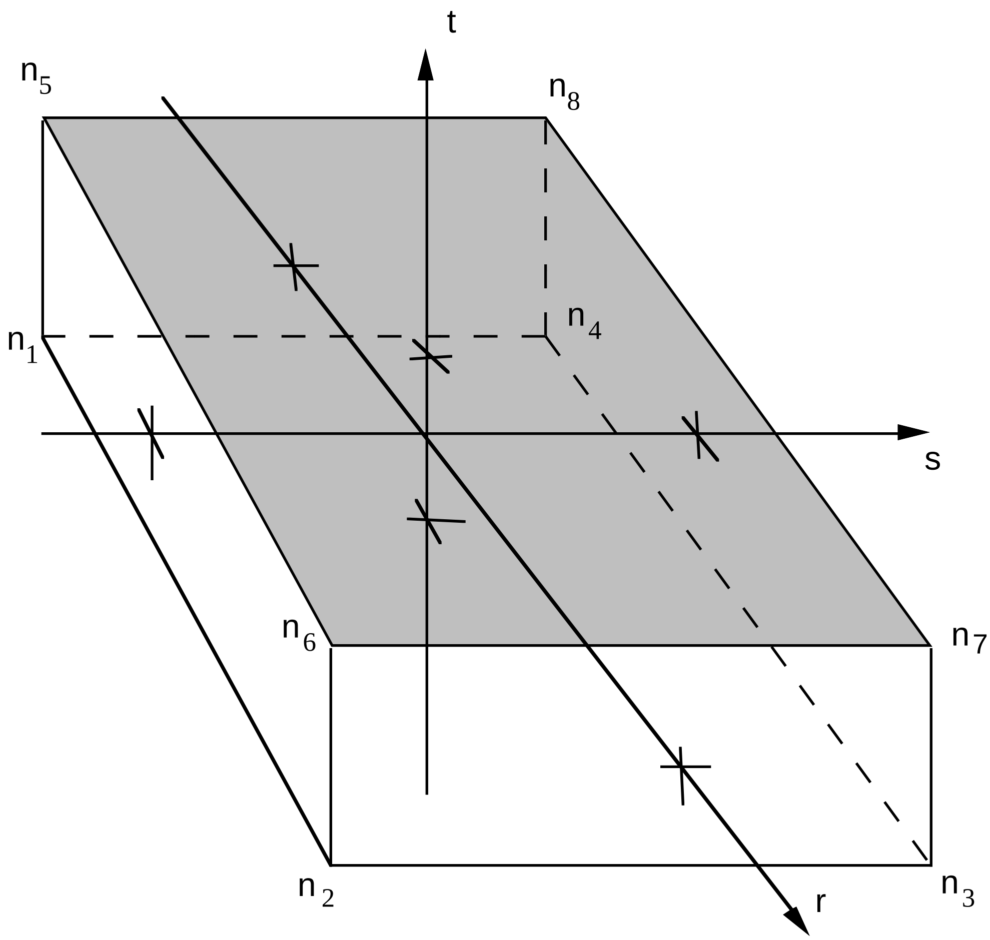


Figure 10.1. Solid 8-node Shell Element.

11. Interface Definitions for Component Analysis

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 master surfaces in the tied interface options. Nodal definitions are used (in the later calculations) to move the edges of shell elements or nodes connected to springs and beams. For the former case, the input described in Section 32, TIE BREAKING SHELL DEFINITIONS, is used, and for the latter, the input described in Section 34, NODES MOVED VIA SECTION 11 INTERFACE FILE, is used. Segment definitions are used in later calculations to move the surfaces of brick elements as described in Section 31, SLIDING INTERFACE DEFINITIONS. It is important to output data at reasonably small interval to accurately resolve the boundary movement in the later calculation.

Define NUMIFS (see Control Card 7, columns 11-15) interface definitions. Each definition consists of a control card followed by a set of cards that define the interface. Interfaces may consists of a set of four node segments for moving interfaces of solid elements, a line of nodes for treating interfaces of shells, or a single node for treating beam and spring elements.

Card 1
(2I5), or (2I8) for LARGE option

Columns		Quantity	Format
1-5	(1-8)	Interface type, ITYPE EQ.1: nodal EQ.2: segments	I5 (I8)
6-10	(9-16)	Number of nodes or segments, NUM	I5 (I8)

If the interface type applies to beams or spring elements then the definition should include just one nodal point.

If ITYPE=1 then define the following cards:

Cards 2,....,NUM+1
(2I5), or (2I8) for LARGE option

Columns		Quantity	Format
1-5	(1-8)	Number ≤ NUM	I5 (18)
6-10	(9-16)	Nodal point number	I5 (18)

Omitted nodes are automatically generated by incrementing the nodal point numbers by

$$\frac{n_i - n_j}{sn_i - sn_j}$$

where sn_i and sn_j are numbers on two successive cards and n_i and n_j are their corresponding node numbers.

If ITYPE=2 then define the following interface segment cards:

Cards 2,....,NUM+1
(6I5), or (5I8) for LARGE option

Columns		Quantity	Format
1-5	(1-8)	Interface segment number	I5 (I8)
6-10	(-)	Increment, k	I5 (-)
11-15	(9-16)	Nodal point n_1	I5 (I8)
16-20	(17-24)	Nodal point n_2	I5 (I8)
21-25	(25-32)	Nodal point n_3	I5 (I8)
26-30	(33-40)	Nodal point n_4	I5 (I8)

Omitted data are automatically generated with respect to the first card prior to the omitted data as

$$n_j^{i+1} = n_j^i + k$$

The generation parameter k is taken from the first card. Nodal numbering can be either clockwise or counterclockwise. Nodal points n_1 - n_4 define the corner nodes of the segments as shown in Figure 23.2. Triangular segments are defined by repeating a node.

12. DYNA3D/JOY Interface Definition
(2I5), or (2I8) for LARGE options

Define the number of nodes, NUMSNC, specified on Control Card 2, columns 31-35.

Columns		Quantity	Format
1-5	(1-8)	Interface node number	I5 (I8)
6-10	(9-16)	Nodal point number	I5 (I8)

Omitted interface nodes are automatically generated by incrementing the nodal point numbers by

$$\frac{n_i - n_j}{in_i - in_j}$$

where in_i and in_j are interface node numbers on two successive cards and n_i and n_j are their corresponding node numbers.

13. Nodal Single Point Constraints
(2I5,4X,6I1,2I5), or (I8,I5,4X,6I1,2I8) for LARGE option

Define NODSPC (see Control Card 2, columns 1-5) cards.

Columns		Quantity	Format
1-5	(1-8)	First nodal point number, IFIRST	I5 (I8)
6-10	(9-13)	Local coordinate system number < NSPCOR+1	I5 (I5)
11-14	(14-17)	Blank	
15	(18)	Insert 1 (0) for (no) translational constraint in local x-direction	I1 (I1)
16	(19)	Insert 1 (0) for (no) translational constraint in local y-direction	I1 (I1)
17	(20)	Insert 1 (0) for (no) translational constraint in local z-direction	I1 (I1)
18	(21)	Insert 1 (0) for (no) rotational constraint about local x-axis	I1 (I1)
19	(22)	Insert 1 (0) for (no) rotational constraint about local y-axis	I1 (I1)
20	(23)	Insert 1 (0) for (no) rotational constraint about local z-axis	I1 (I1)
21-25	(24-31)	Last nodal point number, ILAST EQ.O: ILAST = IFIRST	I5 (I8)
26-30	(32-39)	Increment for generation EQ.O: default set to 1	I5 (I8)

14. Local Coordinate Systems

Define NSPCOR (see Control Card 2, columns 6-10) local coordinate systems for single point constraints, nodal forces, etc. The xy plane is described by two vectors: the local x axis and another vector lying in the plane. The local z axis is the cross product of these two vectors. The local y axis is found by taking the cross product of the local z and x axes.

(I1,I4,6E10.0)

Columns	Quantity	Format
1-1	Definition type, ITYPE EQ.0: origin lies at zero and two points are defined EQ.1: three nodal points are defined EQ.2: origin is defined followed by two points	I1
2-5	Local coordinate system number < NSPCOR+1	I4

If ITYPE=0 then in columns 6-65 define:

Columns	Quantity	Format
6-15	x-coordinate of local x-axis. Origin lies at (0,0,0)	E10.0
16-25	y-coordinate of local x-axis	E10.0
26-35	z-coordinate of local x-axis	E10.0
36-45	x-coordinate of local inplane vector	E10.0
46-55	y-coordinate of local inplane vector	E10.0
56-65	z-coordinate of local inplane vector	E10.0

If ITYPE=1 then in columns 6-35 define:

Columns	Quantity	Format
6-15	node, n_1	E10.0
16-25	node, n_2	E10.0
26-35	node, n_3	E10.0

Columns	Quantity	Format
If ITYPE=2 then in columns 6-65 define:		
6-15	x-coordinate of origin	E10.0
16-25	y-coordinate of origin	E10.0
26-35	z-coordinate of origin	E10.0
36-45	x-coordinate of local x-vector	E10.0
46-55	y-coordinate of local x-vector	E10.0
56-65	z-coordinate of local x-vector	E10.0

Optional Card 2 (define if and only if ITYPE=2 above)

(3E10.0)

Columns	Quantity	Format
1-10	x-coordinate of local in plane vector	E10.0
11-20	y-coordinate of local in plane vector	E10.0
21-30	z-coordinate of local in plane vector	E10.0

If three nodes are used the local coordinate system is defined 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 $\mathbf{y} = \mathbf{x} \times \mathbf{z}$

The vectors defining the local coordinate system are internally normalized so there is no need to define unit vectors.

15. Vehicle Kinematics Intialization Cards

Define the number of vehicles, NVHINI, specified on Control Card 17, columns 71-75. Each vehicle is initialized with kinematical information provided below. 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. This feature is not restricted to vehicle initialization and may be used to reorient and apply initial conditions to a general body.

Card 1
(I5), or (I5) for LARGE option

Columns		Quantity	Format
1-5	(1-5)	gravity direction code, IGRAV 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	I5 (I5)

Define NVHINI card sets below, one for each vehicle to be initialized.

Card 2
(I5), or (I5) for LARGE option

Columns		Quantity	Format
1-5	(1-5)	NPID, number of part ID's comprising the vehicle.	I5 (I5)

Card 3,...,NPID+2
(I5), or (I8) for LARGE option

Columns		Quantity	Format
1-80	(1-80)	Part IDs, up to sixteen (ten) per card	I5 (I8)

Card NPID+3
(I5,6E10.0), or (I5,6E10.0) for LARGE option

Columns		Quantity	Format	
1-10	(1-10)	x-coordinate of initial position of mass center	E10.0	(E10.0)
11-20	(10-20)	y-coordinate of initial position of mass center	E10.0	(E10.0)
21-30	(20-30)	z-coordinate of initial position of mass center	E10.0	(E10.0)
31-40	(30-40)	x-coordinate of final position of mass center	E10.0	(E10.0)
41-50	(40-50)	y-coordinate of final position of mass center	E10.0	(E10.0)
51-60	(50-60)	z-coordinate of final position of mass center	E10.0	(E10.0)

Card NPID+4
(3E10.0,3I5), or (3E10.0,3I5) for LARGE option

Columns		Quantity	Format	
1-10	(1-10)	x-component of mass center velocity	E10.0	(E10.0)
11-20	(11-20)	y-component of mass center velocity	E10.0	(E10.0)
21-30	(21-30)	z-component of mass center velocity	E10.0	(E10.0)
31-35	(31-35)	first rotation axis code, AAXIS EQ.1: initiallly aligned with global x-axis EQ.2: initiallly aligned with global y-axis EQ.3: initiallly aligned with global z-axis	I5	(I5)
36-40	(36-40)	second rotation axis code, BAXIS	I5	(I5)
41-45	(41-45)	third rotation axis code, CAXIS	I5	(I5)

Card NPID+5
(6E10.0), or (6E10.0) for LARGE option

Columns		Quantity	Format	
1-10	(1-10)	rotation angle about the 1st axis, AANG (degrees)	E10.0	(E10.0)
11-20	(11-20)	rotation angle about the 2nd axis, BANG (degrees)	E10.0	(E10.0)
21-30	(21-30)	rotation angle about the 3rd axis, CANG (degrees)	E10.0	(E10.0)
31-40	(31-40)	angular velocity component for 1st axis, WA (rad/s)	E10.0	(E10.0)
41-50	(41-50)	angular velocity component for 2nd axis, WB (rad/s)	E10.0	(E10.0)
51-60	(51-60)	angular velocity component for 3rd axis, WC (rad/s)	E10.0	(E10.0)

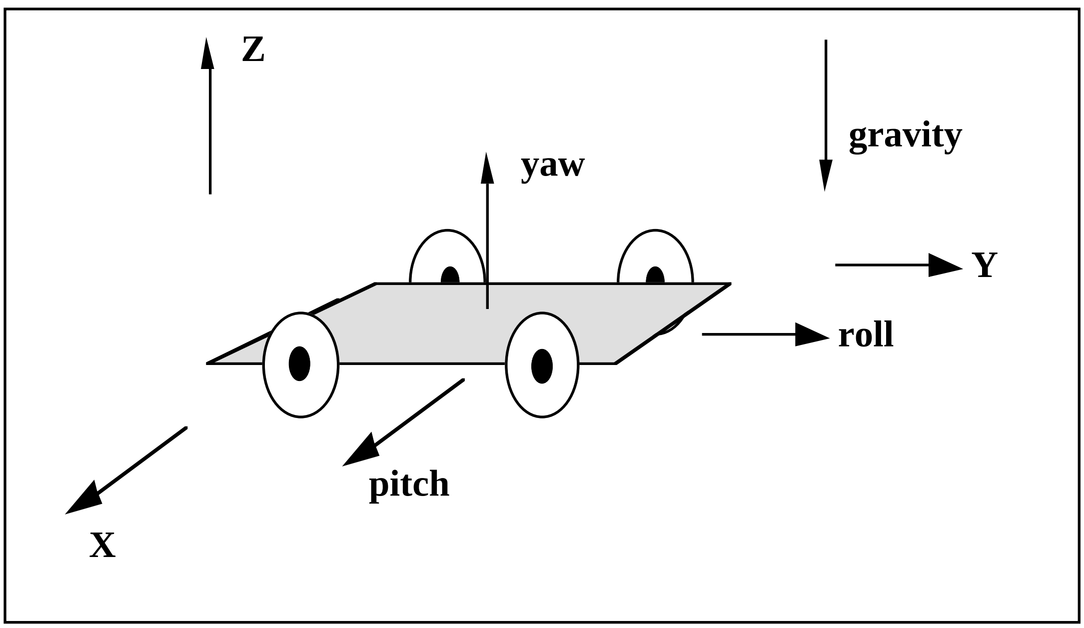


Figure 15.1. The vehicle pictured is to be oriented with a successive rotation sequence about the yaw, pitch, and roll axes, respectively. Accordingly, AAXIS=3, BAXIS=1 and CAXIS=2. The direction of gravity is given by IGRAV=-3.

16. Sliding Boundary Plane Cards
(4E10.0)

Define the number of cards, NUMRC, specified on Control Card 2, columns 21-25.

Columns	Quantity	Format
1-10	x-coordinate of vector	E10.0
11-20	y-coordinate of vector	E10.0
21-30	z-coordinate of vector	E10.0
31-40	Constraint option EQ.0.0: node is constrained to move on normal plane EQ.1.0: node is constrained to translate in vector direction	E10.0

Any node may be constrained to move on an arbitrarily oriented plane or line. 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.

17. Symmetry Planes with Failure

Define the number of cards, NUMRCF, specified on Control Card 2, columns 26-30.

Card 1 (I5,7E10.0)

Columns	Quantity	Format
1-5	Number of segments in symmetry plane, NSGNDS	I5
6-15	x-coordinate of tail of a normal vector originating on the wall (tail) and terminating in the body (head) (i.e., vector points from symmetry plane into body)	E10.0
16-25	y-coordinate of tail	E10.0
26-35	z-coordinate of tail	E10.0
36-45	x-coordinate of head	E10.0
46-55	y-coordinate of head	E10.0
56-65	z-coordinate of head	E10.0
66-75	Tensile failure stress (> 0)	E10.0

Card 2,3,...,NSGNDS+1 (5I5,E10.0), or (5I8,E10.0) for LARGE option

Columns		Quantity	Format	
1-5	(1-8)	Segment number	I5	(I8)
6-10	(9-16)	Nodal point n_1	I5	(I8)
11-15	(17-24)	Nodal point n_2	I5	(I8)
16-20	(25-32)	Nodal point n_2	I5	(I8)
21-25	(33-40)	Nodal point n_4	I5	(I8)
26-35	(41-50)	Failure stress if different from default value	E10.0	(E10.0)

Triangular segments are defined by repeating the last node.

18. Nodal Time History Blocks
(16I5), or (10I8) for LARGE option

Skip this section if the number of nodal time history blocks is zero. Otherwise, define up to 2000 history blocks that may contain a total of 2000 nodes. Use only the number of cards required to define NDTH (see Control Card 20, columns 31-35) blocks.

Columns		Quantity	Format
1-5	(1-8)	First node of first time history block	I5 (I8)
6-10	(9-16)	Last node of first time history block	I5 (I8)
11-15	(17-24)	First node of second time history block	I5 (I8)
16-20	(25-32)	Last node of second time history block	I5 (I8)
.	.	.	.
.	.	.	.
.	.	.	.

19. Element Time History Blocks

(16I5), or (10I8) for LARGE option

(Solid Elements)

Skip this section if the number of solid element time history block is zero. Otherwise, define up to 2000 time history blocks that may contain a total of 2000 elements. Use only the number of cards required to define NSTH (see Control Card 20, columns 36-40) blocks.

Columns		Quantity	Format
1-5	(1-8)	First element of first time history block	I5 (I8)
6-10	(9-16)	Last element of first time history block	I5 (I8)
11-15	(17-24)	First element of second time history block	I5 (I8)
16-20	(25-32)	Last element of second time history block	I5 (I8)
.	.	.	.
.	.	.	.
.	.	.	.

(Beam Elements)

Skip these cards if the number of beam element time history blocks, NSTB (see Control Card 20, columns 41-45), is zero. Up to 2000 time history blocks may be defined, containing a total of 2000 elements

(Shell Elements)

Skip these cards if the number of shell element time history blocks, NSTS (see Control Card 20, columns 46-50) is zero. Up to 2000 time history blocks may be defined, containing a total of 2000 elements

(Solid Shell Elements)

Skip these cards if the number of solid shell element time history blocks, NSTT (see Control Card 20, columns 51-55) is zero. Up to 2000 time history blocks may be defined, containing a total of 2000 elements.

20. Density versus Depth Curve for Gravity Loading

Skip this section if the number of points in the density versus depth curve is zero; otherwise, supply NUMDP+1 (see Control Card 3, columns 36-40) cards.

Card 1 (E10.0,4x,I1,I3I5) or (E10.0,I1,I5 / (8i10)) for MLARG

For the standard format define the following card.

Columns	Quantity	Format
1-10	Gravitational acceleration	E10.0
15	Direction of loading EQ.1: global x EQ.2: global y EQ.3: global z	I1
16-20	Materials to be initialized EQ.0: all EQ.n: define list of n materials below (n < 13)	I5
21-25	Material number of first material to be initialized	I5
26-30	Material number of second material to be initialized	I5
.	.	.
.	.	.
.	.	.

Cards 2,3,...,NUMDP+1 (2E10.0)

Columns	Quantity	Format
1-10	Mass density	E10.0
11-20	Depth	E10.0

Density versus Depth Curve

For the **MLARG** format define the following cards.

Columns	Quantity	Format
1-10	Gravitational acceleration	E10.0
15	Direction of loading EQ.1: global x EQ.2: global y EQ.3: global z	I1
16-20	Materials to be initialized (nmat) EQ.0: all EQ.n: define list of n materials below (n < 13)	I5

Define nmat materials below if nmat is greater than zero. Define eight materials per card.

Cards 2 ...

Columns	Quantity	Format
1-10	first material to be initialized	E10.0
11-20	second material to be initialized	E10.0

.

Define NUMDP cards.

Cards

Columns	Quantity	Format
1-10	Mass density	E10.0
11-20	Depth	E10.0

21. Brode Function Data

Skip this section if column 35 of Control Card 11 is blank; otherwise, enter two cards for the pertinent Brode function [Stout et al. 1985] data.

Card 1 (8E10.0)

Columns	Quantity	Format
1-10	Yield (Kt)	E10.0
11-20	Height of burst	E10.0
21-30	XBO	E10.0
31-40	YBO	E10.0
	DYNA coordinates of Brode origin (space, time)	
41-50	ZBO	E10.0
51-60	TBO	E10.0
61-65	*Load curve number giving time of arrival versus range relative to Brode origin (space, time)	I5
66-70	Load curve giving yield scaling versus scaled time (time relative to Brode origin divided by [yield$^{(**1/3)}$])	I5

Card 2 (3E10.0)

Columns	Quantity	Format
1-10	Conversion factor - kft to DYNA length units	E10.0
11-20	Conversion factor - milliseconds to DYNA time units	E10.0
21-30	Conversion factor - psi to DYNA pressure units	E10.0

*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.

22. Cross Section Definition for Force Output

This option provides for the ability to determine resultant forces, moments and section properties for various slices across a model. The slice must follow mesh lines and requires the identification of all of the nodes for the section. To determine which side of the section is to be included requires all of the elements on one side to be identified. For resultant moments, the areas of the faces of the elements are integrated to determine section properties. Bending moments are about the center of area of the sections.

For each cross section, NUMCSD (see Control Card 7, columns 1-5), define the following control cards.

Control Card Sets 1,2,...,NUMCSD

Control Card 1 (8I5) or (6I5,I10,I5) for MLARG

Columns		Quantity	Format
1-5	(1-5)	Number of nodes, NCSNOD EQ. 0: automatic generation of cut plane GT. 0: manual generation	I5 (I5)

Define the following input if manual generation is active, i.e., NCSNOD>0

Columns		Quantity	Format
6-10	(6-10)	Number of beam elements	I5 (I5)
11-15	(11-15)	Number of shell elements	I5 (I5)
16-20	(16-20)	Number of thick shell elements	I5 (I5)
21-25	(21-25)	Number of brick elements	I5 (I5)
26-30	(26-30)	Number of spring/damper elements	I5 (I5)

Define the following input if the data is to be output into the SECFORC file in local system

Columns		Quantity	Format
31-35	(31-40)	Rigid body or accelerometer ID. The force resultants are output in the local system of the rigid body or accelerometer.	I5 (I10)
36-40	(41-45)	Flag for local system type EQ. 0: rigid body EQ. 1: accelerometer	I5 (I5)

DEFINE THE OPTIONAL INPUT BELOW IF AND ONLY IF NCSNOD=0

Optional Control Card 2 (6E10.0)

Columns	Quantity	Format
1-10	x-coordinate of tail of any outward drawn normal vector, **N**, originating on wall (tail) and terminating in space (head)	E10.0
11-20	y-coordinate of tail of normal vector **N**	E10.0
21-30	z-coordinate of tail of normal vector **N**	E10.0
31-40	x-coordinate of head of normal vector **N**	E10.0
41-50	y-coordinate of head of normal vector **N**	E10.0
51-60	z-coordinate of head of normal vector **N**	E10.0

Optional Control Card 3 (5E10.0,I5)

Columns	Quantity	Format
1-10	x-coordinate of head of edge vector **L**	E10.0
11-20	y-coordinate of head of edge vector **L**	E10.0
21-30	z-coordinate of head of edge vector **L**	E10.0
31-40	Length of **L** edge, **a** EQ.0.0: extends from negative to positive infinity	E10.0
41-50	Length of **M** edge, **b** EQ.0.0: extends from negative to positive infinity	E10.0
51-55	Number of materials in cutting plane, NMCP	I5

Define NMCP material ID's below. Skip if NMCP=0.

Optional Control Cards 4,..., (8I10)

Columns	Quantity	Format
1-10	First material in cutting plane	I10
11-20	Second material in cutting plane	I10
.	.	.
.	.	.
.	.	.

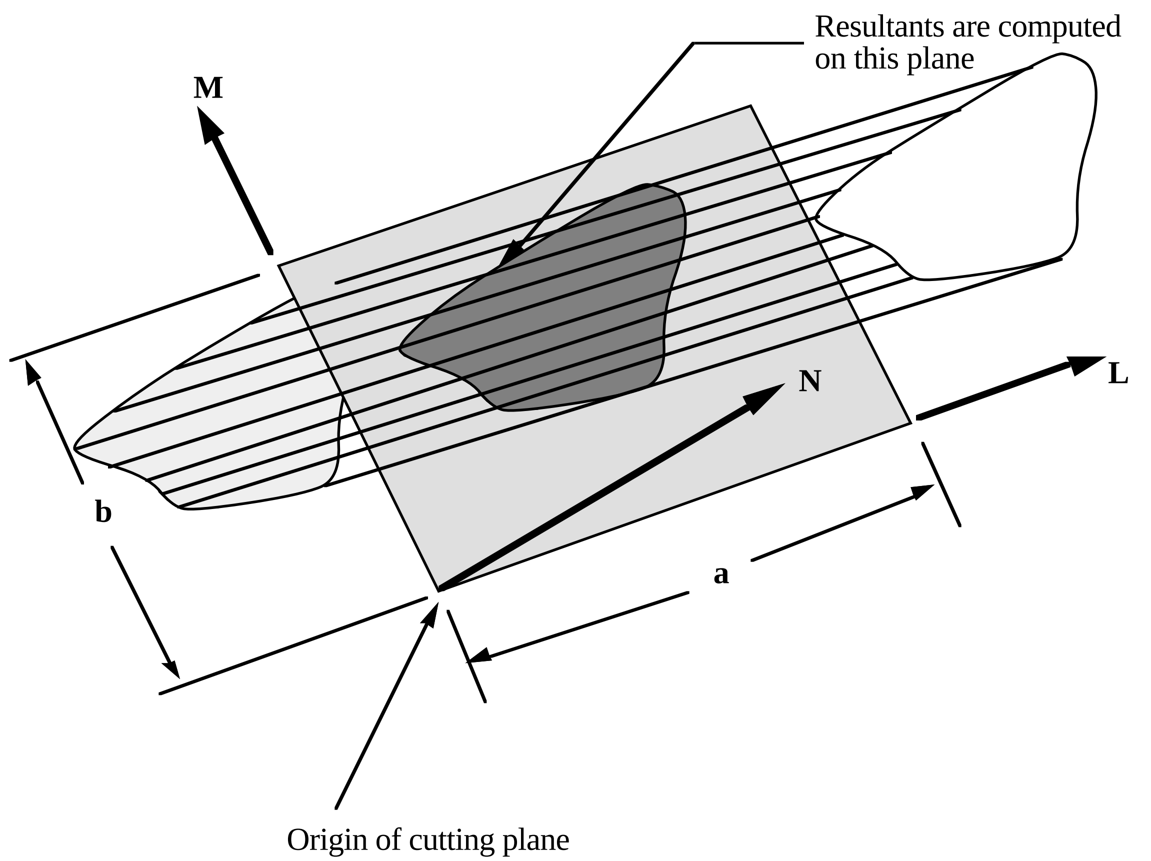


Figure 22.1. Definition of cutting plane for automatic definition of interface for cross-sectional forces. The automatic definiton does not check for springs and dampers in the section. For best results the cutting plane should cleanly pass through the middle of the elements, distributing them equally on either side.

Cross Section Definition

Define one set of cards for each manually defined interface with the information requested below. All of the nodes and elements for a cross section must be input before beginning the next cross section's nodes and elements.

Card NUMCSD+1,...(8I10)

Columns	Quantity	Format
1-10	First cross section node	I10
11-20	Second cross section node	I10
.	.	.
.	.	.
.	.	.
71-80	Eighth cross section node	I10

Use as many cards as needed to define the cross section nodes.

Columns	Quantity	Format
1-10	First beam element	I10
11-20	Second beam element	I10
.	.	.
.	.	.
.	.	.
71-80	Eighth beam element	I10

Use as many cards as needed to define the beam elements in the cross-section.

Columns	Quantity	Format
1-10	First shell element	I10
11-20	Second shell element	I10
.	.	.
.	.	.
.	.	.
71-80	Eighth shell element	I10

Use as many cards as needed to define the shell elements in the cross-section.

Columns	Quantity	Format
1-10	First thick shell element	I10
11-20	Second thick shell element	I10
.	.	.
.	.	.
.	.	.
71-80	Eighth thick shell element	I10

Use as many cards as needed to define the thick shell elements in the cross-section.

Columns	Quantity	Format
1-10	First brick element	I10
11-20	Second brick element	I10
.	.	.
.	.	.
.	.	.
71-80	Eighth brick element	I10

Use as many cards as needed to define the brick elements in the cross-section.

Columns	Quantity	Format
1-10	First spring/damper element	I10
11-20	Second spring/damper element	I10
.	.	.
.	.	.
.	.	.
71-80	Eighth spring/damper element	I10

Use as many cards as needed to define the spring/damper elements in the cross-section.

23. Load Curve/Table Definition Cards

Define the number of load curve/table sets, NLCUR, specified on Control Card 3, Columns 1-5. Repeat the following cards for each set:.

Card 1 (3I5,A10,4E10.0,I5)

Columns	Quantity	Format
1-5	Load curve/table ID, numbered sequentially from 1 to NLCUR Arbitrary numbering is possible if this section is moved to the beginning of Section 3. This latter option is flagged by specifying NLCUR as a negative number.	I5
6-10	Number of points in curve, NPTS, for load cruve definition Number of load curves in table, NLDC, for table definition	I5
11-15	Stress initialization by dynamic relaxation EQ.0: load curve used in transient analysis only EQ.1: load curve used in stress initialization but not analysis EQ.2: load curve applies to both initialization and analysis	I5
16-25	Input format for load curve/table. Default= "2E10.0"	A10
26-35	Scale factor for abcissa values, SFA.	E10.0
36-45	Scale factor for ordinate values, SFO.	E10.0
46-55	Offset for time/abcissa values, OFFA, See note below. (load curve definition only). This offset is applied to the abcissa value before applying the scale factor.	E10.0
56-65	Offset for function values, OFFO. See notes below. This offset is applied to the ordinate value before applying the scale factor.	E10.0
66-70	Table option, ITABLE EQ.0: load curve definition EQ.1: table definition EQ.2: general x-y data. Usually 0, set to 2 <u>only</u> 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, ITABLE=0 for time dependent curves, force versus displacement curves, and stress strain curves.	I5

Load Curve/Table Cards

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.

LOAD CURVES

Card 2,...,NPTS+1 (2E10.0 or specified above in Columns 16-25) for Load Curve definitions.

The load curve values input below are scaled after the offsets are applied:

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

Positive offsets for the load curves (ITABLE=0) are intended for time versus function curves since two additional points are generated automatically at time zero and at time .999*OFFO with the function values set to zero. If ITABLE>0, then the offsets do not create these additional points. Negative offsets for the abcissa simply shifts the abcissa values without creating additional points.

Columns	Quantity	Format
1-10	Abcissa value: time, or plastic strain, volumetric strain, etc.	E10.0
11-20	Ordinate valure: load or function value	E10.0

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.

<u>TABLES</u>

Card 2,...,NPTS+1 (2E10.0 or specified above in Columns 16-25) for Table definitions.

Tables are defined by a curve definition that gives for a selected value of an independent variable a curve ID for which the independent variable is constant. For example, a stress versus strain curve ID may be specified for each value of strain rate or temperature. <u>*The independent variable defined below must increase monotonically and the load curve ID's must be consecutive*</u>. There are no requirements for the curves that are used in the table, i.e., the number of points in the curve may differ as may the spaceing along the abcissa.

Columns	Quantity	Format
1-10	Independent variable, for example, strain rate or temperature	E10.0
11-20	Load curve ID, for example, stress versus strain.	E10.0

24. Concentrated Nodal/Rigid Body Forces
(3I5, E10.0,3I5), (I8,2I5,E10.0,3I8) for LARGE option or (I10,2I5,E10.0,5I8) if MLARG

Define the number of concentrated nodal/rigid body point loads, NUMCL, specified on Control Card 3, columns 6-10.

For the standard or **LARGE** format input define the following card

Columns		Quantity	Format
1-5	(1-8)	Nodal point number or rigid body to which this load applies GT.0: nodal point LT.0: absolute value is material or property set number of the rigid body	I5 (I8)
6-10	(9-13)	Direction in which this load acts, IDR. In two-dimensions the force is applied in the xy plane. Note that in axisymmetry, the radial axis coincides with x and the axial with y. EQ.1: x-direction EQ.2: y-direction EQ.3: z-direction EQ.4: follower force 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	I5 (I5)
11-15	(14-18)	Load curve number GT.0: force as a function of time, LT.0: force as a function of the absolute value of the rigid body displacement.	I5 (I5)
16-25	(19-28)	Scale Factor (default =1.0)	E10.0 (E10.0)
26-30	(29-36)	Nodal point m_1 (see comment below)	I5 (I8)
31-35	(37-44)	Nodal point m_2	I5 (I8)
36-40	(45-52)	Nodal point m_3	I5 (I8)
41-45	(53-60)	Local coordinate system number defined in Section 14 EQ.0: global is assumed	I5 (I8)
46-50	(61-68)	Flag to scale load by nodal mass EQ.1: scale by nodal mass	I5 (I8)

Concentrated Nodal Loads

In two-dimensional plane strain and axisymmetric problems the force per unit length must be specified. Radial weighting in axisymmetry is handled internally.

Nodes m_1, m_2, m_3, must be defined if IDR=4. A positive follower force acts normal to the plane defined by these nodes, and a positive follower moment puts a counterclockwise torque about the normal vector. These actions are depicted in Figure 24.1

For the **MLARG** format input define the following cards

Columns	Quantity	Format
1-10	Nodal point number or rigid body to which this load applies GT.0: nodal point LT.0: absolute value is material or property set number of the rigid body	I10
11-15	Direction in which this load acts, IDR. In two-dimensions the force is applied in the xy plane. Note that in axisymmetry, the radial axis coincides with x and the axial with y. EQ.1: x-direction EQ.2: y-direction EQ.3: z-direction EQ.4: follower force 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	I5
16-20	Load curve number	I5
21-30	Scale Factor (default =1.0)	E10.0
31-38	Nodal point m_1 (see comment below)	I8
39-46	Nodal point m_2	I8
47-54	Nodal point m_3	I8
55-62	Local coordinate system number defined in Section 14 EQ.0: global is assumed	I8
63-70	Flag to scale load by nodal mass EQ.1: scale by nodal mass	I8

See notes above.

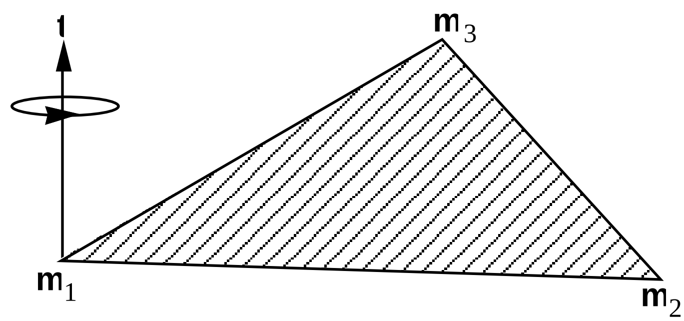


Figure 24.1. Follower force and moment acting on a plane defined by nodes m_1, m_2, and m_3. In this case, the load is applied to node m_1; i.e., m=m_1. A positive force acts in the positive $t$direction, and a positive moment puts a counterclockwise torque about the normal vector.

25. Pressure Boundary Condition Cards

Define the number of card sets, NUMPC, specified on Control Card 3, columns 11-15. This data is followed by the number of pressure load sets, NUMPRM, with masks (see Control Card 3, Columns 51-55). Define NUMPC card sets here.

Card 1 (I5,A1,I4,4I5,5E10.0), or (A1,I4,4I8,4E10.0) for LARGE option

Columns		Quantity	Format	
1-5	(-)	Pressure card number	I5	(omit)
6-6	(1-1)	Continuation flag, *CF*, any alphanumeric character will cause the second card to be read.	A1	(A1)
7-10	(2-5)	Load curve number, LCN EQ.0: Brode function is used to determine pressure LT.0: Applies to 2D only. |LCN| is the load curve ID and the a shear stress is applied as shown in Figure 25.1.	I5	(I5)
11-15	(6-13)	Nodal point n_1 (see Figure 25.1)	I5	(I8)
16-20	(14-21)	Nodal point n_2	I5	(I8)
21-25	(22-29)	Nodal point n_3 (ignored for 2D problems)	I5	(I8)
26-30	(30-37)	Nodal point n_4 (ignored for 2D problems)	I5	(I8)
31-40	(38-47)	Multiplier of load curve at node n_1 EQ.0.0: default set to 1.0	E10.0	(E10.0)
41-50	(48-57)	Multiplier of load curve at node n_2 EQ.0.0: default set to 1.0	E10.0	(E10.0)
51-60	(58-67)	Multiplier of load curve at node n_3 EQ.0.0: default set to 1.0	E10.0	(E10.0)
61-70	(68-77)	Multiplier of load curve at node n_4 EQ.0.0: default set to 1.0	E10.0	(E10.0)
71-80	(-)	TIME, time pressure begins acting on surface.	E10.0	(omit)

Optional Card 2 (E10.0,3I5,E10.0), if CF is defined above.

Columns	Quantity	Format
1-10	$c\rho$, product of material sound speed and density.	E10.0
11-15	Load curve ID, of x-rigid body velocity	I5
16-20	Load curve ID, of y-rigid body velocity	I5
21-25	Load curve ID, of z-rigid body velocity	I5
26-35	TIME, time pressure begins acting on surface, (LARGE format)	E10.0

The load curve multipliers may be used to increase or decrease the pressure. The time value is not scaled. Triangular segments are defined by repeating node n_3.

Card 2 of the input provides a special purpose option for handling Structure Media Interaction problems, SMI. This option requires an initial calculation with the structure modelled as a rigid body to determine the free-field stress distribution, σ_{ff}, and the rigid body motion, υ_{rb}. For the special case addressed here it is assumed that the rotational motion of the structure is negligible relative to the translational components.

In the second calculation the pressure distribution from the initial calculation is used, and the structure is modelled as a deformable body without the media. Each surface segment of the struction has the free-field pressure applied, *p(t)*, which is defined for the ith segment as

$$p(t) = \sigma_{ff} + \rho c(\upsilon_i - \upsilon_{rb})n_i$$

where

σ_{ff} = given pressure history distribution from rigid body calculation

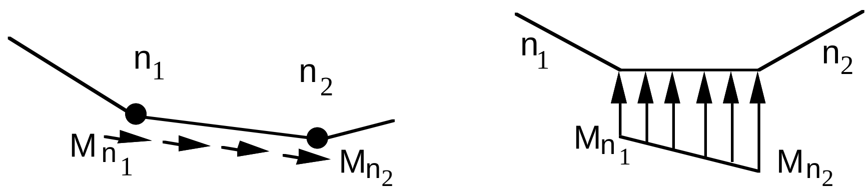
ρc = two constants (density and wave speed)

υ_{rb} = velocity history from the rigid body structure

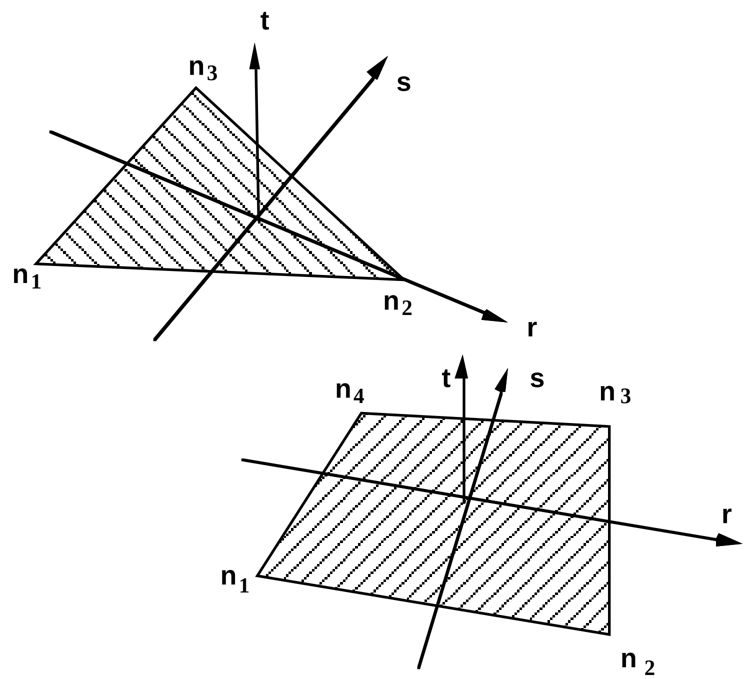
υ_i = segment velocity.

n_i = segment normal vector.

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



3-Dimensional Definition

Figure 25.1. Nodal numbering for pressure cards. Positive pressure acts in the negative *t*direction.

Pressure Boundary Condition Cards

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. Define NUMPRM (see Control Card 3, Columns 51-55) card sets here. Each set consists of 4 cards.

Card 1 (2I10, 4E10.0)

Columns	Quantity	Format
1-10	Shell part ID.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.	I10
11-20	Load curve ID defining the pressure time history,	I10
21-30	V_x, x-component of the vector 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	E10.0 I5
31-40	V_y, y-component of the vector normal to the suface.	E10.0
41-50	V_z, z-component of the vector normal to the suface.	E10.0
51-60	OFF, pressure loads will be discontinued if $\left|\vec{V} \cdot n_{shell}\right| < OFF$ where n_{shell} is the normal vector to the shell element.	E10.0

Card 2 (6E10.0)

Define the box dimensions.

Columns	Quantity	Format
1-10	Xmin	E10.0
11-20	Xmax	E10.0
21-30	Ymin	E10.0
31-40	Ymax	E10.0
41-50	Zmin	E10.0
51-60	Zmax	E10.0

Card 3 (I10, 3E10.0, 2I10)

Columns	Quantity	Format
1-10	Curve ID defining the mask. This curve gives (x,y) pairs of points in a local coordinate system defined by the vector given below. 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 ITABLE=1 curve in the load curve definition section. If no curve ID is given, all elements of part ID, PID, are included with the exception of those deleted by the box.	I10
11-20	x-component of the vector used to project the masking curve onto the surface of part ID, PID. The origin of this vector given on Card 4 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 load curve ID defining the mask is nonzero.	E10.0
21-30	y-component of the vector used to project the masking curve.	E10.0
31-40	z-component of the vector used to project the masking curve.	E10.0
41-50	In or out flag: EQ.0:elements whose center falls inside the projected curve are considered. EQ.1:elements whose center falls outside the projected curve are considered.	I10
51-60	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.	I10

Card 4 (3E10.0)

Columns	Quantity	Format
1-10	x-origin of projection vector.	E10.0
11-20	y-origin of projection vector.	E10.0
21-30	z-origin of projection vector.	E10.0

26. Traction Boundary Cards for Beam Elements
(3I5, E10.0, 2I5)

Define NUMPBC (see Control Card 3, columns 21-25) cards.

Columns	Quantity	Format
1-5	Traction card number	I5
6-10	Load curve number	I5
11-15	Element number *n*	I5
16-25	Multiplier of load curve	E10.0
26-30	Increment *k*	I5
31-35	Direction of applied load (See Figure 26.1) EQ.1: beam *r* axis EQ.2: beam *s* axis (default) EQ.3: beam *t* axis	I5

Omitted cards are automatically generated by incrementing the element number by *k*, i.e., $n^{i+1} = n^i + k$. Positive loads act in the negative *r*, *s*, and *t* direction. The load curve number, multiplier and direction are taken from the first card.

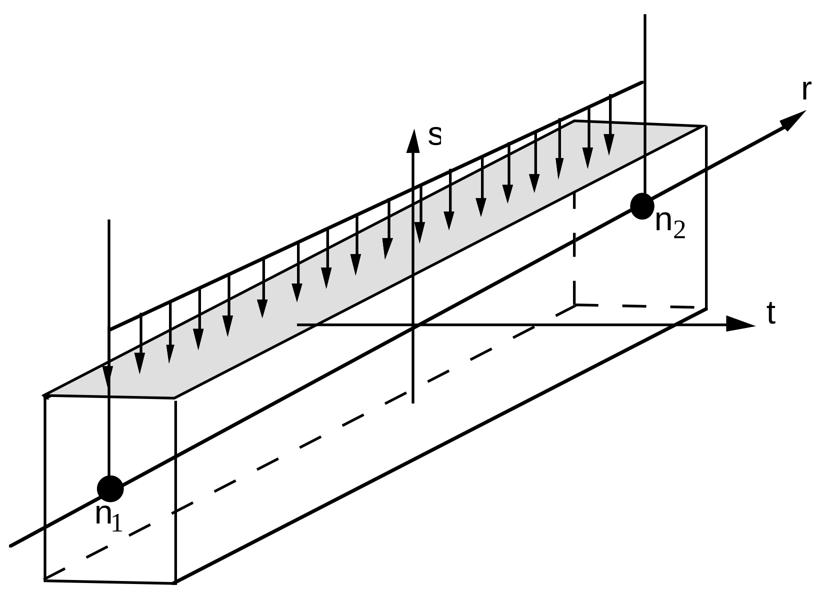


Figure 26.1. Applied traction loads are given in force per unit length.

27. Velocity/Acceleration/Displacement Cards for Nodes/Rigid Bodies

Card 1 (3I5,4E10.0,I5,E10.0,I5),
or (I8,2I5,4E10.0,I5,E10.0,I5) for LARGE option
or (I10,2I5,4E10.0,I5,E10.0,I5) for MLARG
Optional Card 2 (2I10)

Define the number of cards, NUMVC, specified on Control Card 2, columns 11-15. The acceleration flag pertains to nodes only. Translational nodal velocity and acceleration specifications for rigid bodies are allowed and are applied as described at the end of this section.

For the normal or **LARGE** option define the following card(s).

Columns		Quantity	Format
1-5	(1-8)	Nodal point number or rigid body to which this card applies GT.0: nodal point LT.0: absolute value is material number or part ID of the rigid body	I5 (I8)
6-10	(9-13)	Load curve number	I5 (I5)
11-15	(14-18)	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 velocity in direction of vector defined below. Movement on plane normal to the vector is permitted. EQ.-4: translational velocity in direction of vector defined below. Movement on plane normal to the vector is <u>not</u> 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 velocity about vector defined below. Rotation about the normal axes is permitted. EQ.-8: rotational velocity about vector defined below. Rotation about the normal axes is <u>not</u> permitted. This option does not apply to rigid bodies.	I5 (I5)

Columns		Quantity	Format	
		EQ. 9: y and z degrees-of-freedom for node rotatingabout the x-axis at *point (y,z)* defined below. Radial movement of the node is not permitted. EQ.-9: y and z degrees-of-freedom for node rotatingabout the x-axis at *point (y,z)* defined below. Radial movement of the node is permitted EQ. 10: z and x degrees-of-freedom for node rotating about the y-axis at *point (z,x)* defined below. Radial movement of the node is not permitted. EQ.-10: z and x degrees-of-freedom for node rotating about the y-axis at *point (z,x)* defined below. Radial movement of the node is permitted. EQ. 11: x and y degrees-of-freedom for node rotating about the z-axis at *point (xy)* defined below. Radial movement of the node is not permitted. EQ.-11: x and y degrees-of-freedom for node rotating about the z-axis at *point (xy)* defined below. Radial movement of the node is permitted.		
16-25	(19-28)	Scale factor	E10.0	(E10.0)
26-35	(29-38)	x-coordinate of vector	E10.0	(E10.0)
36-45	(39-48)	y-coordinate of vector	E10.0	(E10.0)
46-55	(49-58)	z-coordinate of vector	E10.0	(E10.0)
56-60	(59-63)	velocity/acceleration /displacement flag, IFLAG If IFLAG=4 read card 2 below. 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)	I5	(I5)
61-70	(64-73)	Time imposed motion/constraint is removed, TKILL. EQ.0.0: default set to 10^{28} LT.0.0: TKILL=|TKILL| and card 2 below is read to define the birth time.	E10.0	(E10.0)
71-75	(74-78)	Local coordinate system flag for prescribed motion. The motion is applied in the local coordinate system if this option is active. This option applies ONLY to rigid bodies where the local system is defined in the Material Type 20 input. EQ.0: global system. EQ.1: local system.	I5	(I5)

For the **MLARG** format define the following card.

Columns	Quantity	Format
1-10	Nodal point number or rigid body to which this card GT.0: nodal point LT.0: absolute value is material or property set number of the rigid body	I10
11-15	Load curve number	I5
16-20	Applicable degrees-of-freedom, DOF EQ. 1: x-translational degree-of-freedom EQ. 2: y-translational degree-of-freedom EQ. 3: z-translational degree-of-freedom EQ. 4: translational velocity in direction of vector defined below. Movement on plane normal to the vector is permitted. EQ.-4: translational velocity in direction of vector defined below. Movement on plane normal to the vector is <u>not</u> 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 velocity about vector defined below. Rotation about the normal axes is permitted. EQ.-8: rotational velocity about vector defined below. Rotation about the normal axes is <u>not</u> permitted. This option does not apply to rigid bodies. EQ. 9: y and z degrees-of-freedom for node rotatingabout the he x-axis at *point (y,z)* defined below. Radial movement of the node is not permitted. EQ.-9: y and z degrees-of-freedom for node rotatingabout the he x-axis at *point (y,z)* defined below. Radial movement of the node is permitted EQ. 10: z and x degrees-of-freedom for node rotating about the he y-axis at *point (z,x)* defined below. Radial movement of the node is not permitted. EQ.-10: z and x degrees-of-freedom for node rotating about the he y-axis at *point (z,x)* defined below. Radial movement of the node is permitted. EQ. 11: x and y degrees-of-freedom for node rotating about the he z-axis at *point (xy)* defined below. Radial movement of the node is not permitted. EQ.-11: x and y degrees-of-freedom for node rotating about the he z-axis at *point (xy)* defined below. Radial movement of the node is permitted.	I5

Columns	Quantity	Format
21-30	Scale factor.	E10.0
31-40	x-coordinate of vector or x-offset for DOF=10 and 11.	E10.0
41-50	y-coordinate of vector or z-offset for DOF=9 and 11.	E10.0
51-60	z-coordinate of vector or z-offset for DOF=9 and 10.	E10.0
61-65	velocity/acceleration /displacement flag, IFLAG 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)	I5
66-75	Time imposed motion/constraint is removed, TKILL.q EQ.0.0: default set to 10^{28} LT.0.0: TKILL=|TKILL| and card 2 below is read to define the birth time.	E10.0
76-80	Local coordinate system flag for prescribed motion. The motion is applied in the local coordinate system if this option is active. This option applies ONLY to rigid bodies. EQ.0: global system. EQ.1: local system.	I5

Optional Card 2 (3I10,E10.0)

Define if and only if IFLAG=4 or TKILL<0.0

Columns	Quantity	Format
1-10	Master rigid body for measuring the relative displacement.	I10
11-20	Optional orientation node, n1, for relative displacement	I10
21-30	Optional orientation node, n2, for relative displacement	I10
31-40	TIME, time imposed motion/constraint becomes active.	E10.0

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, TIME, 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 TIME, i.e., (solution time-TIME). Relative displacements that occur prior to reaching TIME are ignored. Only relative displacements that occur after TIME 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 coordinates of the mass center, and x_{node} is the current coordinates of the nodal point.

28. Generalized Stonewall Cards

Define the number of stonewalls, NUMRW, specified on Control Card 4, columns 1-5. Repeat the following set of cards for each stonewall. A stonewall is a planar, cylindrical, spherical, or prismatic surface. Designated slave nodes cannot penetrate.

Card 1
(I5,6E10.0,I5,F5.0,I5), or (I8,6E10.0,I5,F5.0,I2) for LARGE option

Columns		Quantity	Format	
1-5	(1-8)	Number of slave nodes, MAXS EQ.0: all nodes contained in the box defined below are included.	I5	(I8)
6-15	(9-18)	x-coordinate of tail of any outward drawn normal vector, **n**, originating on wall (tail) and terminating in space (head)	E10.0	(E10.0)
16-25	(19-28)	y-coordinate of tail of normal vector **n**	E10.0	(E10.0)
26-35	(29-38)	z-coordinate of tail of normal vector **n**	E10.0	(E10.0)
36-45	(39-48)	x-coordinate of head of normal vector **n**	E10.0	(E10.0)
46-55	(49-58)	y-coordinate of head of normal vector **n**	E10.0	(E10.0)
56-65	(59-68)	z-coordinate of head of normal vector **n**	E10.0	(E10.0)
66-70	(69-73)	Flag for moving stonewall, IMSWF EQ.0: stonewall is fixed in space EQ.1: stonewall has an initial mass and velocity EQ.2: generalized rigid surface has velocity specified by load curve (LIMIT>3) EQ.3: generalized rigid surface has displacement specified by load curve (LIMIT>3)	I5	(I5)
71-75	(74-78)	Stick condition, STICK. Options 2.0 and 3.0 apply only to flat motionless rigid walls. EQ.0.0: frictionless sliding after contact EQ.1.0: no sliding after contact GT.0.0.AND.STICK.LT.1.0: coulomb friction coefficient	F5.0	(F5.0)

Columns	Quantity	Format
	EQ.2.0: node is welded after contact with frictionless sliding. Welding occurs if and only the normal component of the impact velocity exceeds a critical value defined below. EQ.3.0: node is welded after contact with no sliding. Welding occurs if and only the normal component of the impact velocity exceeds a critical value defined below. EQ.-1.0: orthotropic frictional coefficients by defining fixed vectors. EQ.-2.0: orthotropic frictional coefficients by defining nodes.	
76-80 (79-80)	Option for defining the size and shape of the stonewall, extra nodes fixed to stonewall, and softening factor, LIMIT Options 4-7 below are available if and only if IMSWF=0,2, or 3 EQ.0: flat surface extends to infinity EQ.1: size and orientation are defined (Figure 28.1) EQ.2: extra nodes and softening factor EQ.3: activates options 1 and 2 above EQ.4: prescribed motion infinite/finite flat surface (Figure 28.3) EQ.5: prescribed motion infinite/finite cylinder (Figure 28.3) EQ.6: prescribed motion sphere (Figure 28.3) EQ.7: prescribed motion infinite/finite rectangular prism (Figure 28.3)	I5 (I2)

Optional Card for Welding Nodes to Flat Stationary Walls (E10.0)

Define the following optional card only if the stick condition flag is set to 2 or 3 in Columns 71-75 (74-78) above.

Columns	Quantity	Format
1-10	Critical normal velocity component at which nodes welds to wall.	E10.0

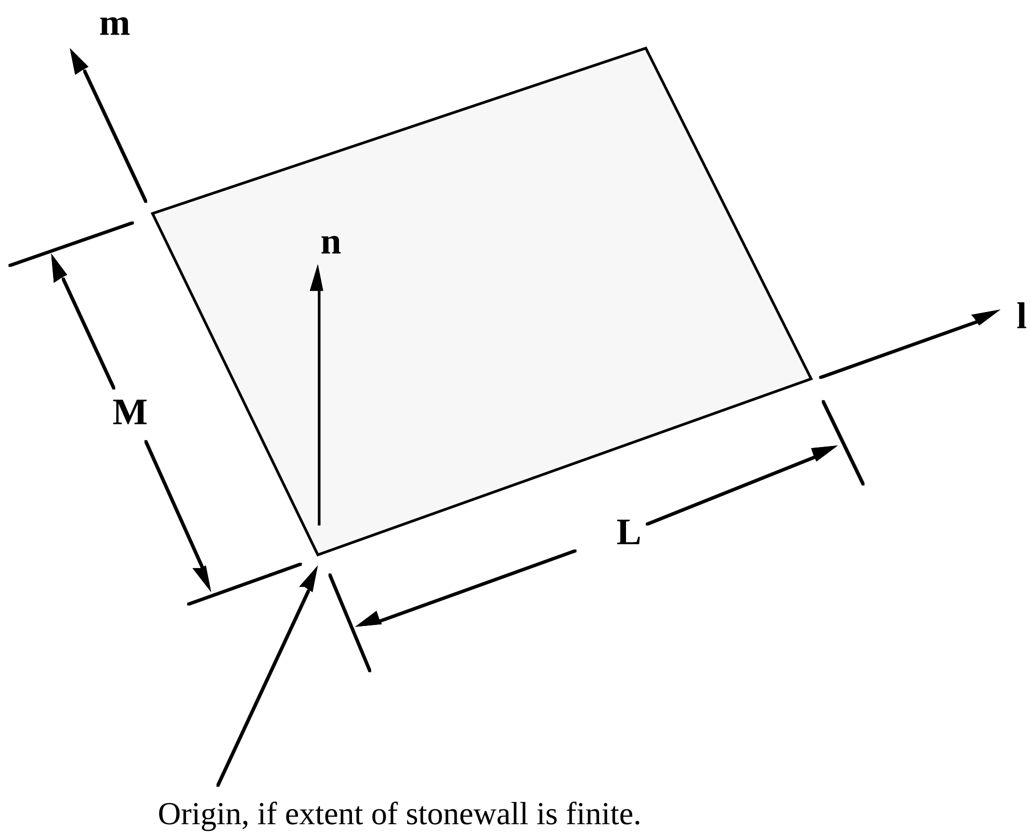


Figure 28.1. Vector **n** is normal to the stonewall. An optional vector **l** can be defined such that **m**=**n**×**l**. The extent of the stonewall is limited by defining **L** and **M**. A zero value for either of these lengths indicates that the stonewall is infinite in that direction.

Optional Card 1 for Orthotropic Friction for Flat Stationary Walls (6E10.0)

Define the following two optional cards only if the stick condition flag is set to -1 or -2 in Columns 71-75 (74-78) above.

Columns	Quantity	Format
1-10	Static friction coefficient in local a-direction, μ_{sa}	E10.0
11-20	Static friction coefficient in local b-direction, μ_{sb}	E10.0
21-30	Dynamic friction coefficient in local a-direction, μ_{ka}	E10.0
31-40	Dynamic friction coefficient in local b-direction, μ_{kb}	E10.0
41-50	Decay constant in local a-direction, d_{va}	E10.0
51-60	Decay constant in local b-direction, d_{vb}	E10.0

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

$$\mu_a = \mu_{ka} + (\mu_{sa}\mu_{ka})e^{d_{va}V_{relative,a}}$$
$$\mu_b = \mu_{kb} + (\mu_{sb}\mu_{kb})e^{d_{vb}V_{relative,b}}$$

Optional Card 2 for Orthotropic Friction for Flat Stationary Walls (3E10.0)

Define a vector $\underset{\sim}{d}$ 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 $\underset{\sim}{n}$ is the normal vector to the stonewall. See Figure 28.2.
If STICK=-1. define the components of $\underset{\sim}{d}$ with the following card:

Columns	Quantity	Format
1-10	d_1, x-component of vector	E10.0
11-20	d_2, y-component of vector	E10.0
21-30	d_3, z-component of vector	E10.0

If STICK=-2. define two nodes defining $\underset{\sim}{d}$ which originates at node 1 and terminates at node 2:

Columns	Quantity	Format
1-10	Node 1	E10.0
11-20	Node 2	E10.0

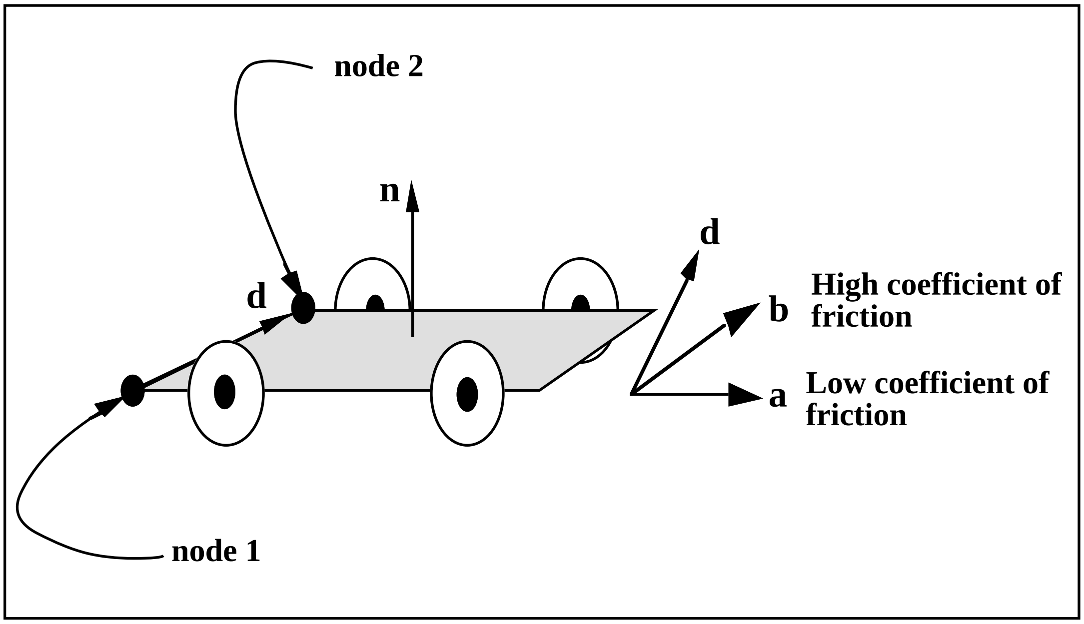


Figure 28.2. Definition of orthotropic friction vectors. The two methods of defining the vector, $\underset{\sim}{d}$ are shown. If vector $\underset{\sim}{d}$ is defined by nodes 1 and 2, the local coordinate system may rotate; otherwise, $\underset{\sim}{d}$ is fixed in space and the local system is stationary.

Card 2 (8E10.0)
Infinite/finite flat surface (LIMIT=1, 3, or 4)

Columns	Quantity	Format
1-10	x-coordinate of head of edge vector **l**	E10.0
11-20	y-coordinate of head of edge vector **l**	E10.0
21-30	z-coordinate of head of edge vector **l**	E10.0

Columns	Quantity	Format
31-40	Length of **l** edge, **L** EQ.0.0: extends from negative to positive infinity	E10.0
41-50	Length of **m** edge, **M** EQ.0.0: extends from negative to positive infinity **(edge vectors l and m should not both be zero for LIMIT=1 or 3).**	E10.0

Infinite/finite cylindrical surface (LIMIT=5)

Columns	Quantity	Format
1-10	Radius of cylinder, R	E10.0
11-20	Length of cylinder, L EQ.0.0: extends from negative to positive infinity	E10.0

Spherical surface (LIMIT=6)

Columns	Quantity	Format
1-10	Radius of sphere, R	E10.0

Infinite/finite rectangular prism (LIMIT=7)

Columns	Quantity	Format
1-10	x-coordinate of head of edge vector **l**	E10.0
11-20	y-coordinate of head of edge vector **l**	E10.0
21-30	z-coordinate of head of edge vector **l**	E10.0
31-40	Length of **l** edge EQ.0.0: extends from negative to positive infinity	E10.0
41-50	Length of **m** edge EQ.0.0: extends from negative to positive infinity	E10.0
51-60	Length of prism, L EQ.0.0: extends from negative to positive infinity	E10.0

Optional Card (IMSWF=1)
(2E10.0)

Define the following card for IMSWF=1.

Columns	Quantity	Format
1-10	Total mass of stonewall	E10.0
11-20	Initial velocity of stonewall in direction of defining vector, **n**	E10.0

Optional Card (IMSWF=2,3)
(I5,3E10.0)

Define the following card for IMSWF=2 or 3 giving the load curve number and direction cosine vector which determines the direction of stonewall motion.

Columns	Quantity	Format
1-5	Load curve number	I5
6-15	x-direction cosine of vector **V**	E10.0
16-25	y-direction cosine of vector **V**	E10.0
26-35	z-direction cosine of vector **V**	E10.0

Define the following card if and only LIMIT =2 or 3.

Optional Card (LIMIT=2,3)
(6I10)

Columns	Quantity	Format
1-10	Nodal point n_1 for viewing stonewall in output. This node and the 3 that follow will have their motion updated if the wall is moving. If a single element is defined for viewing the wall, the part ID of the single element must be deformable or else the segment will be treated as a rigid body and the nodes will have their motion modified independently of the stonewall.	I10
11-20	Nodal point n_2 for viewing stonewall in output	I10
21-30	Nodal point n_3 for viewing stonewall in output	I10
31-40	Nodal point n_4 for viewing stonewall in output	I10
41-50	Number of cycles to zero velocity	I10
51-60	Number of four node segments, *NS*, defining areas for computing resultant forces where NS<NRWSEG+1. These segments translate with the moving stonewall and allow the force distribution to be determined. The resultant forces are written in file “RWFORC.” See Control Card 7 where NS must not exceed NRWSEG defined in columns 16-20.	I10

Define *NS* segments for resultant force calculations.

Optional Card (NS>0)
(4I10)

Columns	Quantity	Format
1-10	Nodal point n_1	I10
11-20	Nodal point n_2	I10
21-30	Nodal point n_3	I10
31-40	Nodal point n_4	I10

Optional Card (MAXS>0)
(2I5), or (2I8) for LARGE option

Define the slave nodes for the stonewall if MAXS is nonzero.

Columns		Quantity	Format
1-5	(1-8)	Slave number. (see notes below)	I5 (I8)
6-10	(9-16)	Nodal point number	I5 (I8)

If the slave number and corresponding node number are defined as zero for the first node, all nodes in the mesh are assumed to be slaves to the stonewall. If the slave number is 1 and the corresponding node number is zero, additional input is required to define the nodes that are exempted from the slave list. (No additional input is required above.)

Omitted slave nodes are automatically generated by incrementing the nodal point numbers by

$$\frac{n_i - n_j}{sn_i - sn_j}$$

where sn_i and sn_j are slave numbers on two successive cards and n_i and n_j are their corresponding node numbers. If arbitrary numbering is used automatic generation is not recommended.

Optional Card (MAXS=0)
(6E10.0)

If MAXS is set to zero all nodes found in the following volume definition are included as slave nodes for the generalized stonewall.

Columns	Quantity	Format
1-10	Xmin	E10.0
11-20	Xmax	E10.0
21-30	Ymin	E10.0

Columns	Quantity	Format
31-40	Ymax	E10.0
41-50	Zmin	E10.0
51-60	Zmax	E10.0

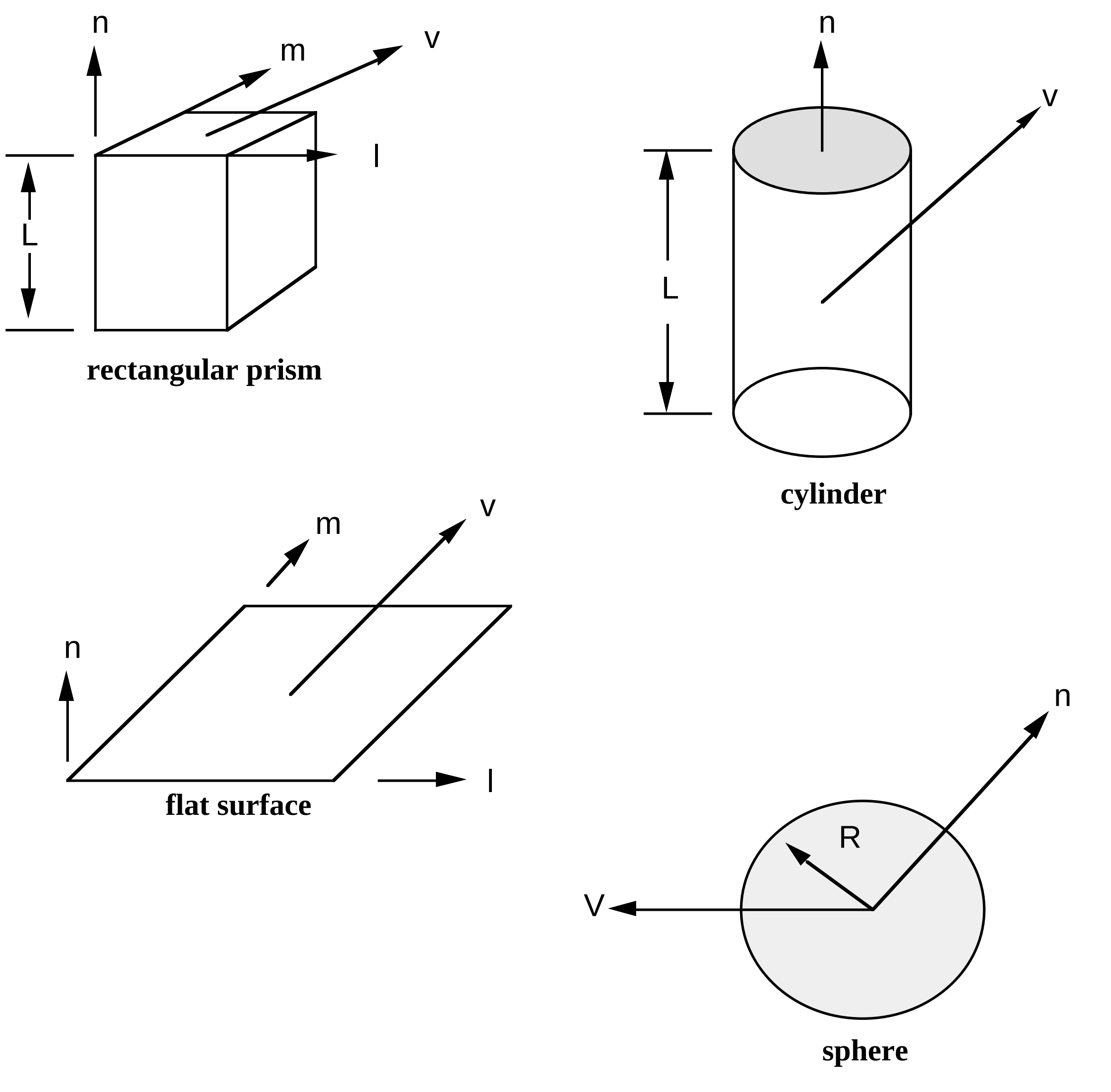


Figure 28.3. Vector **n** determines the orientation of the generalized stonewalls. For the prescribed motion options the wall can be moved in the direction **V** as shown.

Define the number of nodes that are not slave to the stonewall, NUMMSN.

Columns	Quantity	Format
1-5	NUMMSN, number of exempted nodes	I5

Define NUMMSN nodes that are exempted from the slave nodes list for the stonewall.

Columns	Quantity	Format
1-5 (1-8)	Exempted node number	I5 (I8)

29. Nodal Force Groups

Define the number of groups, NODEFR, specified on Control Card 7, columns 6-10. Repeat the following set of cards for each force group. This option will only work on interface nodes, surface nodes, and boundary nodes, and not on internal nodes.

Card 1
(2I10)

Columns	Quantity	Format
1-10	Number of nodes, *NN*	I10
11-20	Local coordinate system number (optional)	I10

Card 2
(2I5), or (2I8) for LARGE option

Columns		Quantity	Format
1-5	(1-8)	Number $\leq NN$	I5 (18)
6-10	(9-18)	Nodal point number	I5 (18)

The nodal and resultant forces are printed for the nodal force group. The resultants are also printed in the optional local system.

Omitted nodes are automatically generated by incrementing the nodal point numbers by

$$\frac{n_i - n_j}{sn_i - sn_j}$$

where sn_i and sn_j are numbers on two successive cards and n_i and n_j are their corresponding node numbers.

30. Nodal Constraint and Spotweld Cards

Define the number of nodal constraint sets, NUMCC, specified on Control Card 4, columns 31-35. **Nodes of a nodal constraint set 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. Please note that this option may lead to nonphysical constraints under some conditions. See Figure 30.1.**

Card 1
(2I5,5E10.0), or (I8,I5,5E10.0) for LARGE option

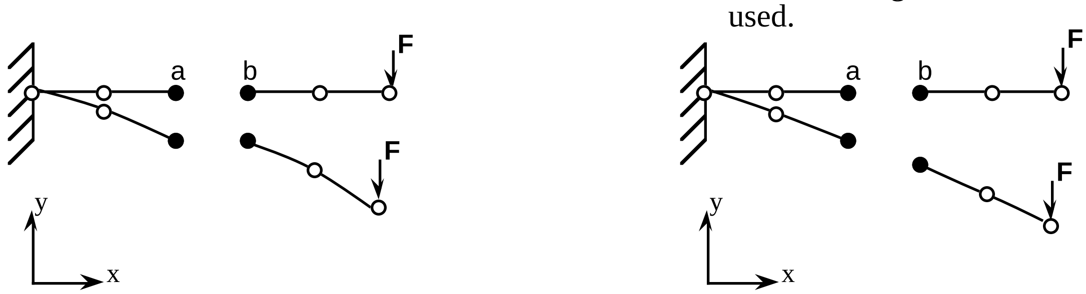
Columns		Quantity	Format
1-5	(1-8)	Number of nodes that share at least one degree-of-freedom, >1	I5 (I8)
6-10	(9-13)	Degrees-of-freedom in common, DOFC, see Figure 30.1 EQ.-2: rigid massless beam - spotweld (filtered force) EQ.-1: rigid massless beam - spotweld EQ.0: rigid massless truss - rivet EQ.1: 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)	I5 (I5)
11-20	(14-23)	S_n, normal force at failure (spotweld option only)	E10.0
21-30	(24-33)	S_s, shear force at failure (spotweld option only)	E10.0
31-40	(34-43)	n, exponent for normal force (spotweld option only)	E10.0
41-50	(44-53)	m, exponent for shear force (spotweld option only)	E10.0
51-60	(54-63)	Failure time for constraint set, t_f. (default=1.0E+20)	E10.0
61-70	(64-73)	ε^p_{fail}, effective plastic strain at failure (spotweld option only)	E10.0

Define Card 2 if and only if DOFC=-2
(I5,E10.0)

Columns	Quantity	Format
1-5	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. 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.	I5
6-15	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	E10.0

Constrained Node Sets option = 7. Since no rotation is permitted, this option should <u>not</u> be used to model rigid body behavior that involves rotations.

Constrained Node Sets option = -1. Behavior is like a rigid beam. This option may be used to model spotwelds. If more than 2 nodes are involved then nodal rigid bodies should be used.



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

Figure 30.1. Constrained node sets can lead to nonphysical responses. Great care should be used in applying this option especially if DOFC>0.

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. Tensile values of f_n are considered in the failure criteria.

Spotweld 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 in slightly increased run times. Failures can include both the plastic and brittle failures.

Only two nodes may be connected together using the beam or truss options (-1,0). If the beam option is used (-1) then the nodes must be connected to nodes having rotary inertias, i.e., beams or shells. Note that shell elements do not have rotary stiffness in the normal direction and, therefore this component will not be transmitted through the rigid, massless beam. **The rigid beams and trusses must have a finite length.** Coincident nodes in spotweld can be handeled as nodal rigid bodies in Section 36.

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

(Card 2)
(16I5), or (10I8) for LARGE option

Columns		Quantity	Format
1-5	(1-8)	Nodal point number of first node to be tied	I5 (I8)
6-10	(9-16)	Nodal point number of second node to be tied	I5 (I8)
11-15	(17-24)	Nodal point number of third node to be tied	I5 (I8)
.	.	.	.
.	.	.	.
.	.	.	.

Define as many cards as necessary.

31. Initial Velocities

(I5,3E10.0,I5), or (I8,3E10.0,I5) for LARGE option

Skip this section if the initial condition parameter, INITV, on Control Card 11, columns 1-5 is zero.

(INITV=1,[5 for arbitrary numbering])

Columns		Quantity	Format	
1-5	(1-8)	Nodal point number	I5	(I8)
6-15	(9-18)	Initial velocity in x-direction	E10.0	(E10.0)
16-25	(19-28)	Initial velocity in y-direction	E10.0	(E10.0)
26-35	(29-38)	Initial velocity in z-direction	E10.0	(E10.0)
36-40	(39-43)	Node increment (no sorting, Control Card 11 columns 6-10) EQ.0: default set to 1	I5	(I5)
41-50	(44-53)	x-angular velocity	E10.0	
51-60	(54-63)	y-angular velocity	E10.0	
61-70	(64-73)	z-angular velocity	E10.0	

If arbitrary node and element numbers are assumed (IARB=1) then each node must be defined.

(INITV=2 or 3)

Columns	Quantity	Format
1-10	x-velocity	E10.0
11-20	y-velocity	E10.0
21-30	z-velocity	E10.0
31-40	x-angular velocity	E10.0
41-50	y-angular velocity	E10.0
51-60	z-angular velocity	E10.0

The above velocity applies to all nodes if INITV=2. If INITV=3 then the nodes defined below are assigned unique velocities. INITV is defined in columns 1-5 on the 11th control card.

Define the following cards if and only if INITV=3.

Cards 1 (I5)

Columns	Quantity	Format
1-5	NUMNOD, the number of nodes that are assigned velocities other than the default.	I5

Cards 2,...,NUMNOD+1 (I5,6E10.0)

Columns		Quantity	Format
1-5	(1-8)	Node numbers	I5 (I8)
6-15	(9-18)	x-velocity	E10.0
16-25	(19-28)	y-velocity	E10.0
26-35	(29-38)	z-velocity	E10.0
36-45	(39-48)	x-angular velocity	E10.0
46-55	(49-58)	y-angular velocity	E10.0
56-65	(59-68)	z-angular velocity	E10.0

Define the following cards if and only if INITV=4.

Cards 1 (I5)

Columns	Quantity	Format
1-5	Number of boxes, NBOX	I5

Define the following NBOX card sets to specify the box dimensions and the velocity of nodal points lying within the box.

Cards 2,4,.... (6E10.0)

Columns	Quantity	Format
1-10	Xmin	E10.0
11-20	Xmax	E10.0
21-30	Ymin	E10.0
31-40	Ymax	E10.0
41-50	Zmin	E10.0
51-60	Zmax	E10.0

Cards 3,5,.... (3E10.0)

Columns	Quantity	Format
1-10	x-translational velocity	E10.0
11-20	y-translational velocity	E10.0
21-30	z-translational velocity	E10.0
31-40	x-angular velocity	E10.0
41-50	y-angular velocity	E10.0
51-60	z-angular velocity	E10.0

Initial Velocities

Define the following cards if and only if INITV=6. The velocities are initialized in the order the cards are defined. Later cards may overwrite the velocities previously set.

(INITV=6 and INITV=7) Card 1 (I5)

Columns	Quantity	Format
1-5	Number of material card sets to be read, NMSETS	I5
6-10	Number of nodal card sets to be read, NNSETS	I5

(INITV=6 and INITV=7) Define NMSETS Card Sets Below

Card 1 (I10,7E10.0)

Columns	Quantity	Format
1-10	Material ID EQ.0: Set all nodal velocities EQ.n: Initialize velocities of material n only	I10
11-20	Angular velocity	E10.0
21-30	x-coordinate on rotational axis	E10.0
31-40	y-coordinate on rotational axis	E10.0
41-50	z-coordinate on rotational axis	E10.0
51-60	x-direction cosine	E10.0
61-70	y-direction cosine	E10.0
71-80	z-direction cosine	E10.0

Card 2 (3E10.0)

Columns	Quantity	Format
1-10	x-rigid body velocity	E10.0
11-20	y-rigid body velocity	E10.0
21-30	z-rigid body velocity	E10.0

(INITV=6 and INITV=7) Define NNSETS Card Sets Below

Card 1 (I10,7E10.0)

Columns	Quantity	Format
1-10	Number of nodal points to be defined below.	I10
11-20	Angular velocity	E10.0
21-30	x-coordinate on rotational axis	E10.0
31-40	y-coordinate on rotational axis	E10.0
41-50	z-coordinate on rotational axis	E10.0
51-60	x-direction cosine	E10.0
61-70	y-direction cosine	E10.0
71-80	z-direction cosine	E10.0

Card 2 (3E10.0)

Columns	Quantity	Format
1-10	global x-rigid body velocity	E10.0
11-20	global y-rigid body velocity	E10.0
21-30	global z-rigid body velocity	E10.0

Nodal Point ID's (10I8)

Define NNOD nodal points with ten nodal ID's per card.

Columns	Quantity	Format
1-8	Slave node n_1	I8
9-16	Slave node n_2	I8
. .	.	.
. .	.	.
. .	.	.

32. Contact Interface Definitions

In this section define the number of sliding interfaces specified on Control Card 4, columns 6-10. Define NUMSV control card sets. A second card is optional and additional cards are required for the automatic generation options.

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.

Optional Part ID's for Flagged for Interior Contact
(I10,3E10.0)

Define the number of part ID's specified in columns 66-70 of Control Card 4 below. Use as many cards as necessary.

Columns	Quantity	Format
1-10	First material or part ID to be included.	I10
11-20	PSF, penalty scale factor (Default=1.00).	E10.0
21-30	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.	E10.0
31-40	ED, Optional modulus for interior contact stiffness.	E10.0

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 defined on Control Card 15 in columns 1-10, *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.

Interface Control Card Sets 1,2,...,NUMSV
Control Card 1
(2I5,I2,A1,I2,3E10.0,i4,i1,2I5,2E10.0), or
(2I8,I2,A1,I2,3E10.0,i4,i1,2I5,2E7.0) for LARGE option

Columns		Quantity	Format
1-5	(1-8)	Number of slave segments (1-4, 9, 10, 13, 14, 15, and 17), nodes (5-8, 16, and 18) EQ.0: automatic generation option (all contact types)	I5 (I8)
6-10	(9-16)	Number of master segments (NMS). Set this to zero for types 4, 13, and 15. EQ.0: automatic generation option (all contact types) LT.0: interface number |NMS| is used from a database generated in a previous calculation via Section 11 input. This option applies to type 2 interfaces only. A description of this capability is provided in the Introduction to this manual.	I5 (I8)
11-12	(17-18)	IREAD, flag to read additonal contact control cards EQ.0: no additional cards are read EQ.1: second control card is read EQ.2: second and third control cards are read EQ.3: second, third and thermal contact control cards are read	I2 (I2)
13-15	(19-21)	Type number (the letters ‘a’, ‘m’, 'o' and ‘p’ must be in col. 13) 1-sliding without penalties p 1-symmetric sliding with penalties (recommended) 2-tied, see note 1 below. f 2-tied with failure. o 2-tied with offsets permitted, see note 1 below. 3-sliding, impact, friction a 3-sliding, impact, friction, no segment orientation m 3-sliding, impact, friction - metalforming option. 4-single surface contact 5-discrete nodes impacting surface a 5-discrete nodes impacting surface, no segment orientation m 5-discrete nodes impacting surface - metalforming option. 6-discrete nodes tied to surface, see note 1 below. o 6-discrete nodes tied to surface with offsets permitted	A3 (A3)

Control Card 1 (continued)

Columns	Quantity	Format
	7-shell edge tied to shell surface, see note 1 below. o 7-shell edge tied to shell surface with offsets permitted 8-nodes spot welded to surface 9-tiebreak interface 10-one way treatment of sliding, impact, friction a10-one way treatment, no segment orientation m10-one way treatment of sliding, impact, friction - metal forming option. 11-box/material limited automatic contact for shells* 12-automatic contact for shells (no additional input required) See note 2 below. 13- automatic single surface with beams and arbitrary orientations. See note 3 below. a13- like above but with extra searching for airbag contact 14- surface to surface eroding contact 15-single surface eroding contact 16-node to surface eroding contact 17-surface to surface symmetic/asymmetric constraint method 18-node to surface constraint method [Taylor and Flanagan 1989] 19-rigid body to rigid body contact with arbitrary force/ deflection curve. (This option may be used with deformable bodies.) 20-rigid nodes to rigid body contact with arbitrary force/ deflection curve. (This option may be used with deformable bodies.) 21-rigid body to rigid body contact with arbitrary force/deflection curve. Unlike option 19, this is a one way treatment. (This option may be used with deformable bodies.) 22-single edge treatment for shell surface edge to edge treatement. 23-simulated draw bead 25-force transducer contact for <u>penalty</u> based contact types. Not for types 2, 6, 7, 17, and 18. See type 27 below for the constraint type. 26-automatic single surface, beams-to-beam, beam-to-shell edge 27-force transducer contact for <u>constraint</u> based contact types. Applies to types 2, 6, 7, 17, and 18 only.	

Columns		Quantity	Format	
16-25	(22-31)	Static coefficient of friction, μ_s For "f 2" contact define the normal tensile stress at failure. EQ.-1: part based friction coefficients are used. Applies to contact types a 3, a 5, a10, 13, 15, and 26 only. EQ. 2: For contact types 5 and 10, the dynamic coefficient of friction points to the table 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.	E10.0	(E10.0)
26-35	(32-41)	Dynamic coefficient of friction, μ_k. For "f 2" contact define the shear stress at failure.	E10.0	(E10.0)
36-45	(42-51)	Exponential decay coefficient, d_v.	E10.0	(E10.0)
46-49	(52-55)	Optional load curve ID defining the resisting stress versus gap opening for the post failure response in type 9 contact.	I4	(I4)
50-50	(56-56)	ISRCH, small penetration in contact search option. If the slave node penetrates more than XPENE*segment thickness (See Control Card 15, Cols. 66-75, for the definition of XPENE), the penetration is ignored and the slave node is set free. The segment thickness is taken as the shell/solid thickness if the segment belongs to a shell/solid element . This option applies to the surface to surface contact algorithms which do not consider thickness, ISHLTK=0 on Control Card 15, Cols 26-30. EQ.0: check is off EQ.1: check is on EQ.2: check is on but shortest diagonal is used Also, see PENMAX on the third optional control card below and Table 31.1.	I1	(I1)
51-55	(57-61)	Include slave side in printed and binary force interface file EQ.0: no EQ.1: yes	I5	(I5)
56-60	(62-66)	Include master side in printed and binary force interface file EQ.0: no EQ.1: yes	I5	(I5)
61-70	(67-73)	Scale factor on default slave penalty stiffness (default=1.0)	E10.0	(E7.0)
71-80	(74-80)	Scale factor on default master penalty stiffness (default=1.0)	E10.0	(E7.0)

Notes:

1. The tied interface contact definitions, types 2, 6, and 7, 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., o 2, o 6, and o 7. Also, it may sometimes be advantageous to use the extra nodes for rigid body option instead of 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.

2. Due to the new automated input options, contact types 11 and 12 are no longer needed. However, for compatibility LS-DYNA will continue to read input prepared for these contact types based on the 1991 LS-DYNA manual. Type 13 contact now effectively replaces types 11 and 12 and is also more general since it includes segments from brick elements automatically.

3. Type 13 automatic single surface, which completely eliminates the need for segment orientation, is sufficiently robust that extremely complex geometries presents no difficulties for the sophisticated searching algorithm. Beam elements are included with the automatic generation options but only beam node to shell segment contact is handled. Thickness is taken into account. The modification of this new contact for the airbag contact (type a13) increases its searching cost considerably. Due to the complexity of the contact in folded configurations a special algorithm is desirable.

INTERFACE TYPE ID	PENCHK	ELEMENT TYPE	FORMULA FOR RELEASE OF PENETRATING NODAL POINT
1, 2, 6, 7	———	———	------------------------------
3, 5, 8, 9, 10 (without thickness)	0	solid	d=PENMAX if and only if PENMAX>0 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) 17, and 18	———	solid	d=XPENE*thickness of solid element
		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 31.1. 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.

Optional Control Card 2, define if IREAD > 0 (8E10.0)

Columns	Quantity	Format
1-10	Coefficient for viscous friction	E10.0
11-20	**For contact type 3-10 and 23** Viscous damping coefficient in percent of critical. **For contact type 17:** Kinematic partition factor for constraint EQ. 0.0: fully symmetic treatment EQ. 1.0: one way treatment with slave nodes constrained to master surface EQ.-1.0: one way treatment with master nodes constrained to slave surface	E10.0
21-30	Optional thickness for slave surface (overrides true thickness) GT.0.0.AND.LE.1.E-04: thickness is ignored. This option allows models set up for the old Type 3 contact without thickness offsets to be analyzed with the new formulation.	E10.0
31-40	Optional thickness for master surface (overrides true thickness) GT.0.0.AND.LE.1.E-04: Thickness is ignored. This option allows models set up for the old Type 3 contact without thickness offsets to be analyzed with the new formulation.	E10.0
41-50	Scale factor for slave surface thickness (scales true thickness) EQ.0.0: default set to 1.0	E10.0
51-60	Scale factor for master surface thickness (scales true thickness) EQ.0.0: default set to 1.0	E10.0
61-70	Birth time (contact surface becomes active at this time)	E10.0
71-80	Death time (contact surface is deactivated at this time) EQ.0.0: default set to 1.0e+20	E10.0

Optional Control Card 3, define if IREAD = 2
(I2, I3, E10.0, I5, 2E10.0, 3I5, E10.0, 3I5)

Columns	Quantity	Format
1-2	Segment searching option EQ.0: search 2D elements (shells) before 3D elements (solids,thick shells) when locating segments. EQ.1: search 3D elements (solids, thick shells) before 2D elements (shells) when locating segments.	I2
3-5	Soft constraint option EQ.0: penalty formulation EQ.1: soft constraint formulation EQ.2: alternate penalty formulation	I3
6-15	Scale factor for constraint forces of soft contraint option (default=.10). Values greater than .5 for single surface contact and 1.0 for a one way treatment are inadmissible.	E10.0
16-20	Load curve ID defining airbag thickness as a function of time for type a13 contact.	I5
21-30	Maximum parametric coordinate in segment search (values 1.025 and 1.20 recommended. Larger values can increase cost.) EQ.0.0: default set to 1.025	E10.0
31-40	EDGE, activates an edge-edge penetration check for the alternate penalty formulation in columns 3-5 EQ.0: Check only surface penetrations (default).\ GT.0: Check both surface and edge-edge penetrations.	E10.0
41-45	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. EQ.0: default set to 2	I5
46-50	Number of cycles between bucket sorts, ncbbs. A value of 25 is recommended for contact type 4 and 100 is recommend for contact type 13. Values of 10-15 are used for the surface to surface contact. EQ.0: LS-DYNA determines interval, LT.0: |ncbbs| load curve ID defining bucket sorting frequency versus time.	I5

51-55	Number of cycles between contact force updates for penalty. contact formuations. 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. EQ.0: force calculations are performed each cycle.	I5
56-65	PENMAX, maximum penetration allowed for old contact types 3, 5, and 10 or the segment thickness multiplied by PENMAX defines the maximum penetration allowed for contact types a 3, a 5, a10, m 3, m 5, m10 and type 13.	E10.0
66-70	Thickness option for contact types 3, 5, and 10, THKOPT: EQ.0: default is taken from control card, EQ.1: thickness offsets are included, EQ.2: thickness effects are not included (old way).	I5
71-75	Define the following parameter if THKOPT equals 1 above. EQ.0: thickness is not considered EQ.1: thickness is considered but rigid bodies are excluded EQ.2: thickness is considered including rigid bodies	I5
76-80	Disable logic in thickness offset contact to avoid shooting nodes: EQ.0: logic is enabled (default), EQ.1: logic is skipped (sometimes recommended for metal forming calculations).	I5

The alternate penalty formulation flag in columns 3-5 of optional card 3 activates a general contact algorithm for general shell and solid element contact. This option is available for sliding interface types 3, 4, 10, 13, a3, and a10. When type a3, a10, or 13 are used, orientation of shell segment normals is automatic. Otherwise, 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 type 10 or a10 in which case the master segment normals are used as input by the user. The EDGE parameter in columns 31-40 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 recommend. Smaller values may be tried if problems occur when the EDGE option is active.

Thermal Control Card, Define if IREAD = 3
(5E10.0,5x,E10.0)

Columns	Quantity	Format
1-10	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.	E10.0
11-20	Radiation conductance (h_{rad}) between the slide surfaces If a gap exists between the slide surfaces, then the contact conductance is calculated by $$h = h_{cond} + h_{rad}$$	E10.0
21-30	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 defined in columns 31-40	E10.0
31-40	l_{min}, use the heat transfer conductance defined in columns 21-30 for gap thicknesses less than this value.	E10.0
41-50	No thermal contact if gap is greater than this value (l_{max})	E10.0
51-55	blank	5x
56-65	characteristic length multiplier for search algorithm EQ.0: default set to 1. search radius = (multiplier) (largest element diagonal)	E10.0

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 $l_{gap} > l_{max}$

Control Card 4, Define for contact types 19, 20, and 21 only (3E10.0)

Columns	Quantity	Format
1-10	Load curve number giving force versus penetration behavior for the interface nodes.	E10.0
11-20	Force calculation method: EQ.1.0: loadcurve gives total normal force on surface versus maximum penetration of any node (Type 21 only), EQ.2.0: load curve gives normal force on each node versus penetration of node through the surface (Types 19, 20, and 21 only), EQ.3.0: load curve give normal pressure versus penetration of node into surface. (Types 19 and 21 only), EQ.4.0: load curve give total normal force versus maximum soft penetration. (Type 21 only). In this case the force will be followed based on the original penetration point.	E10.0
21-30	Unloading stiffness option. The default is to unload along the loading curve.	E10.0

Control Card 4, Define for contact type 23 only. Simulated draw bead (5E10.0)

With the draw bead model, the effect of draw beads in metalforming can be approximately modelled by a line of nodes. Thickness options must be used in the contact when draw beads are included, that is, ISHLTK in columns 26-30 on control card 15 must be set to 1 or 2.

Columns	Quantity	Format
1-10	Load curve ID giving the bending component of the retaining force, $F_{bending}$, per unit draw bead length as a function of displacement, δ. See Figure 32.1. 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 the load curve ID is negative then the absolute value gives the load curve ID defining max bead force versus normalized drawbead length. The abscissa value 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.	E10.0
11-20	Load curve ID giving the normal force per unit draw bead length as a function of displacement, δ. See Figure 32.1. This force is due to the bending 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.	E10.0
21-30	Draw bead depth. See Figure 32.1.	E10.0
31-40	Scale factor for load curve. Default=1.0	E10.0
41-50	Number of integration points along the draw bead. EQ.0:Internally calculated based on element size of elements that interact with draw bead.	E10.0

The draw bead is defined by a consecutive list of 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 the draw bead nodal points do not need to belong to the element connectivities of the die and binder.

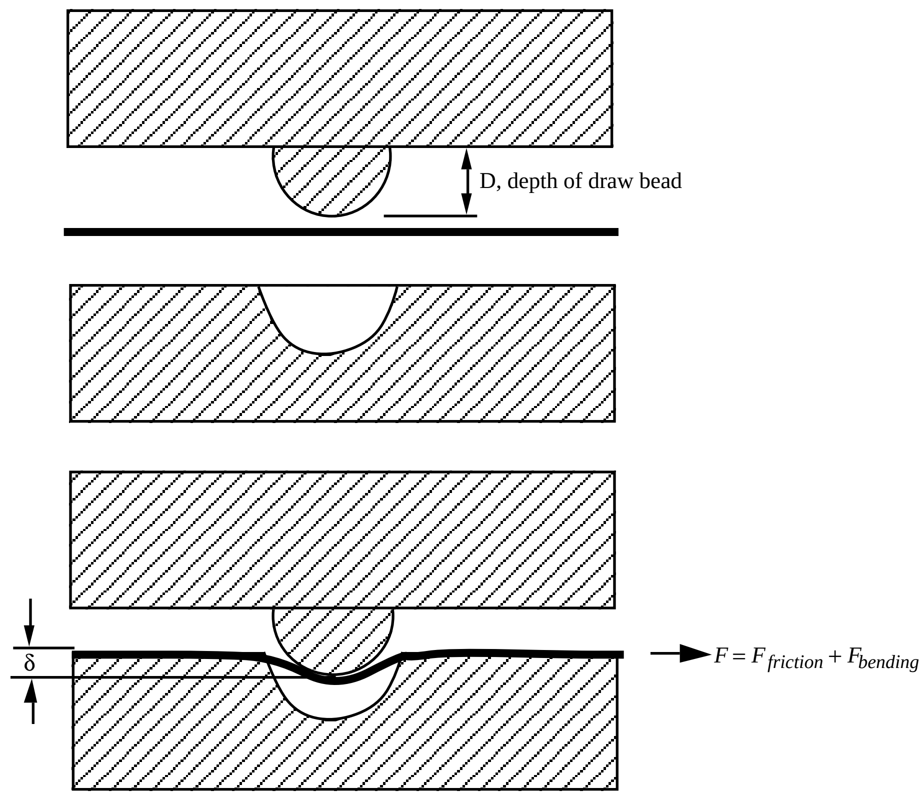


Figure 32.1. Draw bead contact model defines a resisting force as a function of draw bead displacement.

Optional Interface Control Cards for Automatic Contact Input Generation
Defined if automatic generation for slave and/or master surface is desired

Automatic contact control cards belong to the control card section of the specified sliding interface; thus they have to be input IMMEDIATELY after Section 31 control cards 1 and 2. If the number of slave nodes or segments is input as zero, one automatic contact section must be defined. If the number of master segments is input as zero and the single surface contact (type 4 and 13) is not specified then a second set of automatic contact cards will be read. When using solid and brick shell elements with eroding contact, the automatic input should be used since this allows for the contact surface to continuously change as elements are eliminated due to erosion. The automatic input may apply to only the slave definition, the master definition, or to both.

Optional Control Card 5 (4I5,6E10.0)

Columns	Quantity	Format
1-5	Number of materials for exterior boundary determination EQ.0: all materials are included EQ.n: *n* material numbers are defined below EQ.-n: all materials excluding |n| defined below	I5
6-10	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).	I5
11-15	Erosion/Interior node option EQ.0: only exterior boundary information is saved EQ.1: storage is allocated so that eroding contact can occur. For type 15 contact, this option will make all of the bodies' nodes into slaves.	I5
16-20	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.	I5
21-30	Coulomb friction scale factor	E10.0

Columns	Quantity	Format
31-40	Viscous friction scale factor	E10.0
41-50	Normal failure stress (Type 9 contact only) Normal force at failure (Type 8 contact only)	E10.0
51-60	Shear failure stress (Type 9 contact only) Shear force at failure (Type 8 contact only)	E10.0
61-70	*n*, exponent for normal force (Type 8 contact only)	E10.0
71-80	*m*, exponent for shear force (Type 8 contact only)	E10.0

Optional Control Card 6 (Box Definition) (6E10.0)

All materials belonging to shell, thick shell, and solid elements in the specified box are taken into account if all materials are included in the automatic exterior boundary determination on local control card 3. Beam elements are also included for contact Type 13. Alternatively, only the materials specified on optional control cards 6,7,... will be taken into account within the box.

Columns	Quantity	Format
1-10	Minimum x-coordinate (Default=-1.E+20)	E10.0
11-20	Maximum x-coordinate (Default= 1.E+20)	E10.0
21-30	Minimum y-coordinate (Default=-1.E+20)	E10.0
31-40	Maximum y-coordinate (Default= 1.E+20)	E10.0
41-50	Minimum z-coordinate (Default=-1.E+20)	E10.0
51-60	Maximum z-coordinate (Default= 1.E+20)	E10.0

If the maximum and minimum values in a given direction are equal the defaults are taken.

Optional Control Cards 7,8,9,...,..(Material List) (16I5) or (8I10) for MLARG

Automatic generation includes all brick, brick shell, and shell elements. Material ID's can be included for beam elements with contact Types 13 and 26 only. Current the contact with beams assumes a circular cross section for the beam element regardless of the shape of the cross section. The n materials listed below are either included (n is positive) or excluded (n is negative).

Columns		Quantity	Format
1-5	(1-10)	First material number	I5 (I10)
6-10	(11-20)	Second material number	I5 (I10)
11-15	(21-30)	Third material number	I5 (I10)

Sliding Interface Definition Cards. (Follows control card section above)

The following cards apply to the manual input options only. If the number of slave nodes or segments is zero and if the number of master segments is zero, then this section can be skipped.

Define the following slave segment cards for sliding interface types 1-4, 9, 10, 13, 17, 25, and 26. If the slide surface type is 5-8, 16, or 18, then skip the segment definitions and go to the node number definitions below.

(6I5,4E10.0), or (5I8,4E10.0) for LARGE option

Columns		Quantity	Format	
1-5	(1-8)	Slave segment number	I5	(I8)
6-10	(-)	Increment, *k* for generating missing segments	I5	(-)
11-15	(9-16)	Nodal point n_1	I5	(I8)
16-20	(17-24)	Nodal point n_2	I5	(I8)
21-25	(25-32)	Nodal point n_3	I5	(I8)
26-30	(33-40)	Nodal point n_4	I5	(I8)
31-40	(41-50)	Normal failure stress (Type 9 only)	E10.0	(E10.0)
41-50	(51-60)	Shear failure stress (Type 9 only)	E10.0	(E10.0)
51-60	(61-70)	Coulomb friction scale factor	E10.0	(E10.0)
61-70	(71-80)	Viscous friction scale factor	E10.0	(E10.0)

(2I5,4E10.0), or (2I8,4E10.0) for LARGE option types

Define the discrete slave nodes for node to surface interfaces types.

Columns		Quantity	Format	
1-5	(1-8)	Slave number EQ.0: increment slave number by 1	I5	(I8)
6-10	(9-16)	Nodal point number	I5	(I8)
11-20	(17-26)	S_n, normal force at failure (Type 8 only)	E10.0	(E10.0)
21-30	(27-36)	S_s, shear force at failure (Type 8 only)	E10.0	(E10.0)
31-40	(37-46)	*n*, exponent for normal force (Type 8 only)	E10.0	(E10.0)
41-50	(47-56)	*m*, exponent for shear force (Type 8 only)	E10.0	(E10.0)

Omitted data are automatically generated by incrementing the nodal point numbers by:

$$\frac{\left(n_i - n_j\right)}{\left(sn_i - sn_j\right)}$$

where sn_i, sn_j are the slave numbers on two successive cards and n_i and n_j are their corresponding node numbers.

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. For tiebreaking interfaces (type 9) the above failure criterion is used with m=n=2.

The friction model is similar to the model for an elastic-perfectly plastic material. Corresponding to Young's modulus and the strain rate are the surface stiffness and the relative velocity between a node and the surface segment it contacts. The product of the velocity dependent coefficient of friction and the normal force is equivalent to the yield stress. An exponential function is used to smooth the transition between the static and kinetic coefficients of friction.

$$\mu = \mu_k + (\mu_s - \mu_k)e^{-d_v V_{relative}}$$

Buckling analysis requires the single surface contact algorithm to model a surface collapsing onto itself. Simply making the slave segments the same as the master segments will not work. The single surface algorithm is significantly more expensive to use than master-slave contact algorithms. Its use should be restricted to those situations where it is absolutely required.

For solid elements and thick shell elements nodal numbering can be either clockwise or counterclockwise, but things are not so simple for thin shell elements. For shell elements the normal vectors should be consistently oriented or the flag should be set on Control Card 15, col. 45, to invoke the automatic reorientation. For surface to surface contact using shell elements the normal vectors in each surface must be consistently oriented and must point towards the opposite contact surface. This can also be done automatically by LS-DYNA if there is a small separation between the surfaces, which is always the case if the shell thickness is considered.

Cards NUMSI+NSS+1,...,NUMSI+NSS+NMS
(6I5), or (5I8) for LARGE option
(master segment cards)
(for interface types 1-3, 5-9, and 17)

Columns		Quantity	Format	
1-5	(1-8)	Master segment number	I5	(I8)
6-10	(-)	Increment k	I5	(omit)
11-15	(9-16)	Nodal point n_1	I5	(I8)
16-20	(17-24)	Nodal point n_2	I5	(I8)
21-25	(25-32)	Nodal point n_3	I5	(I8)
26-30	(33-40)	Nodal point n_4	I5	(I8)
51-60	(61-70)	Coulomb friction scale factor	E10.0	(E10.0)
61-70	(71-80)	Viscous friction scale factor	E10.0	(E10.0)

Slave and master segment cards are assumed to be in sequence though the particular number assigned to a master segment is arbitrary. Omitted data are automatically generated with respect to the first card prior to the omitted data as

$$n_j^{i+1} = n_j^i + k$$

The generation parameter k is taken from the first card. Nodal points n_1 - n_4 define the corner nodes of the segments as shown in Figure 32.2. Triangular segments are defined by repeating the third node.

For solid elements and thick shell elements nodal numbering can be either clockwise or counterclockwise, but things are not so simple for shell elements. For shell elements the normal vectors should be consistently oriented or the flag should be set on Control Card 15 to invoke the automatic reorientation. For surface to surface contact using shell elements the normal vectors in each surface must be consistently oriented and must point towards the opposite contact surface. This can also be done automatically by LS-DYNA3D if there is a small separation between the surfaces, which is always the case if the shell thickness is considered. FOR THE AUTOMATED INPUT OPTIONS HAVING A SMALL OFFSET BETWEEN THE SHELL CONTACT SURFACES IS IMPORTANT.

Every slave and master segment in the contacting surfaces must be defined. No ordering is assumed or expected.

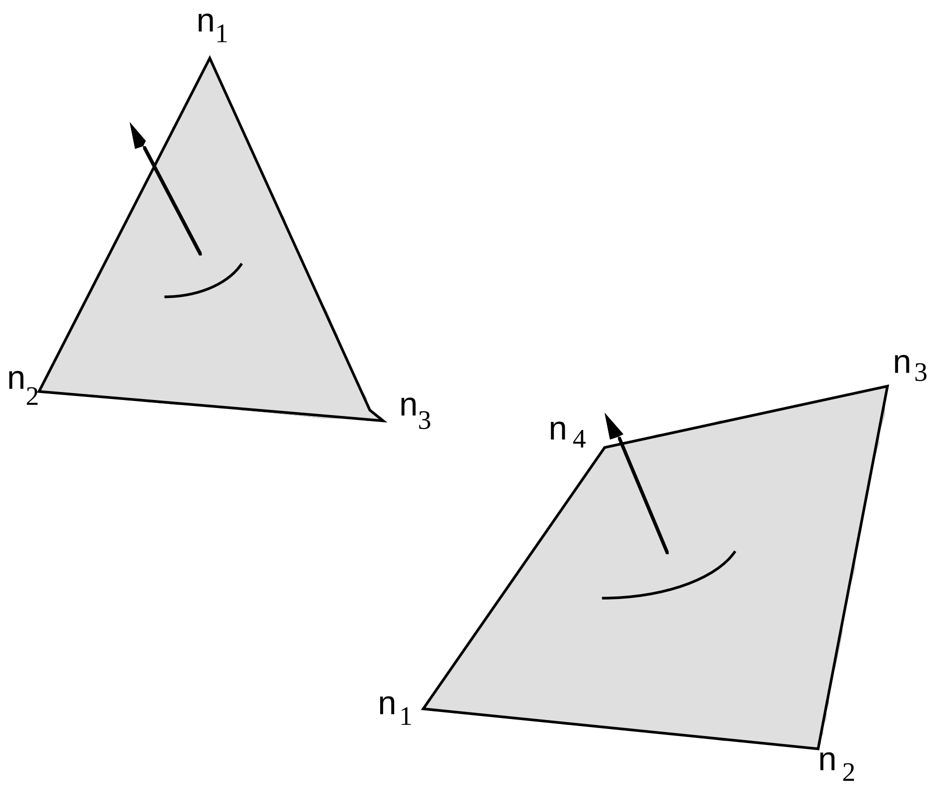


Figure 32.2. Numbering of slave and master segments. The ordering of the numbers determines the outward normal.

33. Tie-Breaking Shell Definitions

For each tiebreak shell slideline, NTBSL (see Control Card 4, columns 16-20), define the following control card.

Cards 1,2,...,NTBSL
(2I5), or (2I8) for LARGE option

Columns		Quantity	Format
1-5	(1-8)	Number of slave nodes, *NSN*	I5 (I8)
6-10	(9-16)	Number of master nodes, *NMN* LT.0: interface number |*NMN*| is used from a database generated in a previous calculation via Section 11 input.	I5 (I8)

Define the following card sets for each tie-breaking shell slideline.

Card 1 (E10.0)

Columns	Quantity	Format
1-10	Default plastic failure strain	E10.0

Cards 2,3,4,...,NSN+1
(2I5,E10.0), or (2I8,E10.0) for LARGE option
(Slave Nodes)

Columns		Quantity	Format
1-5	(1-8)	Slave number EQ.0: the preceding slave number is incremented by 1	I5 (I8)
6-10	(9-16)	Nodal point number	I5 (I8)
11-20	(17-26)	Plastic strain at failure	E10.0 (E10.0)

Tie-Breaking Shell Definitions

Omitted data are automatically generated by incrementing the nodal point numbers by

$$\frac{(n_i - n_j)}{(sn_i - sn_j)}$$

where sn_i, sn_j are the slave numbers on two successive cards and n_i and n_j are their corresponding node numbers. Care must be taken in automatic generation with arbitrary numbering.

Cards NSN+2, NSN+3,...,NSN+NMN+1
(2I5), or (2I8) for LARGE option
(Master nodes)

Columns		Quantity	Format
1-5	(1-8)	Master number EQ.0: the preceding master number is incremented by 1	I5 (I8)
6-10	(9-16)	Nodal point number	I5 (I8)

Omitted data is generated as described above. The master nodes must be given in the order in which they appear as one moves along the edge of the surface. Tie-breaking slidelines may not cross.

Tie-breaking shell slidelines 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 tied slideline is released at that node. The default plastic strain at failure is defined for the entire tie-breaking shell slideline, but can be overridden on the slave node cards to define a unique failure plastic strain for each node.

Tie-breaking Shell Slidelines may be used ot 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, nods along the slideline will begin to separate. As this effect propagates, the tied slideline will appear to "unzip," thus simulating failure of the connection.

34. Tied Node Sets with Failure

Cards 1,2,3,...,NTNWF (3I5)

Define NTNWF (see Control Card 4, columns 21-25) card sets.

Columns	Quantity	Format
1-5	Number of tied node constraints, NTNC	I5
6-10	Number of nodes tied at each constraint (maximum)	I5
11-15	Element type in group EQ.0: shell elements EQ.1: solid elements	I5

For each group of tied nodes (after definition of groups):

Cards NTNWF+1 (E10.0)

Columns	Quantity	Format
1-10	Default plastic strain for failure	E10.0

Card NTNWF+2,....,NTNWF+NTNC
(E10.0,12I5/(14I5)), or (E10.0,6x 6I8/(10I8)) for LARGE option

Columns		Quantity	Format	
1-10	(1-10)	Plastic strain required for failure	E10.0	(E10.0)
11-70	(-)	List up to 12 nodes tied together	12I5	
***		**for LARGE option**	***	
	(17-64)	List up to 6 nodes tied together	(6I8)	

Use additional cards as necessary continuing with a format of 14I5 (or 10I8 for the LARGE option) on subsequent cards.

Tied Node Sets

This section applies only to deformable plastic three and four noded shell elements and four to eight noded solid elements. The specified nodes are tied together until the average volume weighted plastic strain at the nodes exceeds the specified value. When the failure value is reached for a group of constrained nodes, the nodes of the elements whose plastic strain exceeds the failure valure are released to simulate the formation of a crack.

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 Tied Nodes Set with Failure option ties the nodal points together. As plastic strain develops and exceeds the failure strain, cracks will form and propagate through the mesh.

Entire regions of individual shell elements may be tied together, unlike the tie-breaking shell slidelines. Tie breaking shell slidelines are recommend when the location of failure is known, e.g., as in the plastic covers which hid airbags in automotive structures.

When using surfaces of shell elements defined using the Tied Nodes Set with Failure option in contact, it is best to defined each node in the surface as a slave node with the type 5 contact options. If this is not possible, the automatic contact algorithms such as a 3, a5, a10, and 13 all of which include thickness offsets are recommended.

35. Nodes Moved Via Section 11 Interface File
Cards 1,2,3,...,NTNPFL (2I5), or (2I8) for the LARGE option

Define NTNPFL (see Control Card 4, columns 26-30) card sets.

Columns		Quantity	Format
1-5	(1-8)	Nodal point number of tied node	I5 (I8)
6-10	(9-16)	Interface number in database which defines the motion	I5 (I8)

This capability applies to springs and beams only.

36. Rigid Body Merge Cards

(2I5) or (2I10) for MLARG option

Define NRBC (see Control Card 5, columns 6-10) rigid body merge cards in this section.

Columns		Quantity	Format
1-5	(1-10)	Master rigid body material number	I5 (I10)
6-10	(11-20)	Slave rigid body material number	I5 (I10)

The slave rigid body is merged to the master rigid body and the inertial properties which are computed by LS-DYNA are based on the combination of the master rigid body and all the rigid bodies which are slaved to it. 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 above.

It is also possible to merge rigid bodies that are completely separated and share no common nodal points.

All actions valid for the master rigid body, e.g., constraints, given velocity, are now also valid for the newly-created rigid body.

37. Nodal Rigid Body Constraint and Welded Node Sets

Input NUMRBS (Control Card 5, columns 1-5) rigid body nodal constraint and welded node sets.

When possible we recommend the use of the spotwelds in Section 27 for modeling two node spotwelds and rivets. The treatment here allows the nodes in the nodal rigid body to be coincident since an optional local coordinate system is defined to orient the spotweld relative to the structures they connect. In Section 27 a local system is defined based on the nodal separation so coincident nodes are not permitted. Unlike the nodal constraint sets of Section 27 which permit only translational motion (the rivets and spotwelds are the exceptions), here the equations of rigid body dynamics are used to update the motion of the nodes and therefore rotation of the nodal sets is admissible. Mass properties are determined from the nodal masses and coordinates. Inertial properties may be defined in the "Rigid Body Inertial Properties" section which follows.

Card 1 (I15,3I5,E10.0, 2I5)

Columns	Quantity	Format
1-15	Number of nodes in the rigid body, NRB	I15
16-20	Optional local coordinate system ID (see Section 14) for output or weld failure.	I5
21-25	Weld failure option, IWELD. A local coordinate system must be defined for fillet and butt welds in Columns 16-20 for brittle type failures to orient the weld relative to the rigid body. *A local system is needed for spot welds if the spot welded nodes are coincident.* This local system is incrementally updated in time to account for rotations. If failure is due to plastic straining only, the local coordinate system is not needed. EQ.0: no failure is permitted EQ.1: spot weld, Figure 37.1. EQ.2: fillet weld, Figure 37.2. EQ.3: butt weld, Figure 37.3. EQ.4: cross fillet weld, Figure 37.4. EQ.5: general weld, Figure 37.5.	I5

Columns	Quantity	Format
26-30	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.	I5
31-40	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	E10.0
41-45	NFW, number of individual nodal pairs in the cross fillet and general welds.	I5
46-50	Print option in file RBDOUT. EQ.0: default from Control Card is used EQ.1: data is printed EQ.2: data is not printed	I5

Spotweld Weld Failure IWELD=1
Optional Card (6E10.0)

Columns	Quantity	Format
1-10	Failure time for constraint set, t_f. (default=1.E+20)	E10.0
11-20	ε^p_{fail}, effective plastic strain at failure	E10.0
21-30	S_n, normal force at failure	E10.0
31-40	S_s, shear force at failure	E10.0
41-50	n, exponent for normal force	E10.0
51-60	m, exponent for shear force	E10.0

Failures can include both the plastic and brittle failures. These can be used either independently or together. Failure occurs when either criteria is met.

Spotweld 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.

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 37.1 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.

Fillet Weld Failure IWELD=2
Optional Card (8E10.0)

Columns	Quantity	Format
1-10	Failure time for constraint set, t_f. (default=1.E+20)	E10.0
11-20	ε^p_{fail}, effective plastic strain at failure	E10.0
21-30	σ_f, stress at failure	E10.0
31-40	β, failure parameter	E10.0
41-50	*L*, length of fillet weld	E10.0
51-60	*w*, width of flange	E10.0
61-70	*a*, width of fillet weld as shown in Figure 36.2	E10.0
71-80	α, weld angle in degrees	E10.0

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 37.2 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.

Butt Weld Failure IWELD=3
Optional Card (7E10.0)

Columns	Quantity	Format
1-10	Failure time for constraint set, t_f. (default=1.E+20)	E10.0
11-20	ε_{fail}^p, effective plastic strain at failure	E10.0
21-30	σ_f, stress at failure	E10.0
31-40	β, failure parameter	E10.0
41-50	L, length of butt weld	E10.0
51-60	d, thickness of butt weld	E10.0
61-70	L_t, length of butt weld in transverse direction. Define for corner butt welds only.	E10.0

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.

Cross Fillet Weld Failure IWELD=4
Optional Cards

Card 1 for Cross Fillet Weld Failure (8E10.0)

Columns	Quantity	Format
1-10	Failure time for constraint set, t_f. (default=1.E+20)	E10.0
11-20	ε^p_{fail}, effective plastic strain at failure	E10.0
21-30	σ_f, stress at failure	E10.0
31-40	β, failure parameter	E10.0
41-50	L, length of fillet weld	E10.0
51-60	w, width of flange	E10.0
61-70	a, width of fillet weld as shown in Figure 37.2	E10.0
71-80	α, weld angle in degrees	E10.0

Cards 2,...,NFW+1 for Cross Fillet Weld Failure IWELD=4
(3I5), or (2I8,I5) for LARGE option

Columns		Quantity	Format
1-8	(1-8)	Node A in weld pair	I5 (I8)
6-10	(9-16)	Node B in weld pair	I5 (I8)
11-15	(17-21)	Local coordinate system ID	I5

Card 2, 3, ..., (10I5), or (10I8) for LARGE option

Columns		Quantity	Format
1-50	(1-80)	Define nodes, up to ten per card	10I5 (10I8)

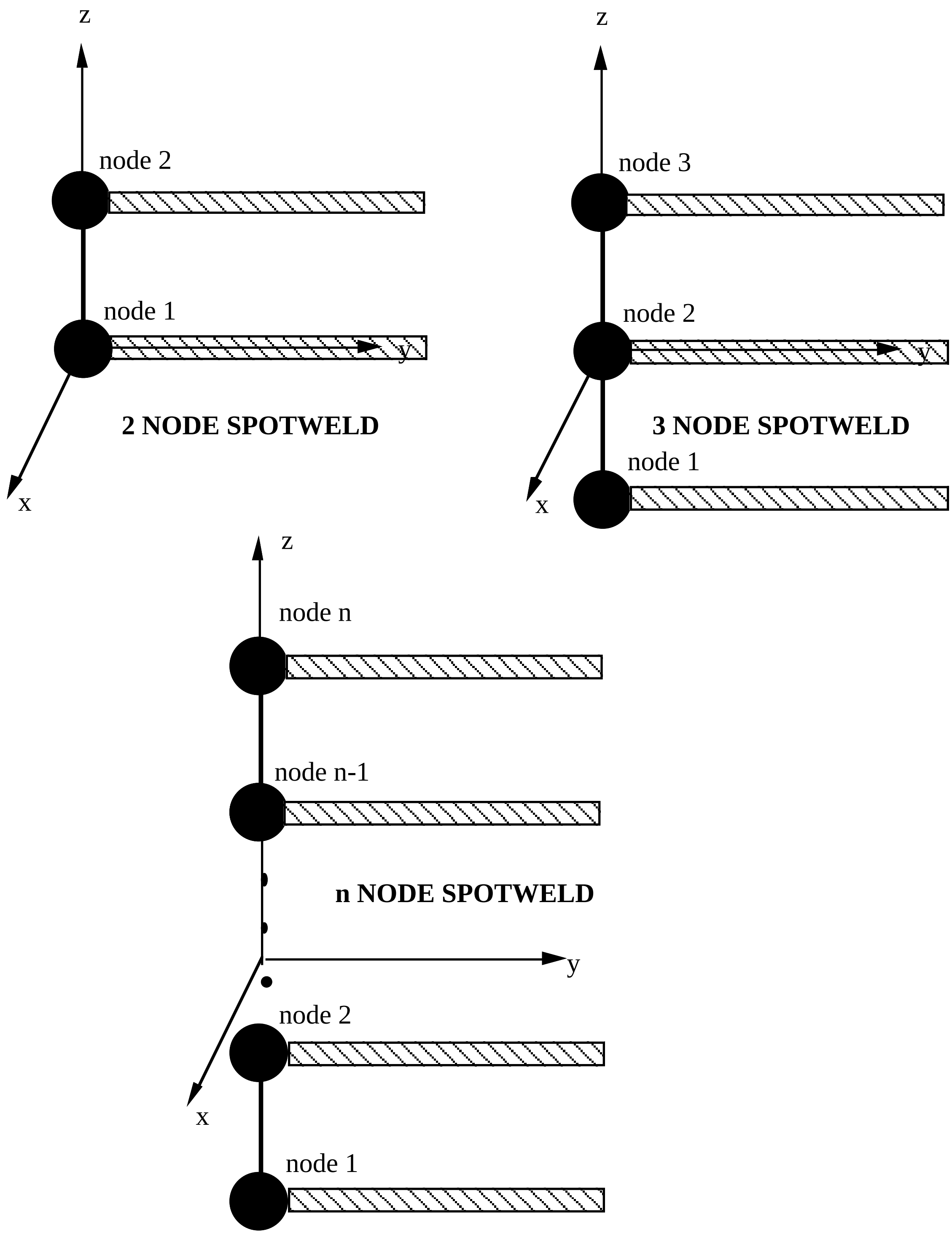


Figure 37.1. Nodal ordering and orientation of the local coordinate system is important for determining spotweld failure.

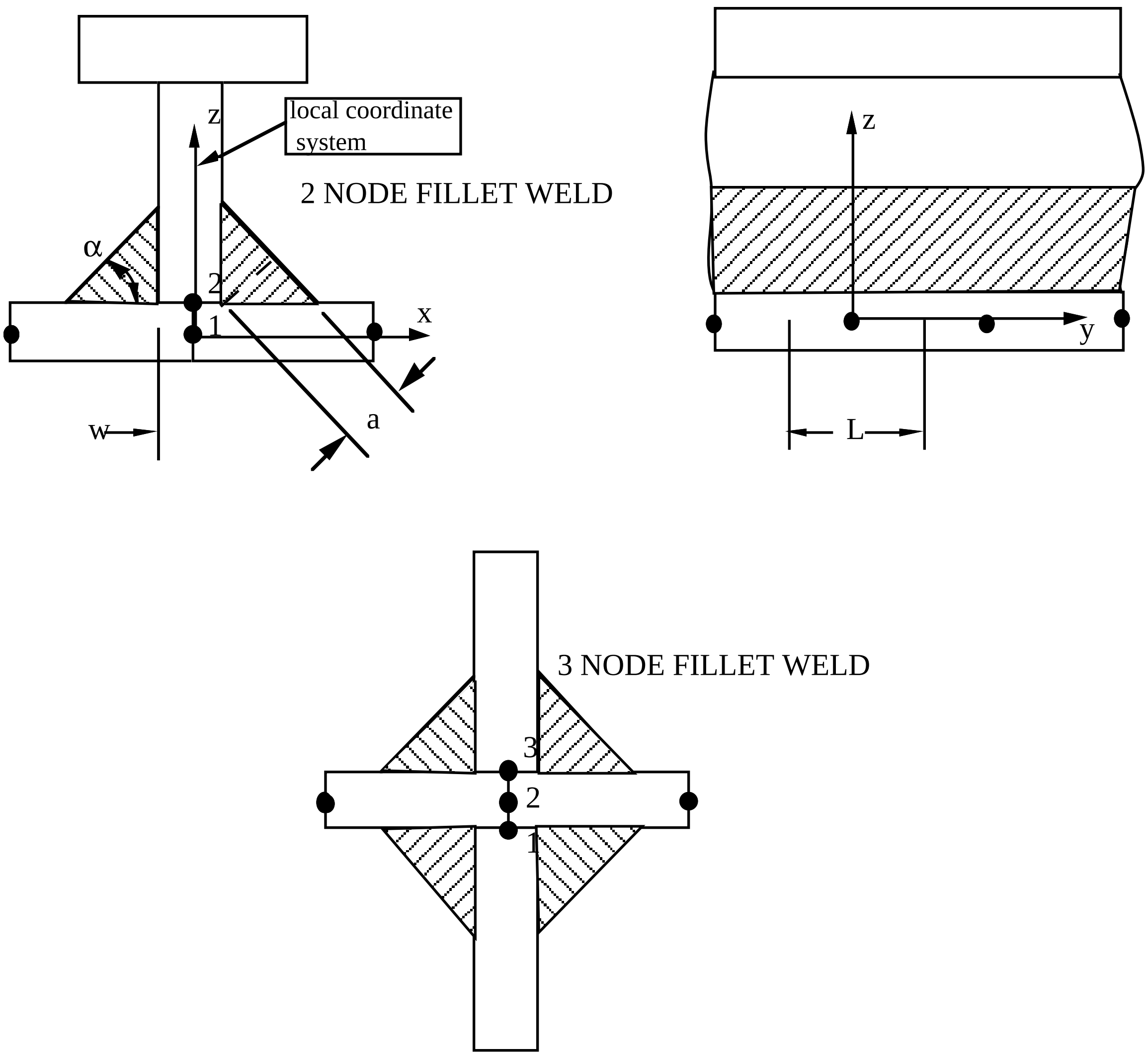


Figure 37.2. Nodal ordering and orientation of the local coordinate system is shown for fillet weld failure.

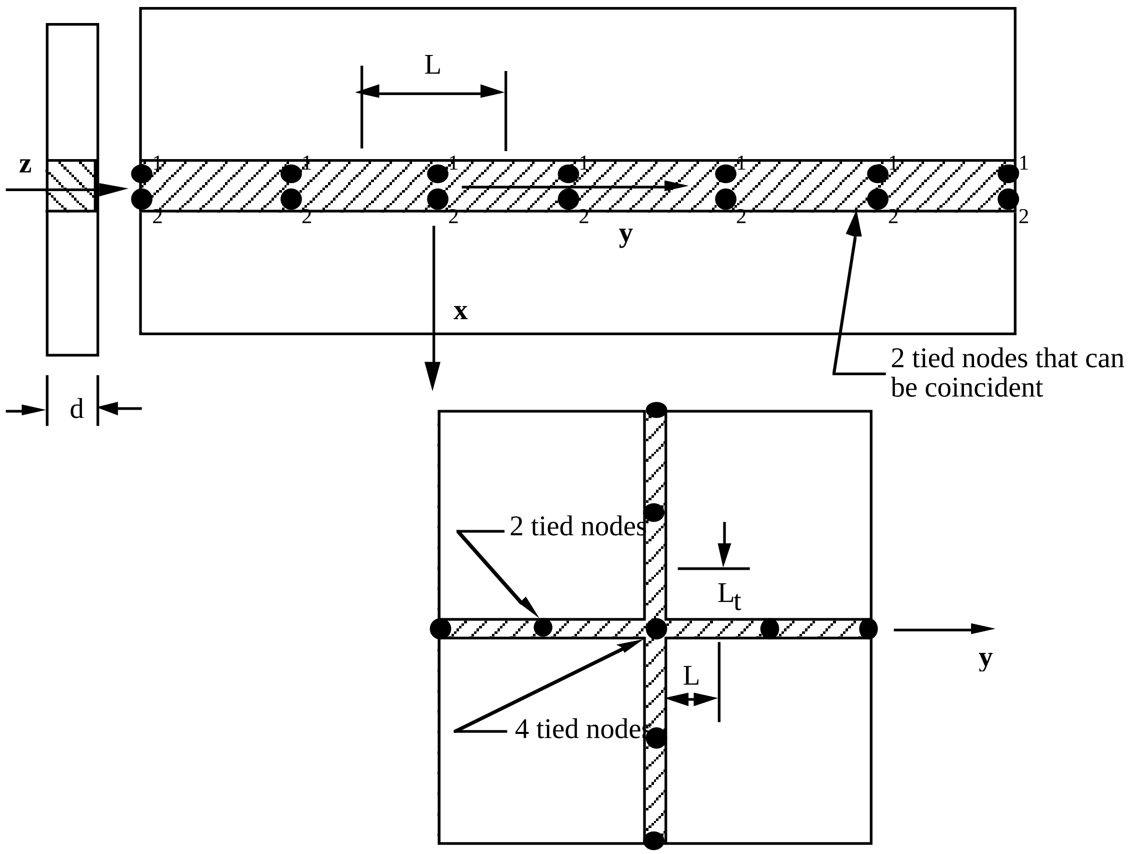


Figure 37.3. Orientation of the local coordinate system and nodal ordering is shown for butt weld failure.

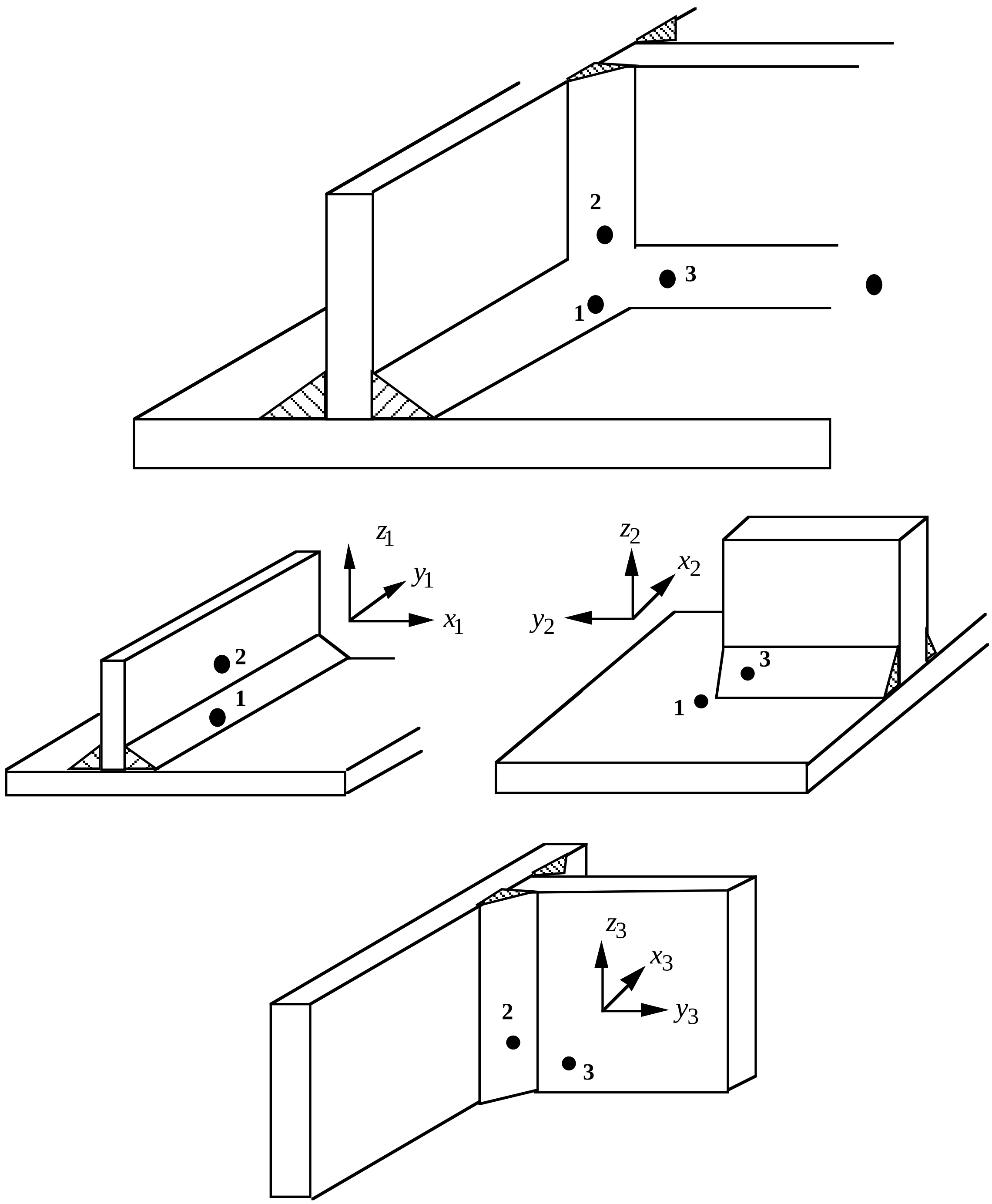


Figure 37.4. A simple cross fillet weld illustrates the required input. Here NFW=3 with nodal pairs (A=2, B=1), (A=3, B=1), and (A=3, B=2). The local coordinate axes are shown. These axes are fixed in the rigid body and are referenced to the local rigid body coordinate system which tracks the rigid body rotation.

General Weld Failure IWELD=5
Optional Card Sets 1,2,...,NFW

Card 1 of 2 for General Weld (4I5), or (2I8,2I5) for LARGE option

Columns		Quantity	Format
1-8	(1-8)	Node A in weld pair	I5 (I8)
6-10	(9-16)	Node B in weld pair	I5 (I8)
11-15	(17-21)	Local coordinate system ID	I5
16-20	(22-26)	Weld pair type EQ.0: fillet weld EQ.1: butt weld	I5

Card 2 of 2 for General Weld (8E10.0)

General welds are a mixture of fillet welds and butt welds as shown in Figure 37.5. Their definition is similar to that for cross fillet welds above.

Columns	Quantity	Format
1-10	Failure time for constraint, t_f, (default=1.E20)	E10.0
11-20	ε^p_{fail}, effective plastic strain at failure	E10.0
21-30	σ_f, stress at failure	E10.0
31-40	β, failure parameter	E10.0
41-50	L, length of fillet/butt weld	E10.0
51-60	w/d, plate thickness for the fillet/butt welds, respectively	E10.0
61-70	a, width of fillet weld (input for fillet weld only)	E10.0
71-80	α, weld angle (input for fillet weld only)	E10.0

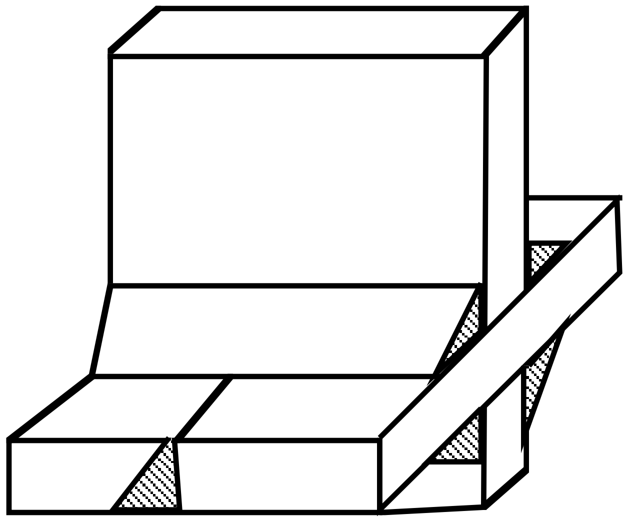


Figure 37.5. A general weld is a mixture of fillet and butt welds.

38. Extra Nodes for Rigid Bodies

Define the number of card sets, NXTRA, specified on Control Card 5, columns 16-20.

Card 1 (2I5) or (2I10) for MLARG option

Columns		Quantity	Format
1-5	(1-10)	Rigid body material number	I5 (I10)
6-10	(11-20)	Number of extra nodes, NEN	I5 (I10)

Cards 2,3,4,...,NEN+1 (10I5), or (10I8) for LARGE option

Columns		Quantity	Format
1-50	(1-80)	Extra nodes, up to ten per card	10I5 (10I8)

Extra nodes for rigid bodies may be placed anywhere, even outside the body. These extra nodes 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.

39. Joint Definition Cards

First define the number of joint definitions, NJT, specified on Control Card 5, columns 11-15. Following the joint definitions, define the NJTS optional stiffness characteristics, i.e., the moment/torsion versus angle behavior.

Joint may be used between two rigid bodies, designated here as rigid bodies A and B.

Card 1 (E10.0,7I5,E10.0,2I5)
(E10.0,I5,6I8,E10.0,I5,I2) for LARGE option

Columns		Quantity	Format	
1-10	(1-10)	Relative penalty stiffness (default is 1.0)	E10.0	(E10.0)
11-15	(11-15)	Joint type, JT EQ.1: spherical EQ.2: revolute EQ.3: cylindrical EQ.4: planar EQ.5: universal EQ.6: translational EQ.7: locking EQ.8: translational motor EQ.9: rotational motor EQ.10: gears EQ.11: rack and pinion EQ.12: constant velocity EQ.13: pulley EQ.14: screw	I5	(I5)
16-20	(16-23)	Node 1, in rigid body A	I5	(I8)
21-25	(24-31)	Node 2, in rigid body B	I5	(I8)
26-30	(32-39)	Node 3, in rigid body A (B if JT=8). Define if JT>1.	I5	(I8)
31-35	(40-47)	Node 4, in rigid body B. Define if JT>1.	I5	(I8)
36-40	(48-55)	Node 5, in rigid body A. Define if JT=6, 7, 9-14.	I5	(I8)
41-45	(56-63)	Node 6, in rigid body B. Define if JT=6, 7, 9-14.	I5	(I8)
46-55	(64-73)	Damping factor (Percent of critical) EQ.0.0: default is set to 1.0 LE.0.01 and GT.0.0: no damping is used.	E10.0	(E10.0)

Columns	Quantity	Format
56-60 (74-78)	Rigid body or accelerometer ID. The force resultants are output in the local system of the rigid body or accelerometer.	I5 (I5)
61-65 (79-80)	Flag for local system type EQ. 0: rigid body EQ. 1: accelerometer	I5 (I2)

Card 2 (E10.0,2I10)

This card is read for joint types 8-11, 13, and 14.

Columns	Quantity	Format
1-10	Specified joint parameter. Joint type: EQ.8: blank EQ.9: blank EQ.10: gear ratio, $\frac{R_2}{R_1}$ EQ.11: distance, h EQ.13: pulley ratio, $\frac{R_2}{R_1}$ EQ.14: helix ratio $\frac{\dot{x}}{\omega}$	E10.0
11-20	If JT=8 or 9: load curve ID or else leave blank.	I10
21-30	If JT=8 or 9: define flag or else leave blank. EQ.0: translational/rotational velocity EQ.1: translational/rotational acceleration EQ.2: translational/rotational displacement	I10

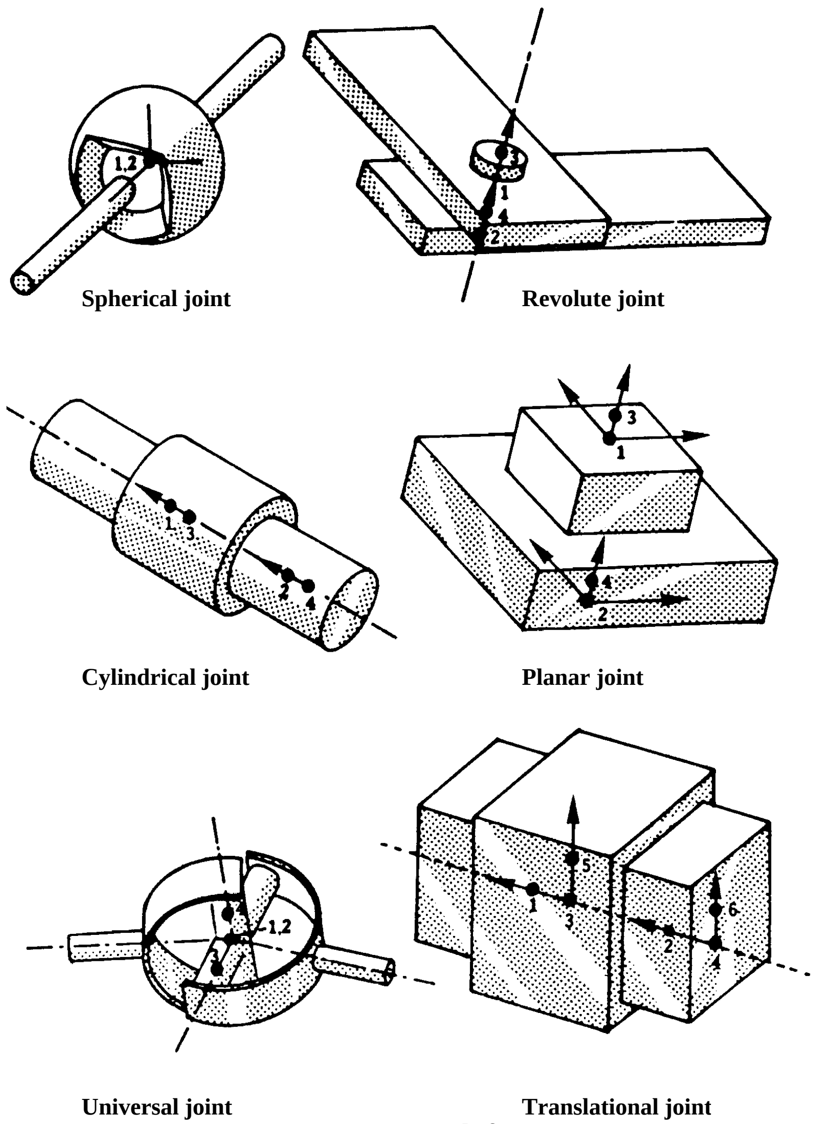
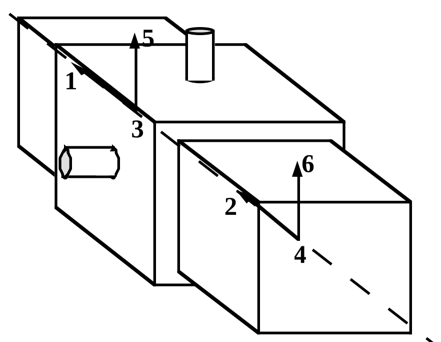


Figure 39.1. Joint definitions 1-6



Locking joint

Figure 39.2. Locking joint.

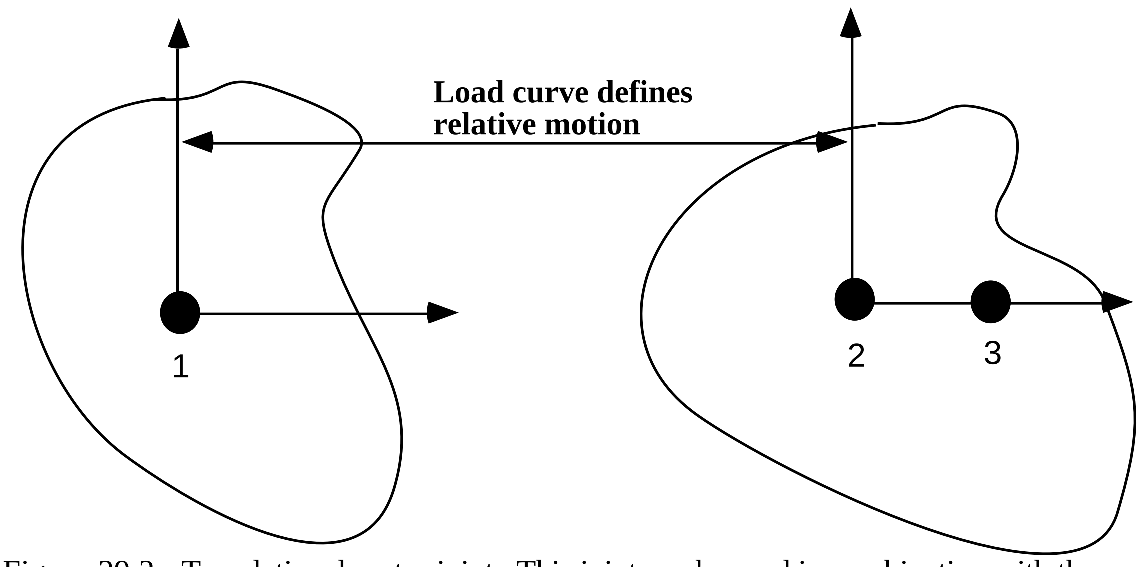


Figure 39.2. Translational motor joint. This joint can be used in combination with the translational or the cylindrical joint.

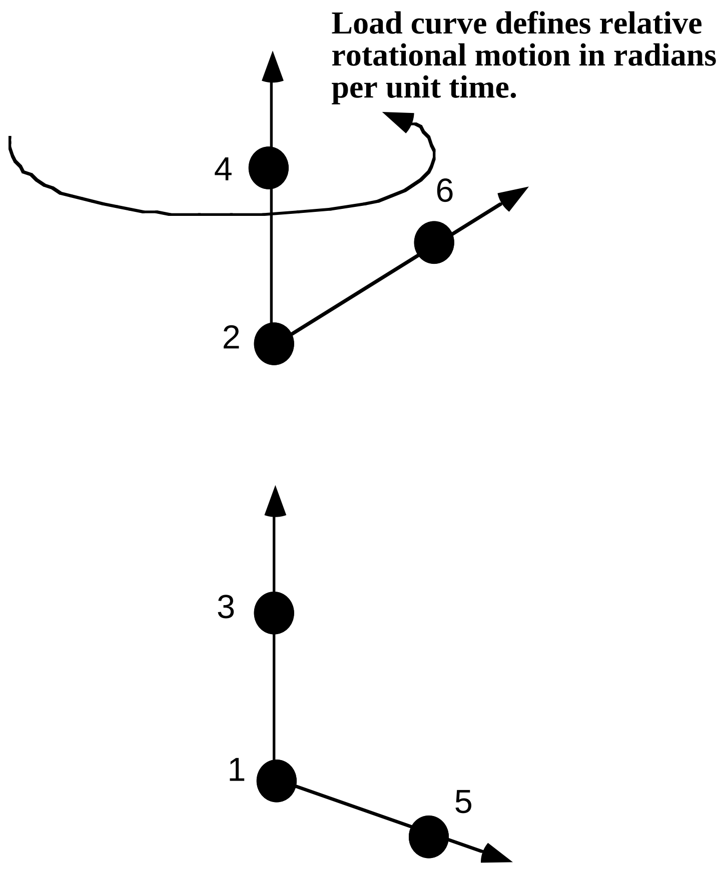


Figure 39.3. Rotational motor joint. This joint can be used in combination with other joints such as the revolute or cylindrical.

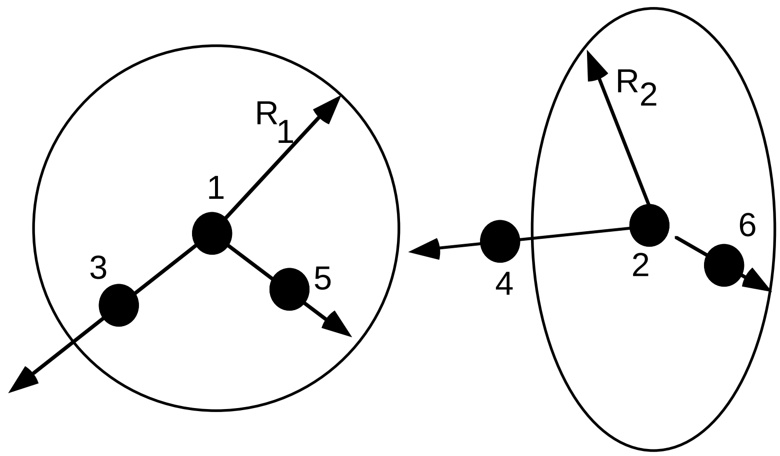


Figure 39.4. Gear joint. Nodal pairs (1,3) and (2,4) define axes that are orthogonal to the gears. Nodal pairs (1,5) and (2,6) define vectors in the plane of the gears. The ratio $\frac{R_2}{R_1}$ is specified.

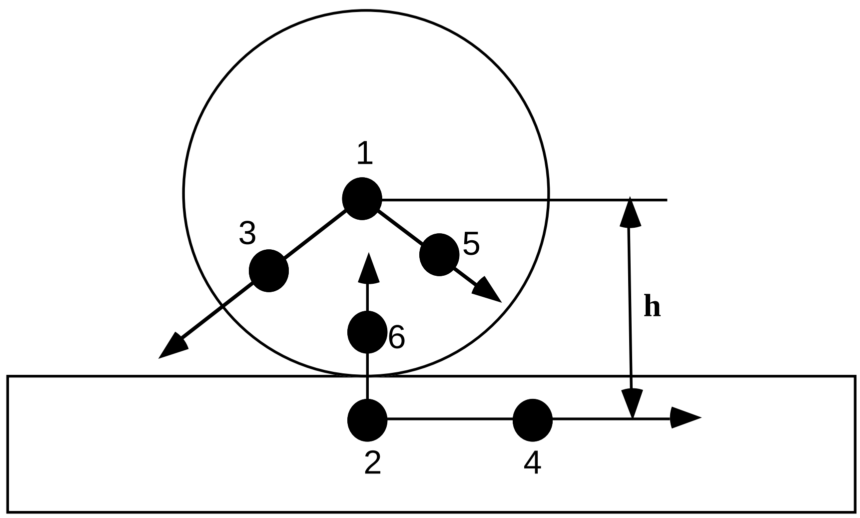


Figure 39.5 Rack and pinion joint. Nodal pair (1,3) defines an axes that is orthogonal to the gear. Nodal pair (1,5) is a vector in the plane of the gear. The value h is specified.

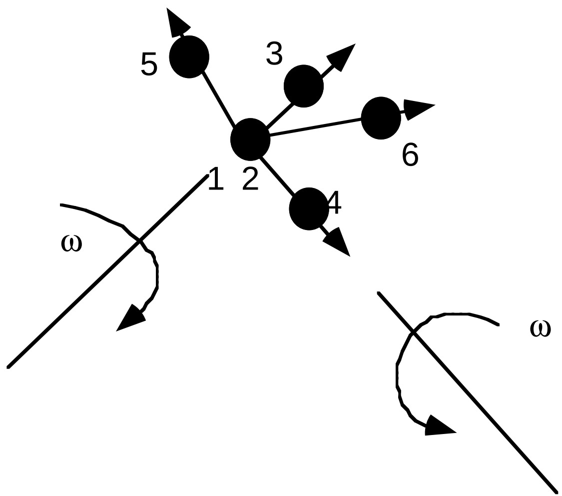


Figure 39.6 Constant velocity joint. Nodal pairs (1,3) and (2,4) define an axes for the constant angular velocity, and nodal pairs (1,5) are orthogonal vectors. Here nodal points 1 and 2 must be coincident.

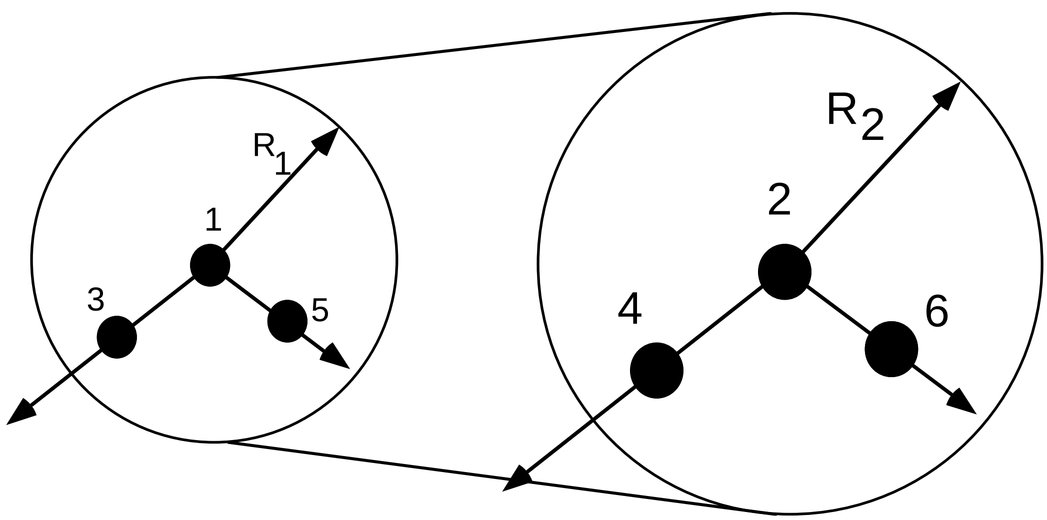


Figure 39.7. Pulley joint. Nodal pairs (1,3) and (2,4) define axes that are orthogonal to the pulleys. Nodal pairs (1,5) and (2,6) define vectors in the plane of the pulleys. The ratio $\frac{R_2}{R_1}$ is specified.

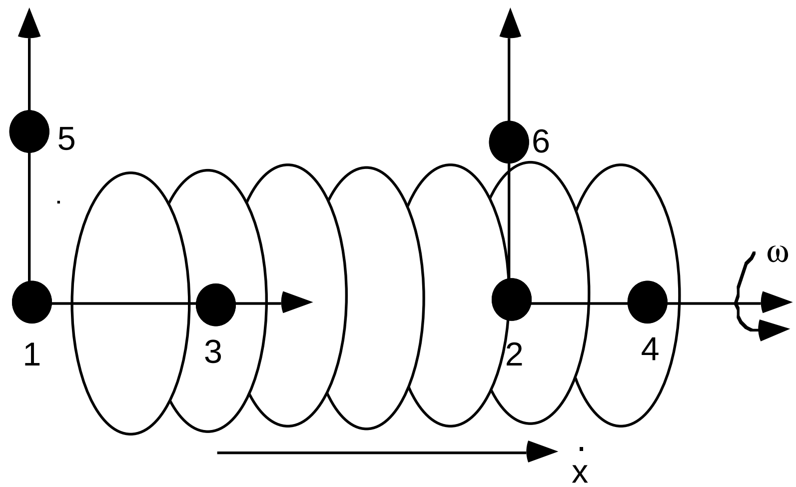


Figure 39.8. Screw joint. Nodal pairs (1,3) and (2,4) lie along the same axis and nodal pairs (1,5) and (2,6) are orthogonal vectors. The helix ratio, $\frac{\dot{x}}{\omega}$, is specified.

The geometry of joints is defined in Figures 39.1 to 39.8. 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 they occur, the recommended way to eliminate these problems is to decrease the time step.

For cylindrical joints, the nodes should be colinear. 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).

With some exceptions including joint types 8-14, the nodal points within the nodal pairs (1,2), (3,4), and (5,6) should coincide in the initial configuration, and the nodal pairs should be as far apart as possible to obtain the best behavior. For the Universal Joint the nodal pair (3,4) do not coincide, but the lines drawn between nodes (1,3) and (2,4) must be perpendicular.

Define NJTS joint stiffness characteristics card sets below. 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.

If the rotation or rotational rate are outside the values defined in the load curves, extrapolated values are used. For stability, care should be taken to insure that extrapolated values will be reasonable.

Card 1 (2I5)

Columns	Quantity	Format
1-5	Joint stiffness ID	I5
6-10	Joint stiffness type EQ.0: generalized stiffness definition EQ.1: flexion-torsion stiffness.	I5

JOINT STIFFNESS TYPE=0, the following cards apply.

Card 2 (4I5) or (2I10,2I5) for MLARG option

Columns		Quantity	Format
1-5	(1-10)	Part ID for rigid body A	I5 (I10)
6-10	(11-20)	Part ID for rigid body B	I5 (I10)
11-15	(21-25)	Coordinate ID for rigid body A	I5 (I5)
16-20	(26-30)	Coordinate ID for rigid body B. If zero, the coordinate ID for rigid body A is used.	I5 (I5)

If the initial position of the local coordinate axes do not coincide, the angles, ϕ, θ, and ψ, are initialized and torques will develop instantaneously based on the defined load curves.

Card 3 (2I5)

Columns	Quantity	Format
1-5	Load curve ID for ϕ–moment versus rotation in radians. If zero, the applied moment is set to 0.0.	I5
6-10	Load curve ID for θ–moment versus rotation in radians. If zero, the applied moment is set to 0.0.	I5
11-15	Load curve ID for ψ–moment versus rotation in radians. If zero, the applied moment is set to 0.0.	I5
16-20	Load curve ID for ϕ–damping moment versus rate of rotation in radians per unit time. If zero, damping is not considered.	I5
21-25	Load curve ID for θ–damping moment versus rate of rotation in radians per unit time. If zero, damping is not considered.	I5
26-30	Load curve ID for ψ–damping torque versus rate of rotation in radians per unit time. If zero, damping is not considered.	I5

Card 4 (6E10.0)

The frictional behavior is elastic-plastic with linear loading, unloading and reloading. A load curve ID may be defined if the elastic-perfectly plastic treatment is inadequate.

Columns	Quantity	Format
1-10	Elastic stiffness in radians for friction and stop angles for ϕ rotation. If zero, friction and stop angles are inactive for ϕ rotation.	E10.0
11-20	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 39.10.	E10.0
21-30	Elastic stiffness in radians for friction and stop angles for θ rotation. If zero, friction and stop angles are inactive for θ rotation.	E10.0
31-40	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 39.10.	E10.0
41-50	Elastic stiffness in radians for friction and stop angles for ψ rotation. If zero, friction and stop angles are inactive for ψ rotation..	E10.0
51-60	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 39.10.	E10.0

Card 5 (6E10.0)

Columns	Quantity	Format
1-10	Stop angle in degrees for negative ϕ rotation. Ignored if zero.	E10.0
11-20	Stop angle in degrees for positive ϕ rotation. Ignored if zero.	E10.0
21-30	Stop angle in degrees for negative θ rotation. Ignored if zero.	E10.0
31-40	Stop angle in degrees for positive θ rotation. Ignored if zero.	E10.0
41-50	Stop angle in degrees for negative ψ rotation. Ignored if zero.	E10.0
51-60	Stop angle in degrees for positive ψ rotation. Ignored if zero.	E10.0

After the stop angles are reached the torques increase linearly to resist further angular motion using the stiffness values on Card 4. If the stiffness values are 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 added together.

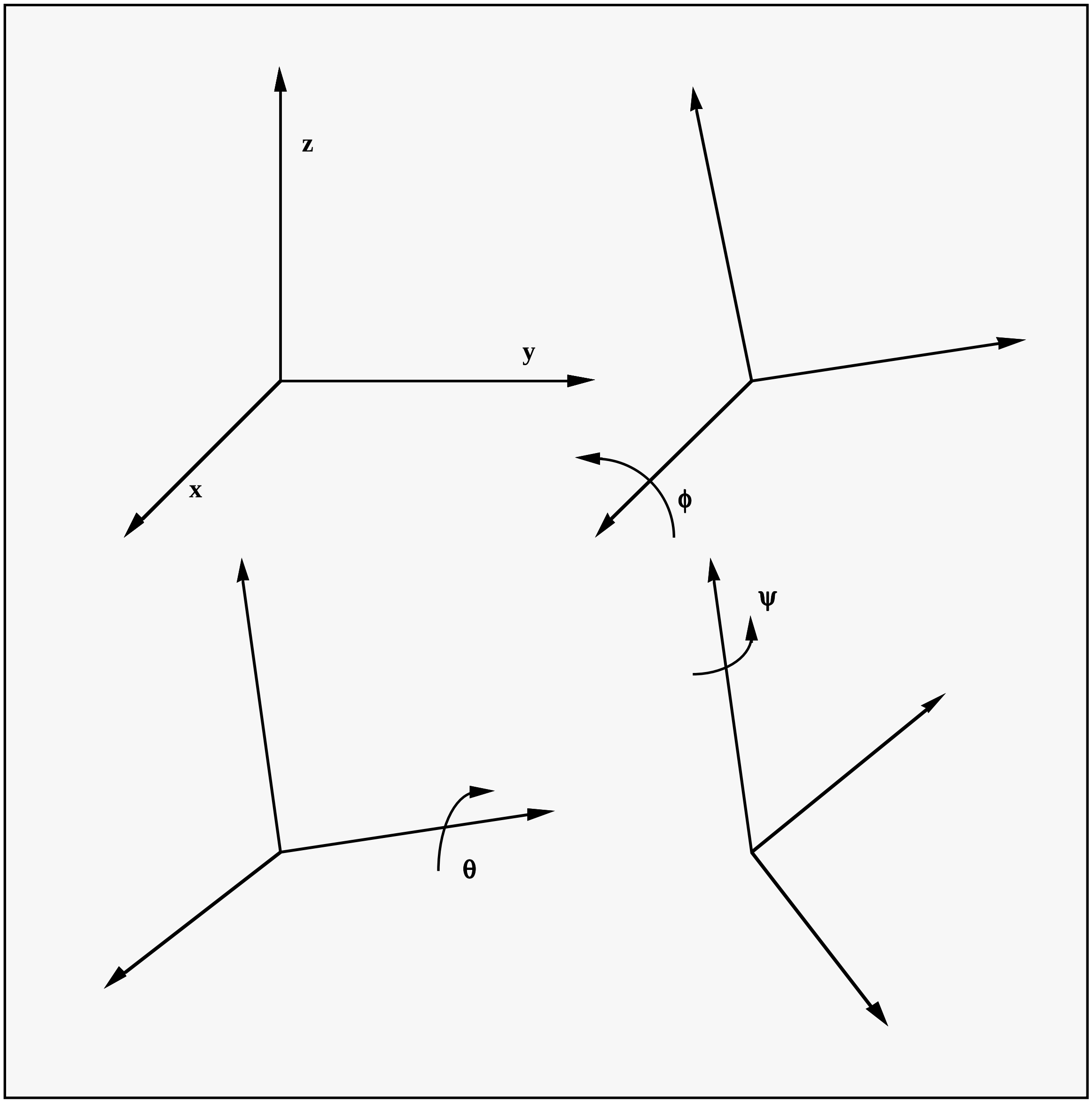


Figure 39.9. Definition of angles for the generalized joint stiffness. The magnitude of the angular rotations are limited by the stop angles defined on Card 5.

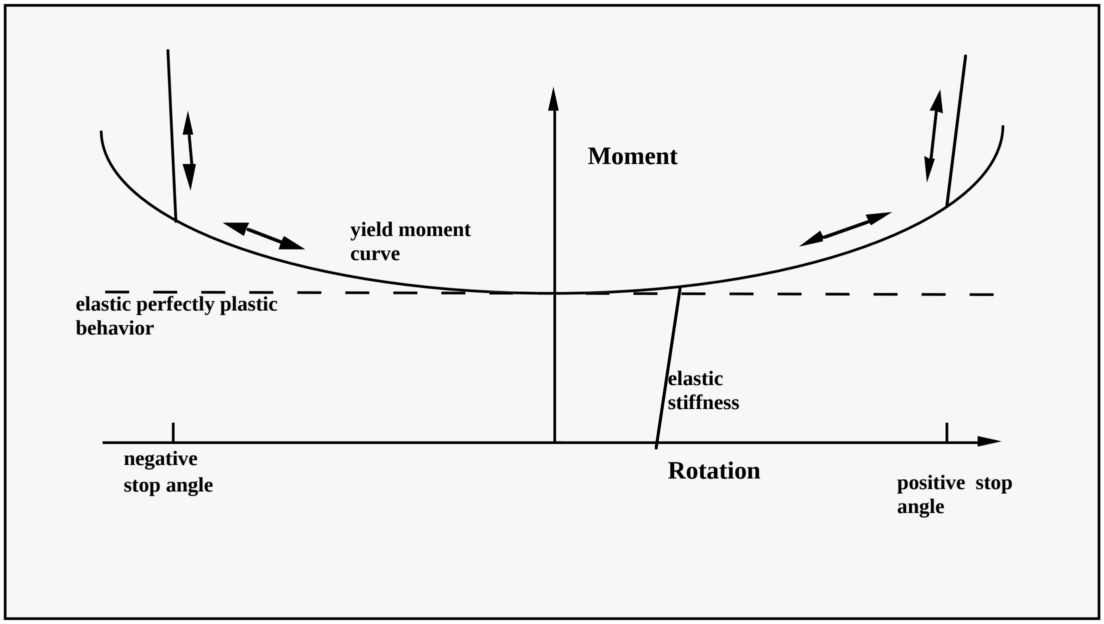


Figure 39.10. Frictional behavior is modeled by a plasticity model. Elastic behavior is obtained once the stop angles are reached.

JOINT STIFFNESS TYPE=1, The following cards apply.

Card 2 (4I5) or (2I10,2I5) for MLARG option

Columns		Quantity	Format
1-5	(1-10)	Part ID for rigid body A	I5 (I10)
6-10	(11-20)	Part ID for rigid body B	I5 (I10)
11-15	(21-25)	Coordinate ID for rigid body A	I5 (I5)
16-20	(26-30)	Coordinate ID for rigid body B. If zero, the coordinate ID for rigid body A is used.	I5 (I5)

If the initial position of the local coordinate axes do not coincide, the angles, α and γ, are initialized and torques will develop instantaneously based on the defined load curves. The angle β is also initialized but no torque will develop about the local axis on which β is measured. Rather, β will be measured relative to the computed offset.

Card 3 (2I5)

Columns	Quantity	Format
1-5	Load curve ID for α–moment versus rotation in radians. If zero, the applied moment is set to 0.0.	I5
6-10	Load curve ID for γ–scale factor. If zero the scale factor defaults to 1.0.	I5
11-15	Load curve ID for β–torsion versus twist in radians. If zero, the applied twist is set to zero.	I5
16-20	Load curve ID for α–damping moment versus rate of rotation in radians per unit time. If zero, damping is not considered.	I5
21-25	Load curve ID for γ–damping scale factor versus rate of rotation in radians per unit time. If zero, the scale factor defaults to one.	I5
26-30	Load curve ID for β–damping torque versus rate of twist in radians per unit time. If zero damping is not considered.	I5

Card 4 (4E10.0)

Columns	Quantity	Format
1-10	Elastic stiffness per radian for friction and stop angles for α rotation. If zero, friction and stop angles are inactive for α rotation..	E10.0
11-20	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.	E10.0
21-30	Elastic stiffness per radian for friction and stop angles for β twist. If zero, friction and stop angles are inactive for β twist.	E10.0
31-40	Frictional moment limiting value for β twist. If zero, friction is inactive for β 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 β rotation.	E10.0

Card 5 (6E10.0)

Columns	Quantity	Format
1-10	Stop angle in degrees for α rotation. Ignored if zero.	E10.0
11-20	Stop angle in degrees for negative β rotation. Ignored if zero.	E10.0
21-30	Stop angle in degrees for positive β rotation. Ignored if zero.	E10.0

After the stop angles are reached the torques increase linearly to resist further angular motion using the stiffness values on Card 4. 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.

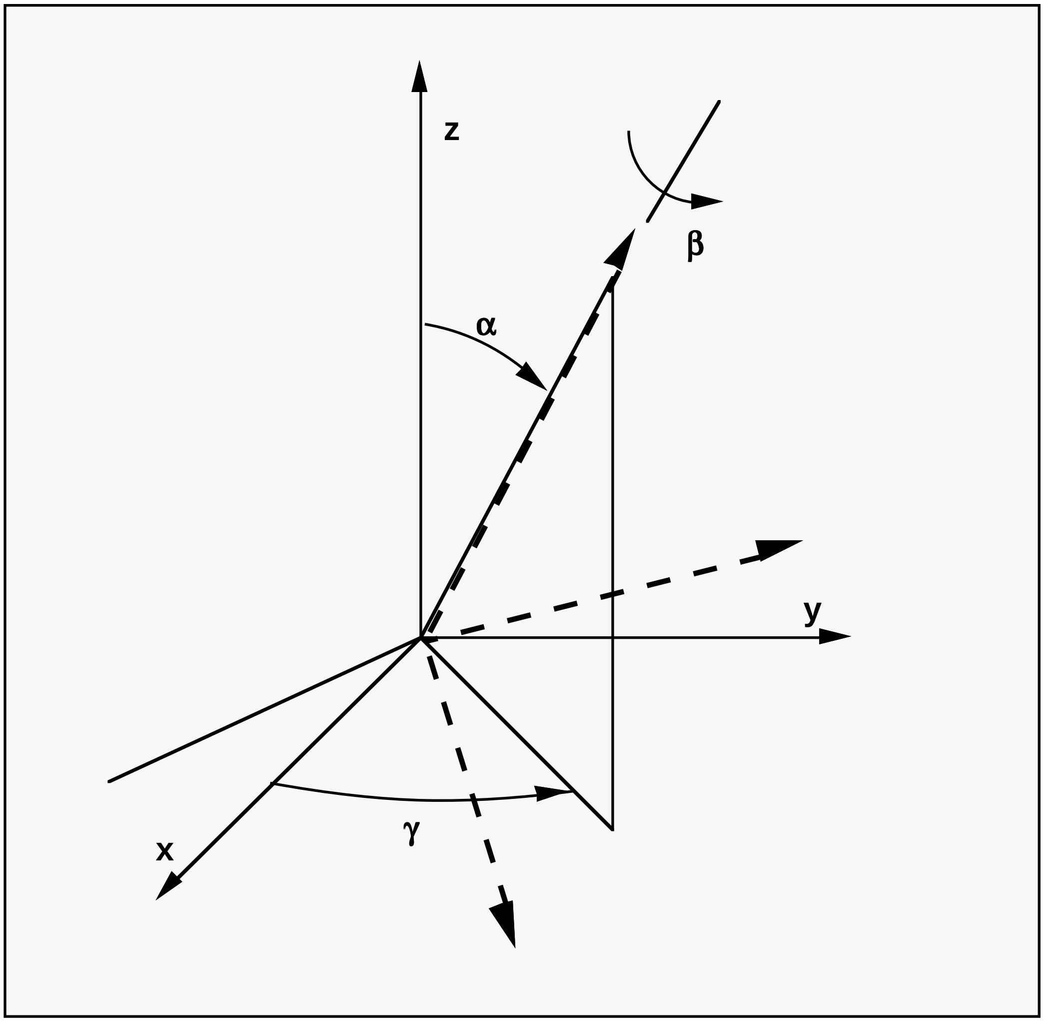


Figure 39.11 Flexion-torsion joint angles.

40. Base Acceleration in X-direction
(I5,E10.0,I5)

Skip this card if columns 1-5 of Control Card 10 (NTHPX) are blank.

Columns	Quantity	Format
1-5	Load curve number	I5
6-15	Scale factor on x-acceleration EQ.0.0: default set to "1.0"	E10.0
16-20	Load curve number for dynamic relaxation phase (optional)	I5

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.

This option is 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.

Base Acceleration in X-direction

41. Base Acceleration in Y-direction
(I5,E10.0,I5)

Skip this card if columns 6-10 of Control Card 10 (NTHPY) are blank.

Columns	Quantity	Format
1-5	Load curve number	I5
6-15	Scale factor on y-acceleration EQ.0.0: default set to “1.0”	E10.0
16-20	Load curve number for dynamic relaxation phase (optional)	I5

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 z-x plane and extended in the positive y-direction, then a positive y-direction base acceleration would tend to shorten the cylinder, i.e., create forces acting in the negative y-direction.

This option is 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.

Base Acceleration in Y-direction

42. Base Acceleration in Z-direction
(I5,E10.0,I5)

Skip this card if columns 11-15 of Control Card 10 (NTHPZ) are blank.

Columns	Quantity	Format
1-5	Load curve number	I5
6-15	Scale factor on z-acceleration EQ.0.0: default set to “1.0”	E10.0
16-20	Load curve number for dynamic relaxation phase (optional)	I5

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 x-y plane and extended in the positive z-direction, then a positive z-direction base acceleration would tend to shorten the cylinder, i.e., create forces acting in the negative z-direction.

This option is 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.

43. Angular Velocity About X-Axis
(I5,E10.0,I5,3E10.0)

Skip this card if columns 16-20 of Control Card 10 (NTHSX) are blank.

Columns	Quantity	Format
1-5	Load curve number	I5
6-15	Scale factor on angular velocity EQ.0.0: default set to "1.0"	E10.0
16-20	Load curve number for dynamic relaxation phase (optional)	I5
21-30	x-center of rotation	E10.0
31-40	y-center of rotation	E10.0
41-50	z-center of rotation	E10.0

Body force loads due to the angular velocity about an axis parallel to the global x-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.

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.

This feature is 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.

44. Angular Velocity About Y-Axis
(I5,E10.0,I5,3E10.0)

Skip this card if columns 21-25 of Control Card 10 (NTHSY) are blank.

Columns	Quantity	Format
1-5	Load curve number	I5
6-15	Scale factor on angular velocity EQ.0.0: default set to “1.0”	E10.0
16-20	Load curve number for dynamic relaxation phase (optional)	I5
21-30	x-center of rotation	E10.0
31-40	y-center of rotation	E10.0
41-50	z-center of rotation	E10.0

Body force loads due to the angular velocity about an axis parallel to the global y-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.

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.

This feature is 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.

45. Angular Velocity About Z-Axis
(I5,E10.0,I5,3E10.0)

Skip this card if columns 26-30 of Control Card 10 (NTHSZ) are blank.

Columns	Quantity	Format
1-5	Load curve number	I5
6-15	Scale factor on angular velocity (default = 1.0)	E10.0
16-20	Load curve number for dynamic relaxation phase (optional)	I5
21-30	x-center of rotation	E10.0
31-40	y-center of rotation	E10.0
41-50	z-center of rotation	E10.0

Body force loads due to the angular velocity about an axis parallel to the global z-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.

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.

This feature is 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.

46. Body Force Material Subset for Sections 39-44
(8I10)

Skip this section if columns 31-35 of Control Card 10 (NMTBF) are blank. If this section is skipped the body forces are applied to all materials. Define NMTBF material or part ID's. Use as many cards as necessary.

Columns	Quantity	Format
1-10	First material or part ID to be included	I10
11-20	Second material or part ID	I10
21-30	Third material or part ID	I10
31-40	Fourth material or part ID	I10
.		
.		
.		
71-80	Eighth material or part ID	I10

47. Generalized Body Force Load Input

NUMGBL Card Sets (2 Cards)

Input IBODYL cards for each generalized body load (see Control Card 3, columns 16-20).

Card 1 (415,6E10.0), or (4I10,4E10.0) for LARGE option

Columns		Quantity	Format	
1-5	(1-10)	Beginning node for body load	I5	(I10)
6-10	(11-20)	Ending node for body load	I5	(I10)
11-15	(21-30)	Load curve number	I5	(I10)
16-20	(31-40)	Load curve number for dynamic relaxation phase (optional)	I5	(I10)
21-30	(41-50)	x-center of rotation	E10.0	(E10.0)
31-40	(51-60)	y-center of rotation	E10.0	(E10.0)
41-50	(61-70)	z-center of rotation	E10.0	(E10.0)
51-60	(71-80)	x-translational acceleration	E10.0	(E10.0)
61-70		y-translational acceleration	E10.0	
71-80		z-translational acceleration	E10.0	

Card 2 (20X,3E10.0), or (5E10.0) for LARGE option

Columns		Quantity	Format	
blank	(1-10)	y-translational acceleration		(E10.0)
blank	(11-20)	z-translational acceleration		(E10.0)
21-30	(21-30)	x-angular velocity	E10.0	(E10.0)
31-40	(31-40)	y-angular velocity	E10.0	(E10.0)
41-50	(41-50)	z-angular velocity	E10.0	(E10.0)

48. Momentum Deposition Data

(I5,4E10.0), or (I8,4E10.0) for LARGE option

Skip this section if NELMD (columns 31-35 of Control Card 3) is blank. Otherwise enter one card as follows for each element receiving momentum deposition. This option applies only to solid elements.

Columns		Quantity	Format	
1-5	(1-8)	Element number	I5	(I8)
6-15	(9-18)	x-momentum	E10.0	(E10.0)
16-25	(19-28)	y-momentum	E10.0	(E10.0)
26-35	(29-38)	z-momentum	E10.0	(E10.0)
36-45	(39-48)	Deposition time	E10.0	(E10.0)

49. Detonation Point Data

Skip this section if NDTPTS (columns 26-30 of Control Card 3) is blank. Otherwise, enter one or two cards as follows for each detonation point. Also, see the control parmeter, ISHADOW, on Control Card 10, columns 41-45, if geometric effects are important.

Card 1 (E10.0,I5,3E10.0)

Columns	Quantity	Format
1-10	Lighting time for detonation point. This option is not used in an acoustic bounary definition. Detonation points are used with material type 8.	E10.0
11-15	Active material ID, MID EQ.-1: an acoustic boundary EQ. 0: all high explosive materials are considered EQ. n: material ID, n, is ignited by this detonation point	I5
16-25	x-coordinate of detonation point	E10.0
26-35	y-coordinate of detonation point	E10.0
36-45	z-coordinate of detonation point (define for 3D problems)	E10.0

For solid elements (not acoustic) two options are available. If ISHADOW (see Control Card 10, Columns 41-45) is equal to 0, 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 ISHADOW is equal to 1, 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 the ISHADOW=1 option but care must be taken when setting up the input. This option works for two and three dimensional solid elements. It is recommended that for best results:

1. Keep the explosive mesh as uniform as possible with elements of roughly the same dimensions.

2. Inert obstacle such as wave shapers within the explosive must be somewhat larger than the characteristic element dimension for the automatic tracking to function properly. Generally, a factor of two should suffice. The characteristic element dimension is found by checking all explosive elements for the largest diagonal

3. The detonation points should be either within or on the boundary of the explosive. Offset points may fail to initiate the explosive.

4. Check the computed lighting times in the post processor LS-TAURUS. 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. Too many detonation points may result in significant initialization cost.

Optional Card 2 for Acoustic Boundary (5E10.0,I10)

Insert Card 2 if and only if MID<0. See Figure 49.1.

Columns	Quantity	Format
1-10	Peak pressure, p_o, of incident pressure pulse.	E10.0
11-20	Decay constant, τ	E10.0
21-30	x-coordinate of standoff point	E10.0
31-40	y-coordinate of standoff point	E10.0

Columns	Quantity	Format
41-50	z-coordinate of standoff point	E10.0
51-60	Reference node near structure.	I10

The pressure versus time curve is defined by: $p(t) = p_o e^{-\frac{t}{\tau}}$.

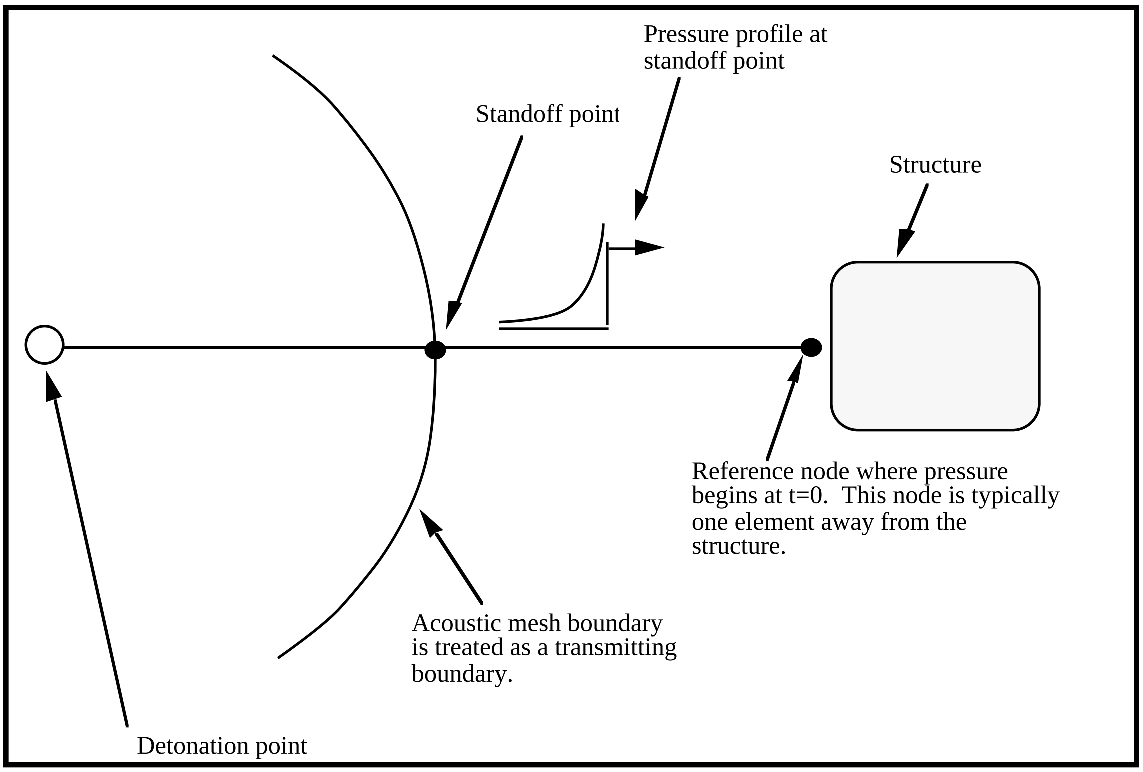


Figure 49.1 Initialization of the initial pressures due to an explosive disturbance is performed in the acoustic media. LS-DYNA automatically determines the acoustic mesh boundary and applies the pressure time history to the boundary. This option is only applicable to the acoustic element formulation.

50. Shell-Brick Interfaces

For each interface, NBLK (see Control Card 4, columns 11-15):

Card 1 (2I5)

Columns	Quantity	Format
1-5	Number of shell nodes	I5
6-10	Number of brick nodes tied to each shell node (maximum of eight)	I5

Cards 2...Number of Shell Nodes +1 (10I5), or (10I8) for LARGE option

Columns		Quantity	Format
1-5	(1-8)	Shell node, s_1	I5 (I8)
6-10	(9-16)	Brick node, n_1	I5 (I8)
11-15	(17-24)	Brick node, n_2	I5 (I8)
.	.	.	.
.	.	.	.
.	.	.	.
41-45	(65-72)	Brick node, n_8	I5 (I8)

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 eight brick nodes lying along the tangent vector to the nodal fiber. See Figure 50.1. 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.

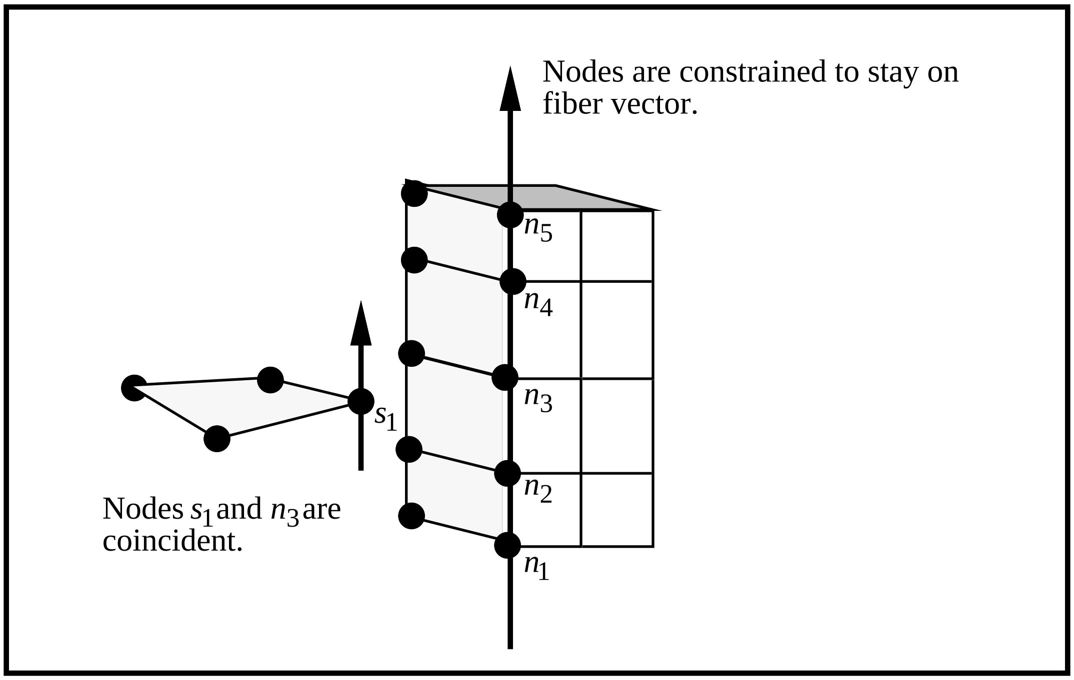


Figure 50.1. The interface between shell elements and solids ties shell node s_1 to a line of nodes on the solid elements n_1-n_5. It is very important for the nodes to be aligned.

51. Discrete Springs, Dampers, and Masses

Skip this section if NMMTDE, NMELDE, NMMASS, and NMCORD all equal zero (see Control Card 6, columns 1-20).

This section provides for the definition of simple lumped masses, node to node, or node to ground, translational and rotational springs and dampers. 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.

Define NMMTDE card sets below.

(3I5,6E10.0) or (I10,2I5,6E10.0) for MLARG option

Columns		Quantity	Format	
1-5	(1-10)	Material number (≤ NMMTDE)	I5	(I10)
6-10	(11-15)	Material type: EQ.1: linear elastic EQ.2: linear viscous EQ.3: isotropic elastoplastic EQ.4: nonlinear elastic EQ.5: nonlinear viscous EQ.6: general nonlinear EQ.7: three parameter viscoelastic EQ.8: inelastic tension or compression only EQ.15: muscle	I5	(I5)
11-15	(16-20)	Displacement/ Rotation Option EQ.0: the material describes a linear spring/damper EQ.1: the material describes a torsional spring/damper	I5	(I10)
16-25	(21-30)	Dynamic magnification factor, k_d	E10.0	(E10.0)
26-35	(31-40)	Test velocity, V_0	E10.0	(E10.0)
36-45	(41-50)	Clearance	E10.0	(E10.0)
46-55	(51-60)	Failure deflection (twist)	E10.0	(E10.0)
56-65	(61-70)	Deflection (twist) limit in compression (see comment below)	E10.0	(E10.0)
66-75	(71-80)	Deflection (twist) limit in tension (see comment below)	E10.0	(E10.0)

The constants in columns 16-75 are optional and do not need to be defined.

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 velocity

V_0 = dynamic test velocity

For example, if it is known that a component shows a dynamic crush force at 15μ/s equal to 2.5 times the static crush force, use k_d = 1.5 and V_0 = 15.

Here, “clearance” defines a compressive displacement which the spring must undergo before beginning the force-displacement relation given by the load curve. If a non-zero clearance is defined, the spring is compressive only.

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 compression is limited.

Cards 2,4,6,...(10E10.0)

Material Type 1 for Discrete Elements (Linear Elastic)

Columns	Quantity	Format
1-10	Elastic stiffness (force/displacement)	E10.0

Material Type 2 for Discrete Elements (Linear Viscous)

Columns	Quantity	Format
1-10	Damping constant (force/displacement rate)	E10.0

Material Type 3 for Discrete Elements (Isotropic Elastoplastic)

Columns	Quantity	Format
1-10	Elastic stiffness (force/displacement)	E10.0
11-20	Tangent stiffness (force/displacement)	E10.0
21-30	Yield (force)	E10.0

Material Type 4 for Discrete Elements (Nonlinear Elastic)

Columns	Quantity	Format
1-10	Load curve number describing force versus displacement relationship	E10.0
11-20	Optional load curve describing scale factor on force as a function of relative velocity	E10.0

Material Type 5 for Discrete Elements (Nonlinear Viscous)

Columns	Quantity	Format
1-10	Load curve number describing force versus rate-of-displacement relationship	E10.0

Material Type 6 for Discrete Elements (General Nonlinear)

Columns	Quantity	Format
1-10	Load curve number describing force versus displacement relationship for loading. See Figure 51.1.	E10.0
11-20	Load curve number describing force versus displacement relationship for unloading	E10.0
21-30	Hardening parameter, β EQ.0.0: tensile and compressive yield with strain softening (negative or zero slope allowed in the force versus displacement load curves). NE.0.0: kinematic hardening **without strain softening** EQ.1.0: isotropic **hardening without strain softening**	E10.0
31-40	Initial yield force in tension (> 0)	E10.0
41-50	Initial yield force in compression (< 0)	E10.0

Load curve points are in the format (displacement, force (2 E10.0)). The points must be in order starting with the most negative (compressive) displacement and ending with the most positive (tensile). The curves need not be symmetrical.

The displacement origin of the “unloading” curve is arbitrary, since it will be shifted as necessary as the element extends and contracts. On reverse yielding the “loading” curve will also be shifted along the displacement axis. The initial tensile and compressive yield forces (F_{YT} and F_{YC}) define a range within which the element remains elastic (i.e. the “loading” curve is used for both loading and unloading). If at any time the force in the element exceeds this range, the element is deemed to have yielded, and at all subsequent times the “unloading” curve is used for unloading.

Material Type 7 for Discrete Elements (Three Parameter Maxwell Viscoelastic)

Columns	Quantity	Format
1-10	K_0, short time stiffness	E10.0
11-20	K_∞, long time stiffness	E10.0
21-30	β, decay parameter	E10.0
31-40	Cut off time	E10.0
41-50	Force after cutoff time	E10.0
51-60	Incremental (default)/continuous (non zero)	E10.0

The time varying stiffness $K(t)$ may be described in terms of the input parameters as

$$K(t) = K_\infty + (K_0 - K_\infty)e^{-\beta t}$$

This equation was implemented by [Schwer et al.] as either a continuous function of time or incrementally following the approach of Herrmann and Peterson [1968]. The continous function of time implementation has the disadvantage of the energy absorber's resistance decaying with increasing time, even without deformation. The advantage of the incremental implementation is that the energy absorber must undergo some deformation before its resistance decays; i.e. there is no decay until impact, even in delayed impacts. The disadvantage of the incremental implementation is that very rapid decreases in resistance cannot be easily matched.

Material Type 8 for Discrete Elements (Inelastic Tension or Compression Only)

Columns	Quantity	Format
1-10	Load curve number describing arbitrary force/torque versus displacement/twist relationship. This curve must be defined in the positive force-displacement quadrant regardless of whether the spring acts in tension or compression.	E10.0
11-20	Unloading stiffness (optional). If zero, the maximum loading stiffness in the force displacement curve is used.	E10.0
21-30	Flag for compression/tension EQ.-1.0: tension only EQ.0.0: default is set to 1.0 EQ.1.0: compression only.	E10.0

Material Type 15 for Discrete Elements (Muscle)

This material is a Hill-type muscle model with activation. See the appendix for a complete derivation and description of the input data.

Card 1 (8E10.0)

Columns	Quantity	Format
1-10	*Lo*, the initial muscle length. (DEFAULT = 1.0)	E10.0
11-20	*Vmax*, the maximum CE shortening velocity.	E10.0
21-30	*Sv*, the scale factor for *Vmax* vs. active state. .LT.0: absolute value gives load curve ID .GE.0: constant value of 1.0 is used	E10.0
31-40	*A*, the activation level vs. time function .LT.0: absolute value gives load curve ID .GE.0: constant value of *A* is used	E10.0
41-50	*Fmax*, the peak isometric force	E10.0
51-60	*TL*, the active tension vs. length function .LT.0: absolute value gives load curve ID .GE.0: constant value of 1.0 is used	E10.0
61-70	*TV*, the active tension vs. velocity function .LT.0: absolute value gives load curve ID .GE.0: constant value of 1.0 is used	E10.0
71-80	*Fpe*, the force/length function for the parallel elastic element .LT.0: absolute value gives load curve ID .EQ.0: exponential function is used (see appendix) .GT.0: constant value of 0.0 is used	E10.0

Card 2 (2E10.0)

Columns	Quantity	Format
1-10	*Lmax*, the relative length when *Fpe* reaches *Fmax*. Required if the exponential function for *Fpe* is chosen above.	E10.0
11-20	*Ksh*, a constant governing the exponential rate of rise of *Fpe*. Required if the exponential function for *Fpe* is chosen above.	E10.0

Orientation Vectors
(I5,3E10.0)

Define NMCORD orientation vectors below.

Columns	Quantity	Format
1-5	Option, IOPT 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 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 nodes projected onto the plane normal to the a vector defined by the following two nodes.	I5
6-15	x-value of orientation vector or node 1 if IOPT=2 & 3	E10.0
16-25	y-value of orientation vector or node 2 if IOPT=2 & 3	E10.0
26-35	z-value of orientation vector (define for IOP=0 & 1)	E10.0

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.

Discrete Spring/Damper Element Cards
(4I5,E10.0,4I5,E10.0), or (4I8,E10.0,4I5,E10.0) for LARGE option

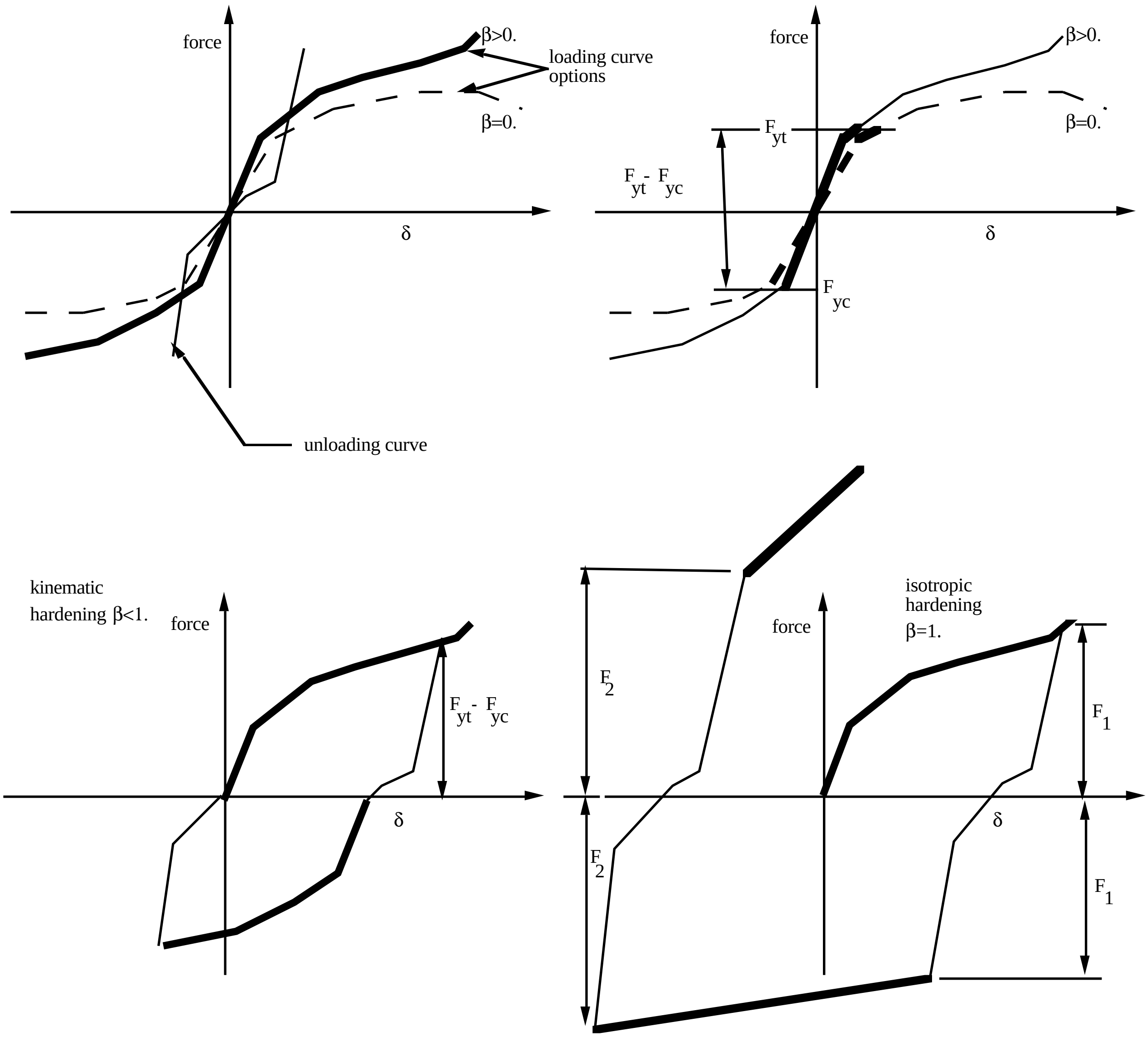
Define NMELDE discrete springs and dampers below.

Columns		Quantity	Format	
1-5	(1-8)	Element number	I5	(I8)
6-10	(9-16)	Node, n_1	I5	(I8)
11-15	(17-24)	Node, n_2 EQ.0: the spring/damper connects node n_1 to ground.	I5	(I8)
16-20	(25-32)	Material number	I5	(I8)
21-30	(33-42)	Scale factor on force (default = 1.0)	E10.0	(E10.0)
31-35	(43-47)	Orientation option (See **Orientation Vectors** above): EQ.0: the spring/damper acts along the axis from node n_1 to n_2 EQ.n: the spring/damper acts along the axis defined by the nth orientation vector	I5	(I5)
36-40	(48-52)	Print flag EQ.0: forces are printed EQ.1: forces are not printed	I5	(I5)
41-45	(53-57)	Generation increment, k (default=1)	I5	(I5)
46-50	(58-62)	Flag to display discrete element as a beam element in the TAURUS Database. A null beam/truss element with the same connectivity as the discrete element must be defined in the beam element input section. The spring resultant forces are written into the TAURUS database and can be plotted by referencing the corresponding beam element ID. The beam element must be defined as a null material type (Type 9). EQ.0: beams are not defined EQ.1: beams are defined	I5	(I5)
51-60	(63-72)	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.	E10.0	(E10.0)

If arbitrary numbering is not used then the omitted data are automatically generated with respect to the first card prior to the omitted data as

$$n_j^{i+1} = n_j^i + k \; .$$

The generation parameter k is taken from the first card.



Discrete Lumped Nodal Masses
(I5,E10.0), or (I8,E10.0) for LARGE option

Define NMMASS lumped masses.

Columns		Quantity	Format	
1-5	(1-8)	Node number	I5	(I8)
6-15	(9-18)	Mass	E10.0	(E10.0)
16-20	(19-26)	Element ID	I5	(I8)

Discrete Lumped Nodal Inertias
(3I5), or (I8,I5,I8) for LARGE option

Define NUMRBI lumped inertias. To define a local coordinate system for the nodal inertia tensor, set IRCS=1 below.

Columns		Quantity	Format
1-5	(1-8)	Node number	I5 (I8)
6-10	(9-13)	Flag for inertia tensor reference coordinate system, IRCS EQ.0: global inertia tensor EQ.1: principal moments of inertias with orientation vectors defined below.	I5 (I5)
11-15	(14-21)	Element ID	I5 (I8)

Card 2 (6E10.0)

Columns	Quantity	Format
1-10	I_{xx}, xx component of inertia tensor	E10.0
11-20	I_{xy} (set to zero if IRCS=1)	E10.0
21-30	I_{xz} (set to zero if IRCS=1)	E10.0
31-40	I_{yy}	E10.0
41-50	I_{yz} (set to zero if IRCS=1)	E10.0
51-60	I_{zz}	E10.0

Optional Card 3 (IRCS=1)

Define these vectors for the orientation of the inertia tensor in a local coordinate system. The xy plane is described by two vectors: the local x axis and another vector lying in the plane. The local z axis is the cross product of these two vectors. The local y axis is found by taking the cross product of the local z and x axes.

(6E10.0)

Columns	Quantity	Format
1-10	x-coordinate on local x-axis. Origin lies at (0,0,0)	E10.0
11-20	y-coordinate on local x-axis	E10.0
21-30	z-coordinate on local x-axis	E10.0
31-40	x-coordinate of local in-plane vector	E10.0
41-50	y-coordinate of local in-plane vector	E10.0
51-60	z-coordinate of local in-plane vector	E10.0

52. Seat Belts

Define this section if values are defined on the seat belt control card, Card 6 in the control card input section.

Belt Material Definition (I5,4E10.0) or (I10,4E10.0) for MLARG option

Define one card for each belt material:

Columns		Quantity	Format	
1-5	(1-10)	Belt material number†	I5	(I10)
6-10	(11-20)	Load curve for loading	E10.0	(E10.0)
11-15	(21-25)	Load curve for unloading	E10.0	(E10.0)
16-20	(26-30)	Mass per unit length	E10.0	(E10.0)
16-20	(26-30)	Minimum length (for elements connected to sliprings and retractors)	E10.0	(E10.0)

†Material numbers must start at 1 and be consecutive.

Each belt material defines stretch characteristics and mass properties for a set of belt elements. The user enters a load curve for loading, the points of which are *(Strain, Force)*. Strain is defined as engineering strain, i.e.

$$Strain = \frac{current\ length}{initial\ length} - 1.$$

Another similar curve is entered to describe the unloading behavior. Both loadcurves should start at the origin *(0,0)* and contain positive force and strain values only. The belt material is tension only with zero forces being generated whenever the strain becomes negative. The first non-zero point on the loading curve defines the initial yield point of the material. On unloading, the unloading curve is shifted along the strain axis until it crosses the loading curve at the ‘yield’ point from which unloading commences. If the initial yield has not yet been exceeded or if the origin of the (shifted) unloading curve is at negative strain, the original loading curves will be used for both loading and unloading. If the strain is less than the strain at the origin of the unloading curve, the belt is slack and no force is generated. Otherwise, forces will then be determined by the unloading curve for unloading and reloading until the strain again exceeds yield after which the loading curves will again be used.

A small amount of damping is automatically included. This reduces high frequency oscillation, but, with realistic force-strain input characteristics and loading rates, does not significantly alter the overall forces-strain performance. The damping forced opposes the relative motion of the nodes and is limited by stability:

$$D = \frac{.1 \times mass \times relative\ velocity}{timestep\ size}$$

In addition, the magnitude of the damping forces is limited to one tenth of the force calculated from the forces-strain relationship and is zero when the belt is slack. Damping forces are not applied to elements attached to sliprings and retractors.

The user inputs a mass per unit length that is used to calculate nodal masses on initialization.

A ‘minimum length’ is also input. This controls the shortest length allowed in any element and determines when an element passes through sliprings or are absorbed into the retractors. One tenth of a typical initial element length is usually a good choice.

Belt Element Definition
(5I5,E10.0,I5), or (5I8,E10.0,I5) for Large Option

Define one card for each belt element:

Columns		Quantity	Format	
1-5	(1-8)	Belt element number	I5	(I8)
6-10	(9-16)	Node 1	I5	(I8)
11-15	(17-24)	Node 2	I5	(I8)
16-20	(25-32)	Material number	I5	(I8)
21-25	(33-40)	Retractor number*	I5	(I8)
26-35	(41-50)	Initial slack length (added to the total length) (generally 0.0)	E10.0	(E10.0)
36-40	(51-55)	Flag to display seatbelt element as a beam element in the TAURUS Database. A null beam/truss element with the same connectivity as the seatbelt element must be defined in the beam element input section. The resultant forces are written into the TAURUS database and can be plotted by referencing the corresponding beam element ID. The beam element must be defined as a null material type (Type 9) EQ.0: beams are not defined EQ.1: beams are defined	I5	(I5)

*The retractor number should only be defined if the element is initially inside a retractor.

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.

Slipring Definition
(3I5,E10.0,I5) or (3I8,E10.0,I8) for Large Option

Define one card for each slipring:

Columns		Quantity	Format	
1-5	(1-8)	Slipring number†	I5	(I8)
6-10	(9-16)	Element 1*	I5	(I8)
11-15	(17-24)	Element 2*	I5	(I8)
16-25	(25-34)	Friction coefficient	E10.0	(E10.0)
26-30	(35-42)	Slipring node*	I5	(I8)

†Slipring numbers should start at 1 and be consecutive.

*Elements 1 and 2 should share a node which is coincident with the slipring node. The slipring 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 52.1) 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 belt s are taken ass 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 52.1). The new unstretched length of e1 is 1.1 $\times$ 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.

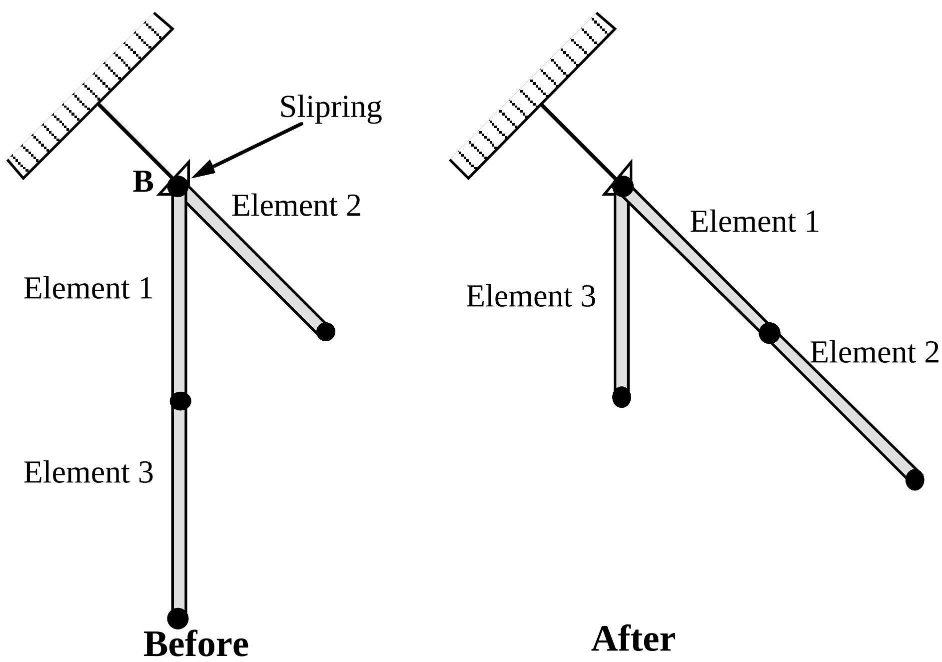


Figure 52.1. Elements passing through slipring.

Retractor Definition

Define two cards for each retractor.

Retractor Card 1
(7I5) or (I5,2I8,4I5) for Large Option

Columns		Quantity	Format	
1-5	(1-5)	Retractor number†	I5	(I5)
6-10	(6-13)	Retractor node*	I5	(I8)
11-15	(16-21)	Belt element number*	I5	(I8)
16-20	(22-26)	Sensor number (at least one sensor should be defined)	I5	(I5)
21-25	(27-31)	Sensor number	I5	(I5)
26-30	(32-36)	Sensor number	I5	(I5)
31-35	(37-61)	Sensor number	I5	(I5)

†Retractor numbers should start at 1 and be consecutive.

*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.

Retractor Card 2 (5E10.0)

Columns	Quantity	Format
1-10	Time delay after sensor triggers	E10.0
11-20	Amount of pull-out between time delay ending and retractor locking	E10.0
21-30	Load curve for loading (pull-out, force)*	E10.0
31-40	Load curve for unloading (pull-out, force)†	E10.0
41-50	Fed length**	E10.0

*The first point of the loadcurve should be (0,T_{min}). Tmin is the minimum tension. All subsequent tension values should be greater than T_{min}.

†The unloading curves should start at zero tension and increase monotonically (i.e. no segments of negative or zero slope).

**This should be at least three times the minimum length.

Retractors allow belt material to be payed out into a belt element. Retractors operate in one of two regimes: unlocked when the belt material is payed 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 payed 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 $\times$ fed length per time step. Because of this, the constant tension value is not always be 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 payed 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 payed 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

$$L_{crit} = \text{fed length} - 1.1 \times \text{minimum length}$$

(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 52.2, 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 stricture; belt elements should not be connected to this node directly, but any other feature can be attached including rigid bodies. The mouth element should have a node coincident with the retractor but should not be inside the retractor. The fed length would typically be set either to a typical element initial length, for the distance between painted marks on a real belt for comparisons with high speed film. The fed length should be at least three times the minimum length.

If there are elements initially inside the retractor (e2, e3 and e4 in the Figure) they should not be referred to on the retractor input, but the retractor should be identified on the element input for these elements. Their nodes should all be coincident with the retractor node and should not be restrained or constrained. Initial slack will automatically be set to $1.1 \times$ minimum length for these elements; this overrides any user-defined value.

Weblockers can be included within the retractor representation simply by entering a 'locking up' characteristic in the force pullout curve, see Figure 52.3. The final section can be very steep (but must have a finite slope).

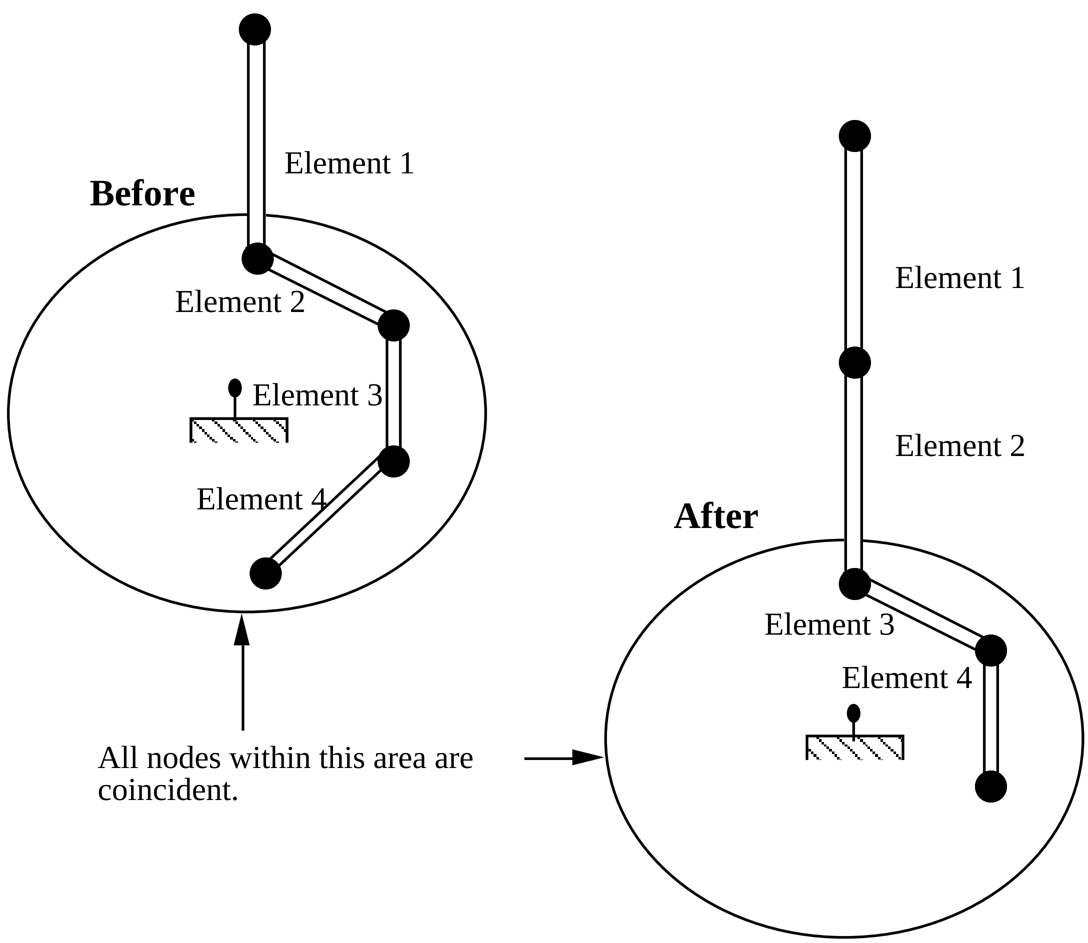


Figure 52.2. Elements in a retractor.

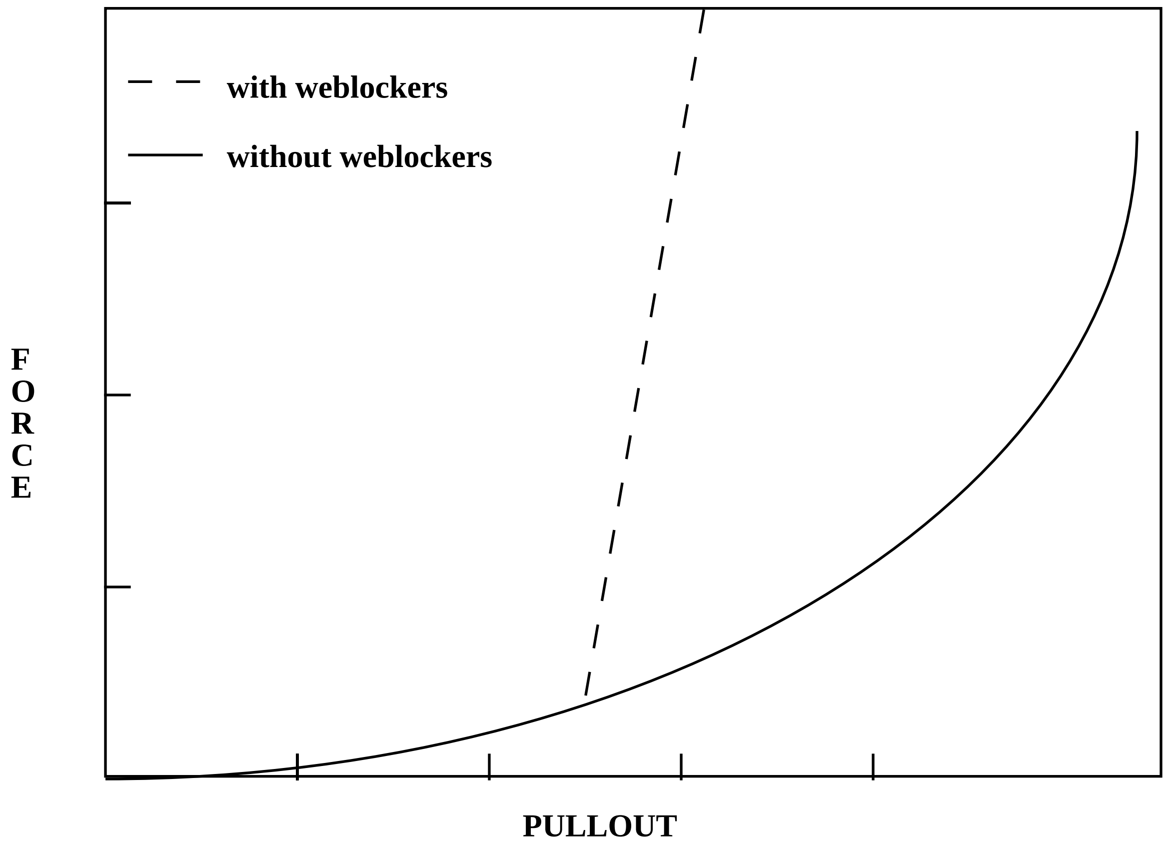


Figure 52.3 Retractor force pull characteristics.

Sensor Definition

Define two cards for each sensor:

Sensor Card 1 (3I5)

Columns	Quantity	Format
1-5	Sensor number†	I5
6-10	Sensor TYPE, EQ.1: acceleration of node EQ.2: retractor pull-out rate EQ.3: time EQ.4: distance between nodes	I5
11-15	Flag EQ.0: sensor inactive during dynamic relaxation EQ.1: sensor can be triggered during dynamic relaxation	I5

†Sensor numbers should start at 1 and be consecutive.

The meaning of Card 2 depends on the sensor type:

Sensor Card 2 (If TYPE=1)
(2I5,2E10.0) or (I8,I5,2E10.0) for Large Option

Columns		Quantity	Format	
1-5	(1-8)	Node number†	I5	(I8)
6-10	(9-13)	Degree of freedom EQ.1: x EQ.2: y EQ.3: z	I5	(I5)
11-20	(14-23)	Acceleration	E10.0	(E10.0)
21-30	(24-33)	Time over which acceleration must be exceeded	E10.0	(E10.0)

†Node should not be on rigid body, velocity boundary condition, or other ‘imposed motion’ feature.

Sensor Card 2 (If TYPE=2) (I5,2E10.0)

Columns	Quantity	Format
1-5	Retractor number	I5
6-15	Rate of pull-out (length/time units)	E10.0
16-25	Time over which rate of pull-out must be exceeded	E10.0

Sensor Card 2 (If TYPE=3) (E10.0)

Columns	Quantity	Format
1-10	Time at which sensor triggers	E10.0

Sensor Card 2 (If TYPE=4)
(2I5,2E10.0) or (2I8,2E10.0) for Large Option

Columns		Quantity	Format	
1-5	(1-8)	Node 1	I5	(I8)
6-10	(9-16)	Node 2	I5	(I8)
11-20	(17-26)	Maximum distance d_{max}	E10.0	(E10.0)
21-30	(27-36)	Minimum distance d_{min}	E10.0	(E10.0)

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 sensor 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 mas/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.

Pretensioner Definition

Define the following two cards for each pretensioner:

Pretensioner Card 1 (6I5)

Columns	Quantity	Format
1-5	Pretensioner number†	I5
6-10	Type EQ.1: pyrotechnic retractor EQ.2: pre-loaded spring becomes active EQ.3: lock spring removed EQ.4: distance between nodes	I5
11-15	Sensor 1 (at least one sensor should be defined)	I5
16-20	Sensor 2	I5
21-25	Sensor 3	I5
26-30	Sensor 4	I5

†Pretensioner numbers should start at 1 and be consecutive.

Pretensioner Card 2 (If TYPE=1)
(I5,2E10.0) or (I8,2E10.0) for Large Option

Columns		Quantity	Format	
1-5	(1-8)	Retractor number	I5	(I8)
6-15	(9-18)	Time between sensor triggering and pretensioner acting	E10.0	(E10.0)
16-25	(19-28)	Loadcurve for pretensioner (time after activation, pull-in)	E10.0	(E10.0)

Pretensioner Card 2 (If TYPE=2 or 3)
(I5,E10.0) or (I8,E10.0) for Large Option

Columns		Quantity	Format	
1-5	(1-8)	Spring element number	I5	(I8)
6-15	(9-18)	Time between sensor triggering and pretensioner acting	E10.0	(E10.0)

Pretensioners allow modelling 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 represents 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 cancelled 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.

Accelerometers

Define this section if the option to use accelerometers is selected on Control Card 6. Define one card for each accelerometer. This option is related to output of rigid body nodal accelerations, velocities, and cross-sectional forces (in the file SECFORC) in the local system of the accelerometer.

Accelerometer Card
(5I5) or (5I8) for LARGE option

Columns		Quantity	Format
1-5	(1-8)	Accelerometer ID in the kth rigid body	I5 (I8)
6-10	(9-16)	Node 1 in the kth rigid body	I5 (I8)
11-15	(17-24)	Node 2 in the kth rigid body	I5 (I8)
16-20	(25-32)	Node 3 in the kth rigid body	I5 (I8)
21-25	(33-40)	Gravitational flag if body force loads are active EQ.0: do not subtract base accelerations EQ.1: subtract base accelerations	I5 (I8)

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 $\mathbf{y} = \mathbf{x} \times \mathbf{z}$

Generally, the three nodes should belong to the same rigid body. The local axis then rotates with the body.

53. Rigid Body Inertial Properties and Constraints

Define the number of sets, NUMRBI, specified on Control Card 5 in columns 21-25. __Please note__: **When rigid bodies are merged to a master rigid body the inertial properties and constraints defined for the master rigid body apply to all members of the merged set.**

Card 1 (A2,3X,4E10.0,I10,I5)

Note: All data must be provided. This data supersedes other input data (e.g. nodal initial velocities).

To define a local coordinate system for a nodal rigid body constraint set for output purposes and airbag sensors, set IRCS=1 below and all other input data on input cards 1-3 to zero. Define the local system on card 4.

Columns	Quantity	Format
1-2	Input “RB” for rigid body or “CS” for nodal rigid body constraint set	A2
3-5	Blank	3X
6-15	x-coordinate of center of mass	E10.0
16-25	y-coordinate of center of mass	E10.0
26-35	z-coordinate of center of mass	E10.0
36-45	Translational mass	E10.0
46-55	Material number of rigid body or constraint set number	I10
56-60	Flag for inertia tensor reference coordinate system, IRCS EQ.0: global inertia tensor EQ.1: principal moments of inertias with orientation vectors defined below.	I5

Card 2 (6E10.0)

Columns	Quantity	Format
1-10	I_{xx}, xx component of inertia tensor	E10.0
11-20	I_{xy} (set to zero if IRCS=1)	E10.0
21-30	I_{xz} (set to zero if IRCS=1)	E10.0
31-40	I_{yy}	E10.0
41-50	I_{yz} (set to zero if IRCS=1)	E10.0
51-60	I_{zz}	E10.0

Card 3 (6E10.0)

Columns	Quantity	Format
1-10	x-rigid body translational velocity	E10.0
11-20	y-rigid body translational velocity	E10.0
21-30	z-rigid body translational velocity	E10.0
31-40	x-rigid body rotational velocity	E10.0
41-50	y-rigid body rotational velocity	E10.0
51-60	z-rigid body rotational velocity	E10.0

Optional Card 4 (IRCS=1)

Define these vectors for the orientation of the inertia tensor in a local coordinate system. The xy plane is described by two vectors: the local x axis and another vector lying in the plane. The local z axis is the cross product of these two vectors. The local y axis is found by taking the cross product of the local z and x axes. The local coordinate system defined by the coordinate system ID in Section 14 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.

(6E10.0, I10)

Columns	Quantity	Format
1-10	x-coordinate of local x-axis. Origin lies at (0,0,0)	E10.0
11-20	y-coordinate of local x-axis	E10.0
21-30	z-coordinate of local x-axis	E10.0
31-40	x-coordinate of local in-plane vector	E10.0
41-50	y-coordinate of local in-plane vector	E10.0
51-60	z-coordinate of local in-plane vector	E10.0
61-70	Local coordinate system ID, see Section 14. With this option leave fields 1-6 blank.	I10

54. Nonreflecting Boundaries

(6I5,2E10.0), or (5I8,2E10.0) for LARGE option

Nonreflecting 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 nonreflecting 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.

If this is a three dimensional analysis, then define NNRBS (Control Card 2, columns 16-20) four noded segment cards in this section. If not, use the two dimensional input which follows the three dimensional input below.

Input for Three Dimensional Solid Elements

Columns		Quantity	Format	
1-5	(1-8)	Surface segment number	I5	(I8)
6-10	(-)	Increment k	I5	(omit)
11-15	(9-16)	Nodal point n_1	I5	(I8)
16-20	(17-24)	Nodal point n_2	I5	(I8)
21-25	(25-32)	Nodal point n_3	I5	(I8)
26-30	(33-40)	Nodal point n_4	I5	(I8)
31-40	(33-42)	Activation flag for dilatational waves (on.eq.0.0, off.ne.0.0)	E10.0	(E10.0)
41-50	(43-52)	Activation flag for shear waves (on.eq.0.0, off.ne.0.0)	E10.0	(E10.0)

Nonreflecting boundary segments are only used with 3D 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. The first and last nonreflecting boundary segments must be explicitly defined. Gaps in intermediate segments numbers are filled by automatically generating segment definitions by adding the generation increment k to each node number of the previous segment. Care should be exercised when using the generation option whenever arbitrary numbering is used.

Input for Two Dimensional Solid Elements

If this is a two dimensional analysis, then define NNRBS (Control Card 2, columns 16-20) transmitting boundaries. If not, use the three dimensional input above.

Card 1 (I10)

Columns	Quantity	Format
1-5	NBNS, number of boundary nodes	I10

Cards 2,..., 1+NBNS (2I10)

Columns	Quantity	Format
1-5	Boundary point number EQ.0: increment last value by 1	I10
6-10	Nodal point ID	I10

A non-reflecting boundary must be defined by at least two points. Nodal point numbers must be given in the order in which they appear as one moves <u>counterclockwise</u> along the boundary. See Figure 54.1. Omitted data are automatically generated by incrementing the nodal point numbers by:

$$\frac{\left(n_i - n_j\right)}{\left(bn_i - bn_j\right)}$$

where bn_i, bn_j are the boundary point numbers on two successive cards and n_i, n_j are their corresponding numbers. Automatic generation is inadvisable if arbitrary numbering is used.

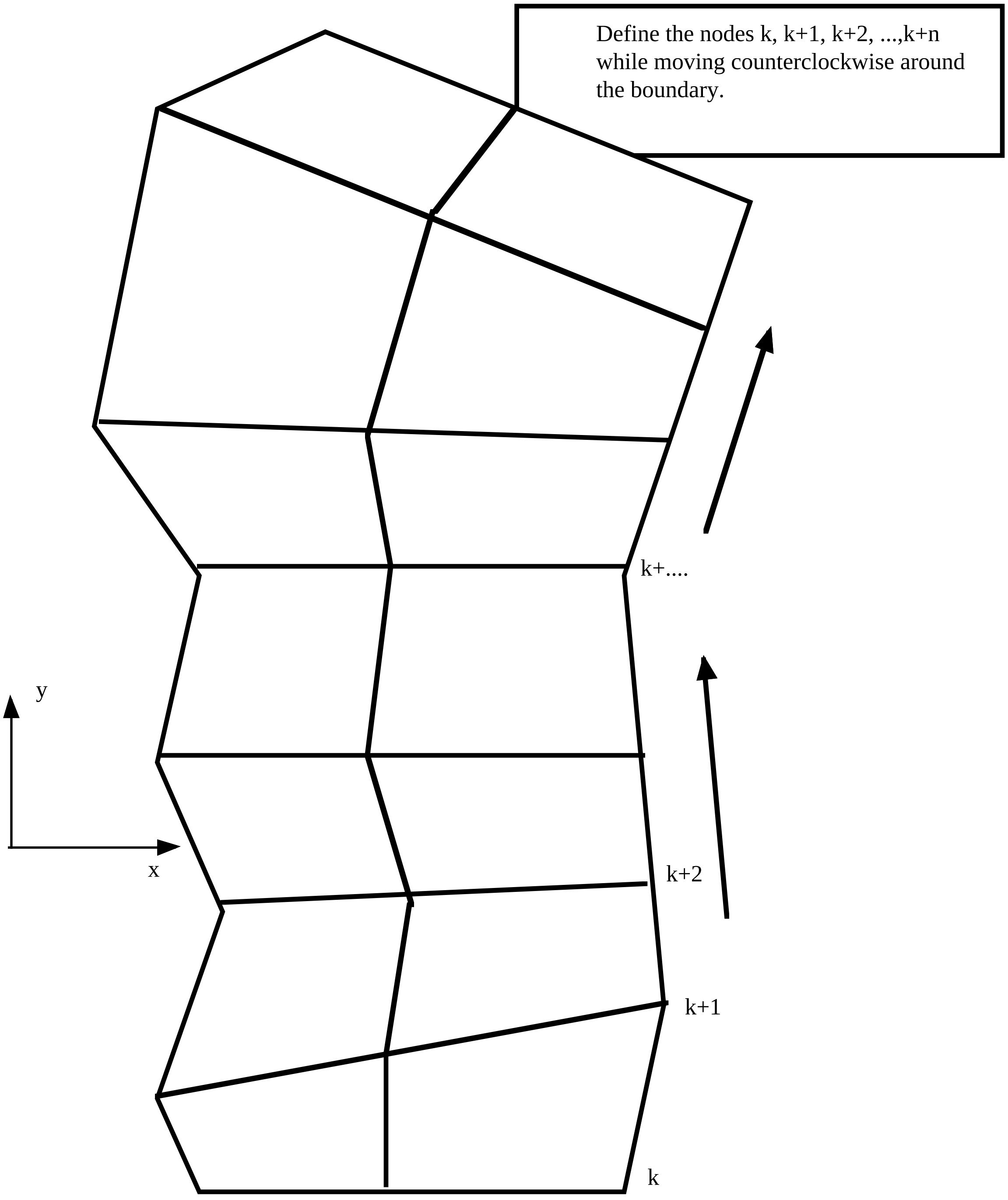


Figure 54.1 When defining a transmitting boundary in 2D define the node numbers consecutively while moving counterclockwise around the boundary.

55. Temperature Data Option I

(I5,2E10.0,2I5), or (I8,2E10.0,2I8) for LARGE option

Define NUMNP (see Control Card 1, columns 11-20) temperature cards in this Section if and only if the thermal effects option, ITEMP (see Control Card 13, columns 51-55), is set to -9999.

Columns		Quantity	Format	
1-5	(1-8)	Nodal point number	I5	(I8)
6-15	(9-18)	Temperature scale factor at this node, S_i	E10.0	(E10.0)
16-25	(19-28)	Base temperature, T_i^{base}	E10.0	(E10.0)
26-30	(29-36)	Load curve ID that multiplies scaled temperature	I5	(I8)
31-35	(37-44)	Generation increment k	I5	(I8)

Temperature data for missing nodes is generated using the specified node number increment, k, if sequential numbering is used; otherwise, define a card for each nodal point. The temperature scale factor S_i and the base temperature T_i^{base} are linearly interpolated between the starting and ending values. At any time t, the temperature at a node n_i is given by:

$$T(t) = T_i^{base} + S_i f(t)$$

where $f(t)$ is the value of the tabulated load curve at the current time.

56. Temperature Data Option II
(I5,E10.0), or (I8,E10.0) for LARGE option

Define NUMNP (see Control Card 1, columns 11-20) temperature cards in this Section if and only if the thermal effects option, ITEMP (see Control Card 13, columns 51-55), is set to -2. The reference temperature state is assumed to be a null state with this option. A nodal temperature state, read in below and held constant throughout the analysis, dynamically loads the structure.

Columns		Quantity	Format
1-5	(1-8)	Nodal point number	I5 (I8)
6-15	(9-18)	Temperature	E10.0 (E10.0)

57. 1D Slideline Definitions

In this section the input description for one-dimensional slidelines are provided. One-dimensional slidelines in three dimensional calculations are used for modeling rebars embedded in concrete. In two dimensional calculations slidelines are used to model fluid structure interactions, mesh transitions, and contact problems. In the input below we first describe the input for three dimensional calculations. This option does not apply in two dimensions. We then provide the input for the two-dimensioal case which conversely does not apply in three dimensions.

Slideline Definitions in Three Dimensions (2I5,5E10.0)

These slidelines were developed by Pelessone [1986] at GA Technologies. Define NUMSL (see Control Card 4, columns 41-45) sets of one-dimensional slidelines.

Columns	Quantity	Format
1-5	Number of slave nodes, NSN	I5
6-10	Number of master nodes, NMN	I5
11-20	External radius of rebar	E10.0
21-30	Compressive strength of concrete	E10.0
31-40	Bond shear modulus	E10.0
41-50	Maximum shear displacement	E10.0
51-60	Exponent in damage curve (Hdmg)	E10.0

Cards 2,...,NSN+1
(2I5), or (2I8) for LARGE option (Slave Nodes)

Columns		Quantity	Format
1-5	(1-8)	Slave number EQ.0: the preceding slave number is incremented by 1	I5 (I8)
6-10	(9-16)	Nodal point number	I5 (I8)

Omitted data are automatically generated by incrementing the nodal point numbers by

$$\frac{(n_i - n_j)}{(sn_i - sn_j)}$$

where sn_i, sn_j are the slave numbers on two successive cards, and n_i and n_j are their corresponding node numbers. If arbitrary node numbering is used, the generation option is not recommended.

Cards NSN+2,...,NSN+NMN+1
(2I5), or (2I8) for LARGE option
(Master Nodes)

Columns		Quantity	Format
1-5	(1-8)	Slave number EQ.0: the preceding slave number is incremented by 1	I5 (I8)
6-10	(9-16)	Nodal point number	I5 (I8)

Omitted data is generated as described above. The master nodes must be given in the order in which they appear as one moves along the line. Slidelines may cross.

Slideline Definitions in Two Dimensions (2I10,I5,4E10.0, I5)

Define NSL control cards below-one for each slideline. A discussion about the proper use of the slideline capability is provided below.

Columns	Quantity	Format
1-10	Number of slave nodes in slideline (NSN)	I10
11-20	Number of master nodes in slideline (NMN)	I10
21-25	Slideline type number, ISLT EQ.1: sliding only EQ.2: tied sliding EQ.3: sliding with voids EQ.4: penalty formulation with friction EQ.5: penalty formulation without friction EQ.6: single surface contact (NMN=0)	I5
26-35	SLFAC, tolerance for determining initial gaps EQ.0.0: SLFAC = 0.001	E10.0
36-45	θ_1, angle in degrees of slideline extension at first master node EQ.0.0: extension remains tangent to first master segment	E10.0
46-55	θ_2, angle in degrees of slideline extension at last master node EQ.0.0: extension remains tangent to last master segment	E10.0
56-65	Scale factor or penalty EQ.0.0: default set to .10	E10.0
66-70	Slideline extension bypass option EQ.1: slideline extensions are not used	I5

Angles θ_1 and θ_2 are measured counterclockwise from the r-axis and remain constant. If θ_1 and θ_2 are zero, the extensions are made tangent to the first and last master segments and remain so throughout the calculation. The force exerted by a slave node lying on an extension of the master node at the origin of the extension diminishes to zero as the slave node moves away a distance equal to the length of one slave segment.

Repeat the following cards for each slideline.

Card 1

Columns	Quantity	Format
1-72	Title card, slide line description.	12A6

For 2D slide line type 4 (penalty formulation with friction) define the following extra card.

Card 1a

Columns	Quantity	Format
1-10	Coefficient of friction	E10.0
11-20	Coefficient of friction (low velocity)	E10.0
21-30	Coefficient of friction (high velocity)	E10.0
31-40	Friction factor (shear friction)	E10.0

Cards 2,3,..., NSN+1 (2I10)

Columns	Quantity	Format
1-10	Slave number	I10
11-21	Nodal point number	I10

Omitted data are automatically generated by incrementing the nodal point numbers by:

$$\frac{n_i - n_j}{sn_i - sn_j} \quad ,$$

where sn_i, sn_j are the slave numbers on two successive cards and n_i and n_j are their corresponding numbers. Automatic generation is not recommended if arbitrary node number is used in the input.

Cards NSN+2,..., NSN+NMN+1

Columns	Quantity	Format
1-10	Master number	I10
11-20	Nodal point number	I10

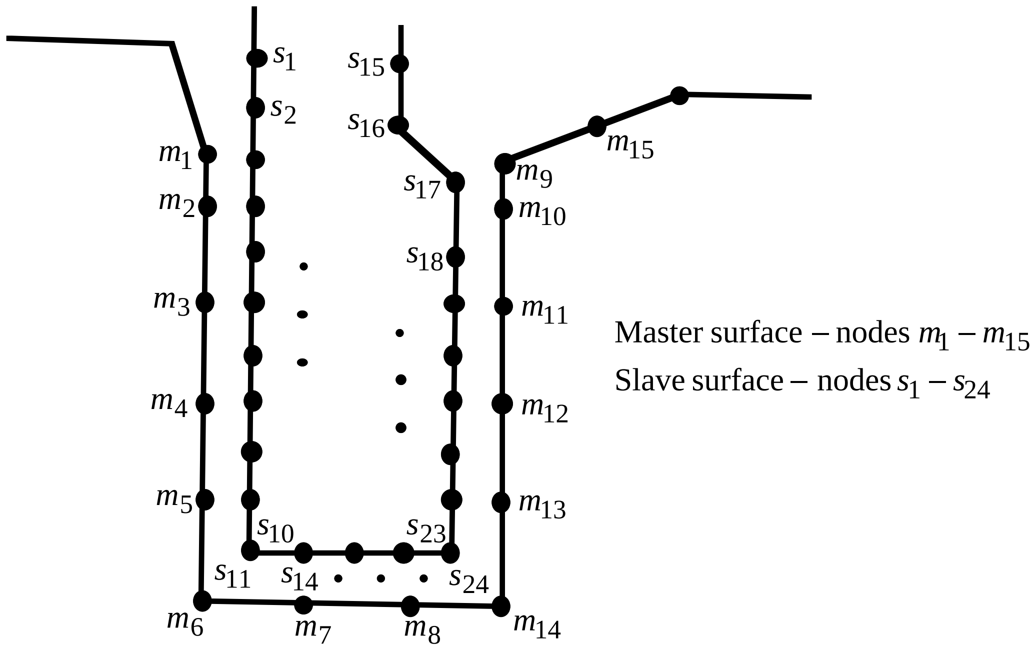
Omitted data are generated as described above. The master and slave nodes must be given in the order in which they appear as one moves along the surface. **The slave surface must be to the left of the master surface.**

Consider two surfaces in contact. In DYNA 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, type 4, 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.
- Penalty, type 4, slidelines handle intersections automatically and should not be defined for this slideline type.
- 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 57.1. 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 57.2 and 57.3, 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 57.1. 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).

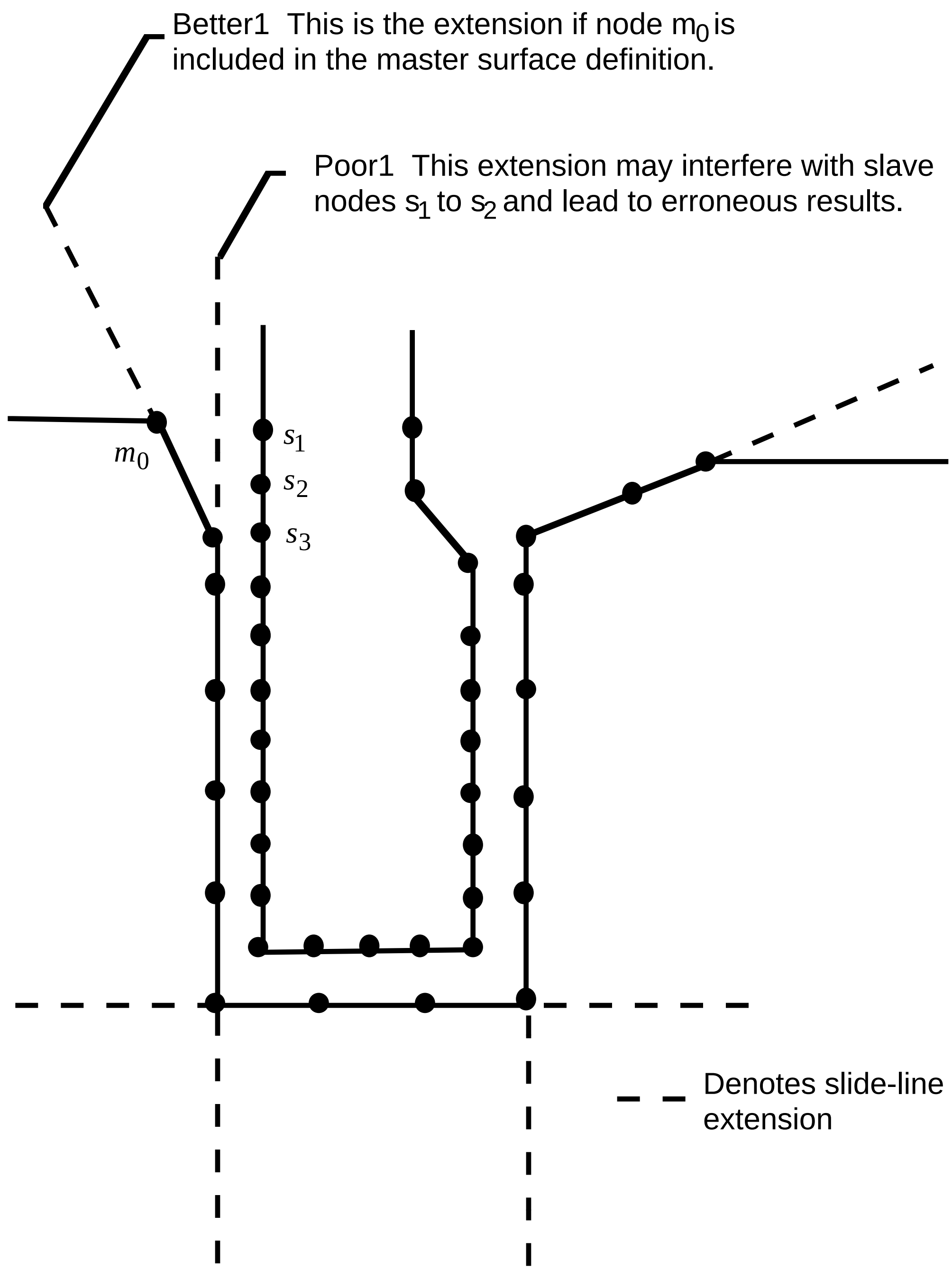


Figure 57.2. Master surface extensions defined automatically by DYNA (extensions are updated every time step to remain tangent to ends of master sides of slidelines unless angle of extension is defined in input).

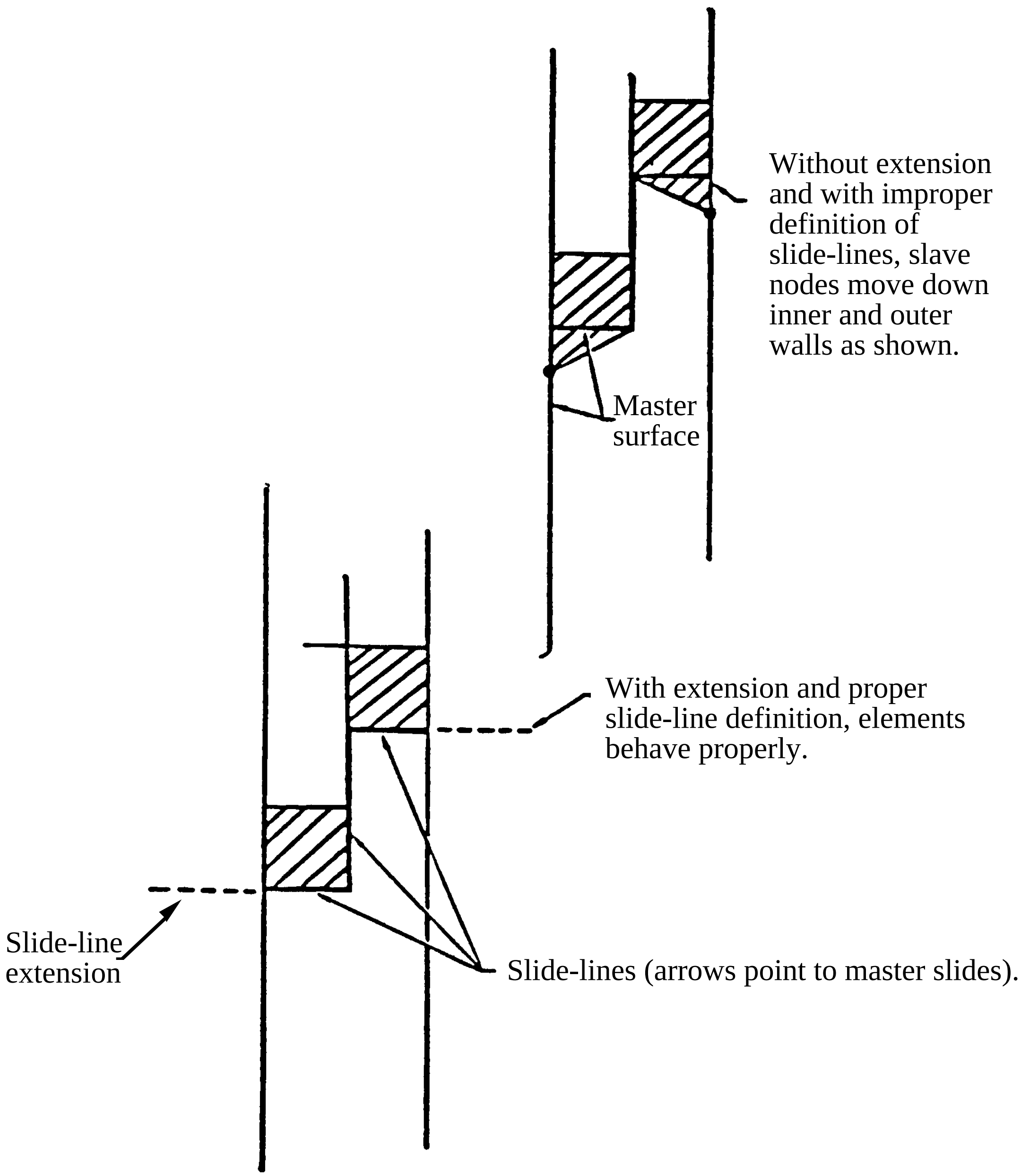


Figure 57.3. Example of slideline extensions helping to provide realistic response.

Automatic Slideline Definitions in Two Dimensions

Define a set of cards for each automatic 2D contact definition. The number of sets must equal NAUTO (Control Card 4, Col 61-65). A set consists of cards 1 and 2 plus cards 3a, 3b, 3c,...,4a, 4b, 4c,... as needed.

Card 1 (I5,E10.0,I5,3E10.0,25X,I5)

Columns	Quantity	Format
1-5	Automatic contact type, ACT (default=0) EQ.0: automatic surface to surface EQ.1: automatic node to surface EQ.2: automatic surface in continuum	I5
6-15	Scale factor for penalty stiffness (default=0.1)	E10.0
16-20	Frequency of bucket sorting (default=50)	I5
21-30	Static coefficient of friction, μ_s (default=0.0)	E10.0
31-40	Dynamic coefficient of friction, μ_k (default=0.0)	E10.0
41-50	Exponential decay coefficient, d_V (default=0.0)	E10.0
75-80	Parameter to allocate memory for bucket sorts (default=6)	I5

Card 2 (2I5,4E10.0,2I5)

Columns	Quantity	Format
1-5	Number of material parts to define the slave surface, NPSS. EQ.0: All parts included if ACT=1, invalid if ACT=2 GT.0: List NPSS material parts on cards 3a,3b,3c,...	I5
6-10	Number of material parts to define the master surface, NPMS. EQ.0: Master surface is identical to the slave surface if NPMS=0, invalid if ACT=2 GT.0: List NPMS material parts on cards 4a,4b,4c,...	I5
11-20	Birth time of contact (default=0.0)	E10.0
21-30	Death time of contact (default=1.e20)	E10.0

31-40	Surface offset from midline for 2D shells of slave surface, SOS EQ.0.0: default to 1.0 GT.0.0: scale factor is applied to actual thickness LT.0.0: absolute value is used as the offset	E10.0
41-50	Surface offset from midline for 2D shells of master surface, SOM EQ.0.0: default to 1.0 GT.0.0: scale factor is applied to actual thickness LT.0.0: absolute value is used as the offset	E10.0
51-55	Normal direction flag for 2D shells of slave surface, NDS 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	I5
56-60	Normal direction flag for 2D shells of master surface, NDM 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	I5

If NPSS>0, include cards 3a, 3b, 3c,... as needed to define the slave surface.

Cards 3a,3b,3c,... (8I10)

Columns	Quantity	Format
1-10	Part ID of material to be included in the slave surface	I10
11-20	Part ID of material to be included in the slave surface	I10
21-30	Part ID of material to be included in the slave surface	I10
31-40	etc.	
41-50	.	
51-60	.	
61-70	.	
71-80	.	

If NPMS>0, include cards 4a, 4b, 4c, ... as needed to define the master surface.

Cards 4a,4b,4c,... (8I10)

Columns	Quantity	Format
1-10	Part ID of material to be included in the master surface	I10
11-20	Part ID of material to be included in the master surface	I10
21-30	Part ID of material to be included in the master surface	I10
31-40	etc.	
41-50	.	
51-60	.	
61-70	.	
71-80	.	

For automatic contact type 0 or 1, penetration of 2D shell elements and external faces of 2D continuum elements is prevented by penalty forces. Parts on the slave surface are checked for contact with parts on the master surface. Self contact is checked for any part on both surfaces. If NPSS=0, all parts are checked for contact. If the NPMS=0, the master surface is assumed to be identical to the slave surface.

For automatic contact type 2, penalty forces prevent the flow of slave surface material (the continuum) through the master surfaces. Flow of the continuum tangent to the surface is permitted. Only 2D solid parts are permitted in the slave part set. Both 2D solid and 2D shell parts are permitted in the master surface.

By default, the true thickness of 2D shell elements is taken into account for type 0 or 1 automatic 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 contact type 2.

By default, the normal direction of 2D shell elements is evaluated automatically for type type 0 or 1 automatic 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.

For type 2 automatic 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.

When using automatic contact type 2, there is no need to mesh the continuum around the structure because contact is not with continuum nodes but with material in the interior of the solid elements. The algorithm works well for Eulerian or ALE fluids since the structure does not interfere with remeshing. However, a structure will usually not penetrate the surface of an ALE fluid 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 fluid is not a problem.

For automatic contact types 0, 1, and 2, eroding materials are treated by default. At present, subcycling is not possible.

58. Lumped Parameter Control Volumes

Skip this section if the lumped parameter control volume flag, ITHCNV, on control card 11, columns. 25, is set to zero.

Card 1 (2I5)

Columns	Quantity	Format
1-5	Input the number of control volumes, NCNV	I5
6-10	Number of interacting control volumes, NICV	I5

Control Volume Definition Cards (2I5,I1,I4,I5,5E10.0), or (I8,I5,I1,I4,I5,10X,4E10.0,E7.0) for LARGE option or (I10,I5,I1,I4,I10,/,5E10.0) for MLARG option.

Input NCNV control volume definition card sets. Each set consists of a control volume definition card and the cards, if any, for the airbag sensor which is used to initiate the inflator.

For the standard and **LARGE** input use the following format.

Columns		Quantity	Format
1-5	(1-8)	Number of polygons defining surface, NPOLY GT.0: manual definition of surface, specify NPOLY polygons below to define surface. LT.0: automatic definiton of surface, specify |NPOLY| material ID's below to define surface.	I5 (I8)
6-10	(9-13)	Thermodynamic relationship number EQ.1: simple pressure volume relationship EQ.2: equation-of-state (not implemented) EQ.3: simple airbag model EQ.4: adiabatic gas with stress initialization EQ.5: Wang and Nefske formulation EQ.6: Wang and Nefske formulation with Nusholtz jetting [Nusholtz, et.al., 1990,1991] EQ.7: pressure defined by load curve EQ.8: multiple jet model EQ.9: linear fluid EQ.10: hybrid inflator model	I5 (I5)

Columns		Quantity	Format	
11-11	(14-14)	PFLAG, a flag for reading card 2A for thermodynamic relationships type numbers 5, 6, and 8. EQ.1: Do not read extra pop data (default) EQ.2: Read extra pop input data.	I1	(1)
12-15	(15-18)	IFLAG, for delay times with material input, (NPOLY<0) EQ.1: delay times are not read (default), EQ.2: delay times are read in.	I5	(I5)
16-20	(19-23)	Rigid body material number, *nrb*, is used as sensor to activate the inflator. This can be accomplished by using either LS-DYNA's sensor subroutine (n is given as a negative number) or by a user defined subroutine (n is positive). EQ.-*nrb*: Sensor subroutine flags initiates the inflator. Load curves are offset by initiation time. EQ.0: the control volume is active from time zero EQ. *nrb*: user sensor subroutine flags the start of the inflation. Load curves are offset by initiation time. See Appendix B.	I5	(I5)
31-40	(34-43)	Volume scale factor, V_{sca} (default=1.0)	E10.0	(E10.0)
41-50	(44-53)	Pressure scale factor, P_{sca} (default=1.0)	E10.0	(E10.0)
51-60	(54-63)	Initial filled volume, V_{ini}	E10.0	(E10.0)
61-70	(64-73)	Mass weighted damping factor, D	E10.0	(E10.0)
71-80	(74-80)	Stagnation pressure scale factor, $0 <= \gamma <= 1$	E10.0	(E7.0)

For the **MLARG** option define the following two cards.

Columns	Quantity	Format
1-10	Number of polygons defining surface, NPOLY GT.0: manual definition of surface, specify NPOLY polygons below to define surface. LT.0: automatic definiton of surface, specify |NPOLY| material ID's below to define surface.	I10
11-15	Thermodynamic relationship number EQ.1: simple pressure volume relationship EQ.2: equation-of-state (not implemented) EQ.3: simple airbag model EQ.4: adiabatic gas with stress initialization EQ.5: Wang and Nefske formulation EQ.6: Wang and Nefske formulation with Nusholtz jetting [Nusholtz, et.al., 1990,1991] EQ.7: pressure defined by load curve EQ.8: multiple jet model EQ.9: linear fluid EQ.10: hybrid inflator model	I5
16-16	PFLAG, a flag for reading card 2A for thermodynamic relationships type numbers 5, 6, and 8. EQ.1: Do not read extra pop data (default) EQ.2: Read extra pop input data.	I1
17-20	FLAG, for delay times with material input, (NPOLY<0) EQ.1: delay times are not read (default), EQ.2: delay times are read in.	I4
21-30	Rigid body material number, *nrb*, is used as sensor to activate the inflator. This can be accomplished by using either LS-DYNA's sensor subroutine (n is given as a negative number) or by a user defined subroutine (n is positive). EQ.-*nrb*: Sensor subroutine flags initiates the inflator. Load curves are offset by initiation time. EQ.0: the control volume is active from time zero EQ. *nrb*: user sensor subroutine flags the start of the inflation. Load curves are offset by initiation time. See Appendix B.	I10

Columns	Quantity	Format
1-10	Volume scale factor, V_{sca} (default=1.0)	E10.0
11-20	Pressure scale factor, P_{sca} (default=1.0)	E10.0
21-30	Initial filled volume, V_{ini}	E10.0
31-40	Mass weighted damping factor, D	E10.0
41-50	Stagnation pressure scale factor, $0 <= \gamma <= 1$	E10.0

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:

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

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

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

$$\vec{F}_i = m_i D\left(\vec{V}_i - \vec{V}_{cg}\right)$$

where F_i 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 control volume, and D is the damping factor.

A separate damping formula 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.

Sensor Input to Activate Inflator

Skip this section if nrb=0. If the rigid body material number 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 material model 20-the rigid material. This is important since the default local system is taken as the principle 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. User Defined Sensor Input (*nrb*>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.

User Defined Parameters Card 1 (I5)

Columns	Quantity	Format
1-5	Number of input parameters (not to exceed 25)	I5

If the number of input parameters is non-zero define the input below with 8 parameters per card. Define only the number of cards necessary; i.e. for 9 constants use 2 cards.

User Defined Parameters Card 2 (8E10.0)

Columns	Quantity	Format
1-80	User defined parameters	8E10.0

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

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

Card 1 (5E10.0) Acceleration Activation
(Up to four options may be used)

Columns	Quantity	Format
1-10	Acceleration level in local x-direction to activate inflator. The absolute value of the x-acceleration is used. EQ.0: inactive	E10.0
11-20	Acceleration level in local y-direction to activate inflator The absolute value of the y-acceleration is used. EQ.0: inactive	E10.0
21-30	Acceleration level in local z-direction to activate inflator The absolute value of the z-acceleration is used. EQ.0: inactive	E10.0
31-40	Acceleration magnitude required to activate inflator EQ.0: inactive	E10.0
41-50	Time duration acceration must be exceeded before the inflator activates. This is the cummulative time from the beginning of the calculation, i.e., it is not continuous.	E10.0

Card 2 (4E10.0) Velocity Change Activation
(Up to four options may be used.)

Columns	Quantity	Format
1-10	Velocity change in local x-direction to activate the inflator. The absolute value of the velocity change is used. EQ.0: inactive	E10.0
11-20	Velocity change in local y-direction to activate the inflator. The absolute value of the velocity change is used. EQ.0: inactive	E10.0
21-30	Velocity change in local z-direction to activate inflator The absolute value of the velocity change is used. EQ.0: inactive	E10.0
31-40	Velocity change magnitude required to activate inflator EQ.0: inactive	E10.0

Card 3 (4E10.0) Displacement Activation
(Up to four options my be used.)

Columns	Quantity	Format
1-10	Displacement increment in local x-direction to activate inflator. The absolute value of the x-displacement is used. EQ.0: inactive	E10.0
11-20	Displacement increment in local y-direction to activate inflator The absolute value of the y-displacement is used. EQ.0: inactive	E10.0
21-30	Displacement increment in local z-direction to activate inflator The absolute value of the z-displacement is used. EQ.0: inactive	E10.0
31-40	Displacement magnitude required to activate inflator EQ.0: inactive	E10.0

For each control volume who's surface is defined automatically repeat the following card sets. The absolute value of NPOLY is the number of materials to input.

Control Volume Boundary Materials
[(16I5) if IFLAG=1, or (I5,2E10.0) if IFLAG=2]
[(8I10) if IFLAG=1, or (I10,2E10.0) if IFLAG=2] for MLARG option
(Cards NCNV+2,......)

If , IFLAG=1, the following input formats are used (define |NPOLY| materials in each set):

Columns		Quantity	Format
1-5	(1-10)	Material number 1	I5 (I10)
6-10	(11-20)	Material number 2	I5 (I10)
	. . .		

or else, if IFLAG=2, the input becomes (input NCNV card sets with |NPOLY| cards each):

Columns		Quantity	Format
1-5	(1-10)	Material number	I5 (I10)
6-15	(11-20)	Time delay, T1, before pressure begins to act.	E10.0 (E10.0)
16-25	(21-30)	Time delay, T2, before full pressure is applied, (default T2=T1)	E10.0 (E10.0)

For each control volume who's surface is defined manually repeat the following card sets. NPOLY is the number of segments to input.

Control Volume Boundary Segments
(10X,4I5), or (5X,4I8) for LARGE option
(Cards NCNV+2,......)

Columns		Quantity	Format	
11-15	(6-13)	Nodal point n_1 (see Figure 58.1)	I5	(I8)
16-20	(14-21)	Nodal point n_2	I5	(I8)
21-25	(22-29)	Nodal point n_3	I5	(I8)
26-30	(30-37)	Nodal point n_4	I5	(I8)
31-40	(38-47)	Time delay, T1, before pressure begins to act.	E10.0	E10.0
16-25	(48-57)	Time delay, T2, before full pressure is applied, (default T2=T1)	E10.0	E10.0

Thermodynamic Relationship Input Cards

Repeat the input which follows for each control volume; i.e., define one set of cards for NCNV control volumes.

Type 1: Simple Pressure Volume Relationship (3E10.0)

Columns	Quantity	Format
1-10	C, constant. Define if a load curve ID is not specified.	E10.0
11-20	β, scale factor. Define if a load curve ID is not specified.	E10.0
21-30	Optional load curve ID defining pressure versus relative volume.	E10.0

The pressure, p, relative volume, V, relationship is given by:

$$p = \beta \frac{C}{V}.$$

If β and C are zero, the pressure is interpolated from the load curve. The pressure is then a function of the ratio of current volume to the initial volume. The constant, C, 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.

Type 2: Equation-of-State (not implemented)

Type 3: (default) Simple Airbag Model (3E10.0,I5,4E10.0,I5)

Columns	Quantity	Format
1-10	Heat capacity at constant volume, c_v	E10.0
11-20	Heat capacity at constant pressure, c_p	E10.0
21-30	Temperature of input gas, T	E10.0
31-35	Load curve specifying input mass flow rate	I5
36-45	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	E10.0
46-55	Exit area, A. GE.0.0: A is the exit area and is constant in time. LT.0.0: $\lvert A\rvert$ is the load curve number defining the exit area as a function of absolute pressure	E10.0
56-65	Ambient pressure, p_e	E10.0
66-75	Ambient density, ρ	E10.0
76-80	Optional load curve ID for output mass flow rate versus gage bag pressure.	I5

<u>Optional</u> Card Insert Here if Applicable (5E10.0)

Define this card <u>if and only if</u> c_v defined on card 1 is zero.

Columns	Quantity	Format
1-10	Ambient temperature.	E10.0
11-20	*a*, first heat capacity coefficient of inflator gas (e.g., Joules/mole/^{o}K)	E10.0
21-30	*b*, second heat capacity coefficient of inflator gas (e.g., Joules/mole/$^o K^2$)	E10.0
31-40	*MW*, Molecular weight of inflator gas (e.g., Kg/mole),	E10.0

Columns	Quantity	Format
41-50	*R*, Universal gas constant.of inflator gas (e.g., 8.314 Joules/mole/^{o}K)	E10.0

With this option the constant-pressure specific heat is given by:

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

and the constant-volume specific heat is then found:

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

Type 4: Perfect Gas with Stress Initialization (E10.0,I5,4E10.0)

Columns	Quantity	Format
1-10	Pressure scale factor (default = 1.0)	E10.0
11-15	Load curve for preload flag	I5
16-25	Ratio of specific heats	E10.0
26-35	Initial pressure (gauge)	E10.0
36-45	Ambient pressure	E10.0
46-55	Initial density of gas	E10.0

Type 5: Wang-Nefske Inflator Model (3E10.0,I5,4E10.0,I5)

Columns	Quantity	Format
1-10	Heat capacity at constant volume, C_v	E10.0
11-20	Heat capacity at constant pressure, C_p	E10.0
21-30	Temperature of input gas, T	E10.0
31-35	Load curve specifying input mass flow rate or tank pressure versus time. If the tank volume in Columns 31-40 on Card 2 below is nonzero the curve is assumed to specify the latter, i.e., tank gauge pressure versus time.	I5
36-45	C23, vent orifice coefficient which applies to exit hole LT.0.0: |C23| is the load curve number defining vent orifice coefficient as a function of *time*	E10.0
46-55	A23, vent orifice area which applies to exit hole LT.0.0: |A23| is the load curve number defining vent orifice area as a function of absolute *pressure*	E10.0

Define C'23 and A'23 in columns 56-75 below if FLC and FAC are zero in the definition of the fabric material.

Columns	Quantity	Format
56-65	C′23, orifice coefficient for leakage (fabric porosity) LT.0.0: |C'23| is the load curve number defining orifice coefficient for leakage as a function of *time*	E10.0
66-75	A′23, area for leakage (fabric porosity) LT.0.0: |A'23| is the load curve number defining the fabric porosity as a function of absolute *pressure*	E10.0

Define OPT in columns 56-65 below and leave columns 66-75 blank if FLC and FAC are nonzero in the definition of the fabric material

Columns	Quantity	Format
56-65	OPT, orifice coefficient for leakage (fabric porosity) EQ. 997: Wang-Nefske formulas for venting through an orifice are used. Blockage is not considered. EQ.-997: Wang-Nefske formulas for venting through an orifice are used. Blockage of venting area due to contact is considered. EQ. 998: Leakage formulas of Graefe, Krummheuer, and Siejak [1990] are used. Blockage is not considered. EQ.-998: Leakage formulas of Graefe, Krummheuer, and Siejak [1990] are used. Blockage of venting area due to contact is considered.	E10.0

Columns	Quantity	Format
	EQ. 999: Leakage formulas based on flow through a porous media are used. Blockage is not considered. EQ.-999: Leakage formulas based on flow through a porous media are used. Blockage of venting area due to contact is considered.	
66-75	Blank	
76-80	Optional load curve number defining temperature of input gas versus time. This overides columns 21-30.	I5

If OPT is defined then for |OPT| set to 997 we have for the mass flow rate out of the bag, $\dot{m}_{out}$ is given by:

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

where

$$Q = \frac{p_e}{p} \;\; if \;\; Q \le Q_{crit} \quad then \; Q = Q_{crit} \quad where \; Q_{crit} = \left(\frac{2}{k+1}\right)^{k/k-1}$$

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

p_e = external pressure

p = internal (absolute) pressure

ρ = density of airbag gas

c_v = the specific heat at constant volume

c_p = the specific heat at constant pressure

nairmats = number of fabrics used in the airbag

$Area_n$ = current unblocked area of fabric number n.

for |OPT| set to 998:

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

and for |OPT| set to 999:

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

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 above.

<u>Optional</u> Card Insert Here if Applicable (5E10.0)

Define this card <u>if and only if</u> c_V defined on card 1 is zero.

Columns	Quantity	Format
1-10	Ambient temperature.	E10.0
11-20	*a*, first heat capacity coefficient of inflator gas (e.g., Joules/mole/^{o}K)	E10.0
21-30	*b*, second heat capacity coefficient of inflator gas (e.g., Joules/mole/^{o}K^2)	E10.0
31-40	*MW*, Molecular weight of inflator gas (e.g., Kg/mole),	E10.0
41-50	*R*, Universal gas constant.of inflator gas (e.g., 8.314 Joules/mole/^{o}K)	E10.0

With this option the constant-pressure specific heat is given by:

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

and the constant-volume specific heat is then found:

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

Card 2 (4E10.0,I5,2E10.0,I5,E10.0)

Columns	Quantity	Format
1-10	Ambient pressure, p_e	E10.0
11-20	Ambient density, ρ	E10.0
21-30	g_c, gravitational conversion constant (default=1.0)	E10.0
31-40	Optional tank volume. See Columns 31-35 on Card 1.	E10.0
41-45	Optional curve for exit flow rate versus pressure	I5
46-55	Initial gauge overpressure in bag. Generally, zero.	E10.0
56-65	Pop pressure for initiating exit flow. Generally, zero.	E10.0
66-70	Load curve for time rate of change of temperature (dT/dt) versus time.	I5
71-80	Initial airbag temperature. (optional, generally not defined)	E10.0

If the inflator is modeled, i.e., the load curve number in columns 31-35 of card 1 is zero, then define the following card. If not, omit the following card.

Card 3 (6E10.0)

Columns	Quantity	Format
1-10	Inflator orifice coefficient	E10.0
11-20	Inflator orifice area	E10.0
21-30	Inflator volume	E10.0
31-40	Inflator density	E10.0
41-50	Inflator temperature	E10.0
51-55	Load curve defining burn fraction versus time	E10.0

Optional Card (7E10.0,I10)

Include the following card if and only if PFLAG=2 on the first control card. Use this card to specify additional criteria for initiating exit flow from the airbag.

Columns	Quantity	Format
1-10	Time delay for initiating exit flow after pop pressure is reached (default=0.0)	E10.0
11-20	Pop acceleration magnitude in local x-direction (default=0.0) EQ. 0.0: Inactive	E10.0
21-30	Pop acceleration magnitude in local y-direction (default=0.0) EQ. 0.0: Inactive	E10.0
31-40	Pop acceleration magnitude in local z-direction (default=0.0) EQ. 0.0: Inactive	E10.0
41-50	Pop acceleration magnitude (default=0.0) EQ. 0.0: Inactive	E10.0
51-60	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 (default=0.0).	E10.0
61-70	Inflator temperature	E10.0
71-80	Part ID of rigid body for checking accelerations against pop accelerations.	E10.0

Type 6: Wang-Nefske Inflator with Jet Model (3E10.0,I5,4E10.0,I5)

Columns	Quantity	Format
1-10	Heat capacity at constant volume, C_v	E10.0
11-20	Heat capacity at constant pressure, C_p	E10.0
21-30	Temperature of input gas, *T*	E10.0
31-35	Load curve specifying input mass flow rate or tank pressure versus time. If the tank volume in Columns 31-40 on Card 2 below is nonzero the curve is assumed to specify the latter, i.e., tank gauge pressure versus time.	I5
36-45	C23, vent orifice coefficient which applies to exit hole LT.0.0: |C23| is the load curve number defining vent orifice coefficient as a function of *time*	E10.0
46-55	A23, vent orifice area which applies to exit hole LT.0.0: |A23| is the load curve number defining vent orifice area as a function of absolute *pressure*	E10.0

Define C'23 and A'23 in columns 56-75 below if FLC and FAC are zero in the definition of the fabric material.

Columns	Quantity	Format
56-65	C′23, orifice coefficient for leakage (fabric porosity) LT.0.0: |C'23| is the load curve number defining orifice coefficient for leakage as a function of *time*	E10.0
66-75	A′23, area for leakage (fabric porosity) LT.0.0: |A'23| is the load curve number defining the fabric porosity as a function of absolute *pressure*	E10.0

Define OPT in columns 56-65 below and leave columns 66-75 blank if FLC and FAC are nonzero in the definition of the fabric material

Columns	Quantity	Format
56-65	OPT, orifice coefficient for leakage (fabric porosity) EQ. 997: Wang-Nefske formulas for venting through an orifice are used. Blockage is not considered. EQ.-997: Wang-Nefske formulas for venting through an orifice are used. Blockage of venting area due to contact is considered. EQ. 998: Leakage formulas of Graefe, Krummheuer, and Siejak [1990] are used. Blockage is not considered. EQ.-998: Leakage formulas of Graefe, Krummheuer, and Siejak [1990] are used. Blockage of venting area due to contact is considered. EQ. 999: Leakage formulas based on flow through a porous media are used. Blockage is not considered.	E10.0

Columns	Quantity	Format
	EQ.-999: Leakage formulas based on flow through a porous media are used. Blockage of venting area due to contact is considered.	
66-75	Blank	
76-80	Optional load curve number defining temperature of input gas versus time. This overides columns 21-30.	I5

Optional Card Insert Here if Applicable (5E10.0)

Define this card if and only if c_v defined on card 1 is zero.

Columns	Quantity	Format
1-10	Ambient temperature.	E10.0
11-20	*a*, first heat capacity coefficient of inflator gas (e.g., Joules/mole/^{o}K)	E10.0
21-30	*b*, second heat capacity coefficient of inflator gas (e.g., Joules/mole/^{o}K^2)	E10.0
31-40	*R*, Universal gas constant.of inflator gas (e.g., 8.314 Joules/mole/^{o}K)	E10.0
41-45	*MW*, Molecular weight of inflator gas (e.g., Kg/mole),	E10.0

With this option the constant-pressure specific heat is given by:

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

and the constant-volume specific heat is then found:

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

Card 2 (4E10.0,I5,2E10.0,I10)

Columns	Quantity	Format
1-10	Ambient pressure, p_e	E10.0
11-20	Ambient density, ρ	E10.0
21-30	g_c, gravitational conversion constant (default=1.0)	E10.0
31-40	Optional tank volume. See Columns 31-35 on Card 1.	E10.0
41-45	Optional curve for exit flow rate versus pressure	I5
46-55	Initial overpressure in bag (gauge). Generally, zero.	E10.0
56-65	Pop pressure for initiating exit flow. Generally, zero.	E10.0
66-75	Optional load curve ID defining the knock down pressure scale factor versus time. 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.	E10.0

If the inflator is modeled, i.e., the load curve number in columns 31-35 of card 1 is zero, then define the following card. If not, omit the following card.

Card 3 (6E10.0)

Columns	Quantity	Format
1-10	Inflator orifice coefficient	E10.0
11-20	Inflator orifice area	E10.0
21-30	Inflator volume	E10.0
31-40	Inflator density	E10.0
41-50	Inflator temperature	E10.0
51-55	Load curve defining burn fraction versus time	E10.0

Optional Card (7E10.0,I10)

Include the following card if and only if PFLAG=2 on the first control card. Use this card to specify additional criteria for initiating exit flow from the airbag.

Columns	Quantity	Format
1-10	Time delay for initiating exit flow after pop pressure is reached (default=0.0)	E10.0
11-20	Pop acceleration magnitude in local x-direction (default=0.0) EQ. 0.0: Inactive	E10.0
21-30	Pop acceleration magnitude in local y-direction (default=0.0) EQ. 0.0: Inactive	E10.0
31-40	Pop acceleration magnitude in local z-direction (default=0.0) EQ. 0.0: Inactive	E10.0
41-50	Pop acceleration magnitude (default=0.0) EQ. 0.0: Inactive	E10.0
51-60	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 (default=0.0).	E10.0
61-70	Inflator temperature	E10.0
71-80	Part ID of rigid body for checking accelerations against pop accelerations.	E10.0

Card 4 (8E10.0)

Columns	Quantity	Format
1-10	x-coordinate of jet focal point EQ.0.0: node defintion is used on card 5 below.	E10.0
11-20	y-coordinate of jet focal point EQ.0.0: node defintion is used on card 5 below.	E10.0
21-30	z-coordinate of jet focal point EQ.0.0: node defintion is used on card 5 below.	E10.0
31-40	x-coordinate of jet vector head EQ.0.0: node defintion is used on card 5 below.	E10.0

Columns	Quantity	Format
41-50	y-coordinate of jet vector head EQ.0.0: node defintion is used on card 5 below.	E10.0
51-60	z-coordinate of jet vector head EQ.0.0: node defintion is used on card 5 below.	E10.0
61-70	Cone angle, α, defined in radians. LT.0.0: $\|\alpha\|$ is the load curve ID defining cone angle as a function of *time*	E10.0
71-80	Efficiency factor, β LT.0.0: $\|\beta\|$ is the load curve ID defining the efficiency factor as a function of *time*	E10.0

Card 5 (3E10.0,I5)

Columns	Quantity	Format
	For coordinate definition use the following input in columns 1-30:	
1-10	x-coordinate of secondary jet focal point, passenger side bag The coordinate option is okay if the bag is fixed in space. If the coordinate of the secondary point is (0,0,0) then a conical jet (drivers side airbag) is assumed.	E10.0
11-20	y-coordinate of secondary jet focal point	E10.0
21-30	z-coordinate of secondary jet focal point	E10.0
	For node definition use the following input in columns 1-30:	
1-10	Node ID for node located at focal point. The nodal point option is recommended when the location of the airbag changes as a function of time. If the node ID of the secondary point is 0 then a conical jet (drivers side airbag) is assumed.	E10.0
11-20	Node ID for node along the axis of the jet	E10.0
21-30	Optional node ID for node located at secondary jet focal point.	E10.0
31-35	Number of materials to be included for jet interaction. If zero, then all airbag materials are included.	I5

Read the following card if and only if the number of materials in columns 31-35 above is nonzero.

Card 6 (8I10) [Use as many cards as necessary]

Columns	Quantity	Format
1-10	First material to be included	I10
11-20	Second material	I10
21-30	Third material	I10
31-40	Fourth material	I10
.		
.		
.		
71-80	Eighth material	I10

Type 7: Pressure Defined by Load Curve (2E10.0)

Columns	Quantity	Format
1-10	Time at which pressure is applied. The load curve is offset by this amount.	E10.0
11-20	Load curve ID defining pressure versus time.	E10.0
21-30	Initial density of gas (ignored if load curve ID >0)	E10.0
31-40	Ambient pressure (ignored if load curve ID >0)	E10.0
41-50	Initial gauge pressure (ignored if load curve ID >0)	E10.0
51-60	Gas Temperature (ignored if load curve ID >0)	E10.0
61-70	Absolute zero on temperature scale (ignored if load curve ID >0)	E10.0

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

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

The pressure is uniform throughout the control volume.

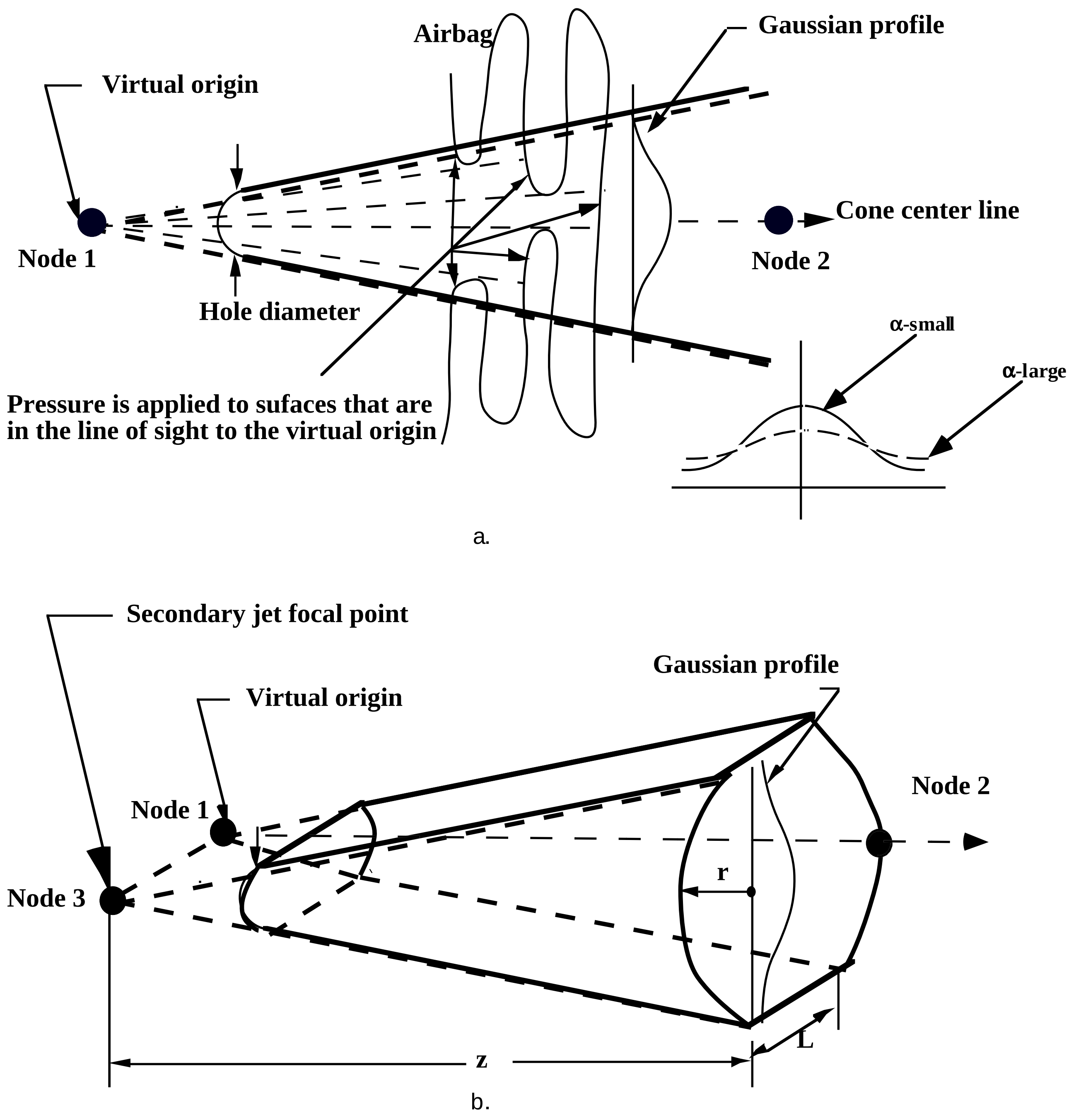


Figure 58.1 Jetting configuration for (a.) driver's side airbag (pressure applied only if centroid of surface is in line-of-sight) and (b.) the passenger’s side bag.

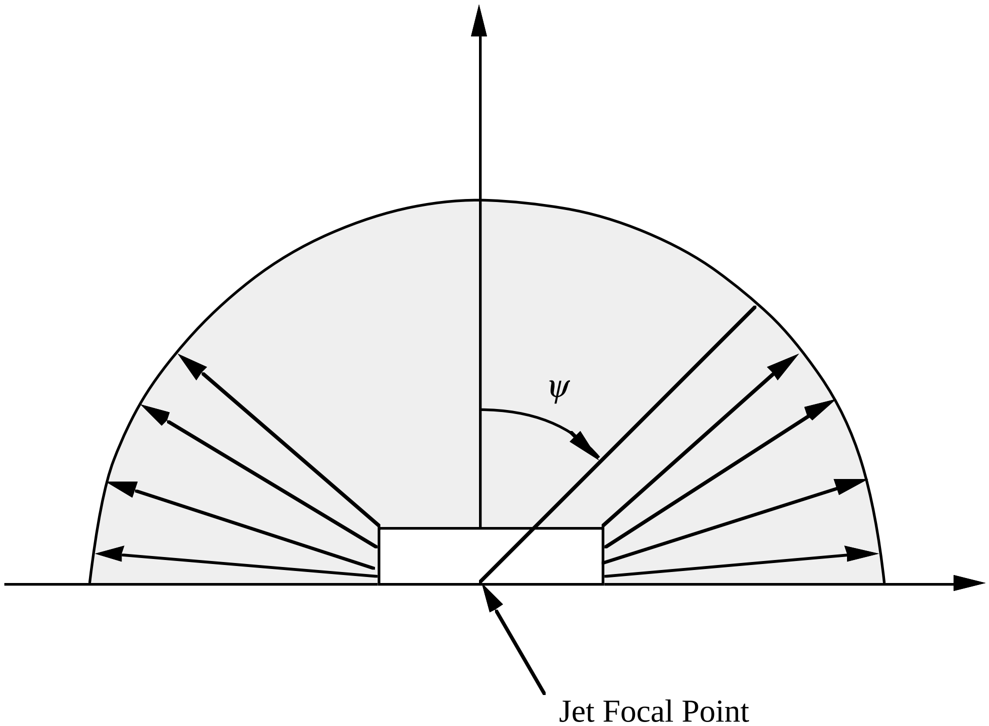


Figure 58.2. Multiple jet model for the drivers side airbag.

Type 8: Wang-Nefske Inflator/Multiple Jet Model (3E10.0,I5,4E10.0,I5)

Columns	Quantity	Format
1-10	Heat capacity at constant volume, C_v	E10.0
11-20	Heat capacity at constant pressure, C_p	E10.0
21-30	Temperature of input gas, T	E10.0
31-35	Load curve specifying input mass flow rate or tank pressure versus time. If the tank volume in Columns 31-40 on Card 2 below is nonzero the curve is assumed to specify the latter, i.e., tank pressure versus time.	I5
36-45	C23, vent orifice coefficient which applies to exit hole LT.0.0: |C23| is the load curve number defining vent orifice coefficient as a function of *time*	E10.0
46-55	A23, vent orifice area which applies to exit hole LT.0.0: |A23| is the load curve number defining vent orifice area as a function of <u>absolute</u> *pressure*	E10.0

Define C’23 and A’23 in columns 56-75 below if FLC and FAC are zero in the definition of the fabric material.

Columns	Quantity	Format
56-65	C′23, orifice coefficient for leakage (fabric porosity) LT.0.0: |C’23| is the load curve number defining orifice coefficient for leakage as a function of *time*	E10.0
66-75	A′23, area for leakage (fabric porosity) LT.0.0: |A’23| is the load curve number defining the fabric porosity as a function of <u>absolute</u> *pressure*	E10.0

Define OPT in columns 56-65 below and leave columns 66-75 blank if FLC and FAC are nonzero in the definition of the fabric material

Columns	Quantity	Format
56-65	OPT, orifice coefficient for leakage (fabric porosity) EQ. 997: Wang-Nefske formulas for venting through an orifice are used. Blockage is not considered. EQ.-997: Wang-Nefske formulas for venting through an orifice are used. Blockage of venting area due to contact is considered. EQ. 998: Leakage formulas of Graefe, Krummheuer, and Siejak [1990] are used. Blockage is not considered. EQ.-998: Leakage formulas of Graefe, Krummheuer, and Siejak [1990] are used. Blockage of venting area due to contact is considered.	E10.0

Columns	Quantity	Format
	EQ. 999: Leakage formulas based on flow through a porous media are used. Blockage is not considered. EQ.-999: Leakage formulas based on flow through a porous media are used. Blockage of venting area due to contact is considered.	
66-75	Blank	
76-80	Optional load curve number defining temperature of input gas versus time. This overides columns 21-30.	I5

Optional Card Inserted Here if Applicable (5E10.0)

Define this card if and only if c_v defined on card 1 is zero.

Columns	Quantity	Format
1-10	Ambient temperature.	E10.0
11-20	a, first heat capacity coefficient of inflator gas (e.g., Joules/mole/^{o}K)	E10.0
21-30	b, second heat capacity coefficient of inflator gas (e.g., Joules/mole/^{o}K^2)	E10.0
31-40	R, Universal gas constant.of inflator gas (e.g., 8.314 Joules/mole/^{o}K)	E10.0
41-45	MW, Molecular weight of inflator gas (e.g., Kg/mole),	E10.0

With this option the constant-pressure specific heat is given by:

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

and the constant-volume specific heat is then found:

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

Card 2 (4E10.0,I5,2E10.0)

Columns	Quantity	Format
1-10	Ambient pressure, p_e	E10.0
11-20	Ambient density, ρ	E10.0
21-30	g_c, gravitational conversion constant (default=1.0)	E10.0
31-40	Optional tank volume. See Columns 31-35 on Card 1.	E10.0
41-45	Optional curve for exit flow rate versus pressure	I5
46-55	Initial overpressure in bag (gauge). Generally, zero.	E10.0
56-65	Pop pressure for initiating exit flow. Generally, zero.	E10.0
66-75	Optional load curve ID defining the knock down pressure scale factor versus time. 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.	E10.0

If the inflator is modeled, i.e., the load curve number in columns 31-35 of card 1 is zero, then define the following card. If not, omit the following card.

Card 3 (6E10.0)

Columns	Quantity	Format
1-10	Inflator orifice coefficient	E10.0
11-20	Inflator orifice area	E10.0
21-30	Inflator volume	E10.0
31-40	Inflator density	E10.0
41-50	Inflator temperature	E10.0
51-55	Load curve defining burn fraction versus time	E10.0

Optional Card (7E10.0,I10)

Include the following card if and only if PFLAG=2 on the first control card. Use this card to specify additional criteria for initiating exit flow from the airbag.

Columns	Quantity	Format
1-10	Time delay for initiating exit flow after pop pressure is reached (default=0.0)	E10.0
11-20	Pop acceleration magnitude in local x-direction (default=0.0) EQ. 0.0: Inactive	E10.0
21-30	Pop acceleration magnitude in local y-direction (default=0.0) EQ. 0.0: Inactive	E10.0
31-40	Pop acceleration magnitude in local z-direction (default=0.0) EQ. 0.0: Inactive	E10.0
41-50	Pop acceleration magnitude (default=0.0) EQ. 0.0: Inactive	E10.0
51-60	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 (default=0.0).	E10.0
61-70	Inflator temperature	E10.0
71-80	Part ID of rigid body for checking accelerations against pop accelerations.	E10.0

Card 4 (8E10.0)

Columns	Quantity	Format
1-10	x-coordinate of jet focal point EQ.0.0: node defintion is used on card 5 below.	E10.0
11-20	y-coordinate of jet focal point EQ.0.0: node defintion is used on card 5 below.	E10.0
21-30	z-coordinate of jet focal point EQ.0.0: node defintion is used on card 5 below.	E10.0
31-40	x-coordinate of jet vector head EQ.0.0: node defintion is used on card 5 below.	E10.0

Columns	Quantity	Format
41-50	y-coordinate of jet vector head EQ.0.0: node defintion is used on card 5 below.	E10.0
51-60	z-coordinate of jet vector head EQ.0.0: node defintion is used on card 5 below.	E10.0
61-70	Load curve ID giving jet relative velocity distribution	E10.0
71-80	Efficiency factor, β	E10.0

Card 5 (3E10.0,I5,E10.0)

Columns	Quantity	Format
For coordinate definition use the following input in columns 1-30:		
1-10	x-coordinate of secondary jet focal point, passenger side bag The coordinate option is okay if the bag is fixed in space. If the coordinate of the secondary point is (0,0,0) then a conical jet (drivers side airbag) is assumed.	E10.0
11-20	y-coordinate of secondary jet focal point	E10.0
21-30	z-coordinate of secondary jet focal point	E10.0
For node definition use the following input in columns 1-30:		
1-10	Node ID for node located at focal point. The nodal point option is recommended when the location of the airbag changes as a function of time. If the node ID of the secondary point is 0 then a conical jet (drivers side airbag) is assumed.	E10.0
11-20	Node ID for node along the axis of the jet	E10.0
21-30	Optional node ID for node located at secondary jet focal point.	E10.0
31-35	Number of materials to be included for jet interaction. If zero, then all airbag materials are included.	I5
36-45	Cutoff angle in degrees. The jet velocity is set to zero for angles greater than the cutoff.	E10.0

Read the following card if and only if the number of materials in columns 31-35 above is nonzero.

Card 7 (8I10) [Use as many cards as necessary]

Columns	Quantity	Format
1-10	First material to be included	I10
11-20	Second material	I10
21-30	Third material	I10
31-40	Fourth material	I10
.		
.		
.		
71-80	Eighth material	I10

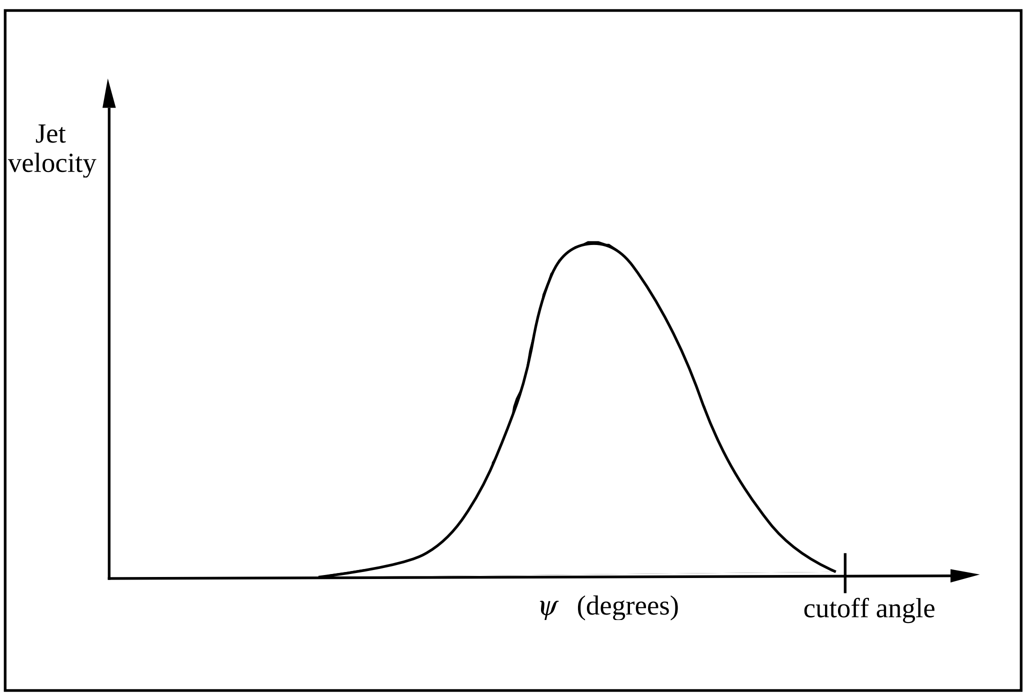


Figure 58.3. Normalized jet velocity versus angle for multiple jet drivers side airbag.

Type 9: Linear Fluid (2E10.0, 6I5)

Columns	Quantity	Format
1-10	K, bulk modulus	E10.0
11-20	ρ, density	E10.0
21-25	$F(t)$ input flow curve defining mass per unit time as a function of time, see *DEFINE_CURVE.	I5
26-30	$G(t)$, output flow curve defining mass per unit time as a function of time. This load curve is optional.	I5
31-35	$H(p)$, output flow curve defining mass per unit time as a function of pressure. This load curve is optional.	I5
36-40	$L(t)$, added pressure as a function of time. This load curve is optional.	I5
41-45	Curve ID defining the bulk modulus as a function of time. This load curve is optional, but if defined, the contant, BULK, is not used.	I5
46-50	Optional Curve ID defining pressure as function of time.	I5

Pressure is determined from the following equation if the optional pressure versus time curve is not defined:

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

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 $P(0) = 0$.

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.

Please note the sign convention used in the the equation for $M(t)$. The mass flow rate should always be defined as positive since the output flow is substracted as shown.

Type 10: Hybrid Inflator

Card 1 (4E10.0)

Columns	Quantity	Format
1-10	Atmospheric temperature	E10.0
11-20	Atmospheric pressure	E10.0
21-30	Atmospheric density	E10.0
31-40	Universal molar gas constant	E10.0
41-50	Gravitational conversion constant EQ.0 : default set to 1.	E10.0
51-60	flag for jetting EQ.0: no jetting (default) EQ.1: include jetting (include cards 4, 5 & 6 below)	I10

Card 2 (4E10.0,2I10)

Columns	Quantity	Format
1-10	C23, vent orifice coefficient which applies to exit hole LT.0.0: |C23| is the load curve number defining vent orifice coefficient as a function of *time*	E10.0
11-20	A23, vent orifice area which applies to exit hole LT.0.0: |A23| is the load curve number defining vent orifice area as a function of <u>absolute</u> *pressure*	E10.0

Define C’23 and A’23 in columns 21-40 below if FLC and FAC are zero in the definition of the fabric material.

Columns	Quantity	Format
21-30	C′23, orifice coefficient for leakage (fabric porosity) LT.0.0: |C’23| is the load curve number defining orifice coefficient for leakage as a function of *time*	E10.0
31-40	A′23, area for leakage (fabric porosity) LT.0.0: |A’23| is the load curve number defining the fabric porosity as a function of <u>absolute</u> *pressure*	E10.0

Columns	Quantity	Format

Define OPT in columns 21-30 below and leave columns 31-40 blank if FLC and FAC are nonzero in the definition of the fabric material

Columns	Quantity	Format
21-30	OPT, orifice coefficient for leakage (fabric porosity) EQ. 996: Density formula for venting through an orifice is used. Blockage is not considered. EQ.-996: Density formula for venting through an orifice is used. Blockage of venting area due to contact is considered. EQ. 997: Wang-Nefske formulas for venting through an orifice are used. Blockage is not considered. EQ.-997: Wang-Nefske formulas for venting through an orifice are used. Blockage of venting area due to contact is considered. EQ. 998: Leakage formulas of Graefe, Krummheuer, and Siejak [1990] are used. Blockage is not considered. EQ.-998: Leakage formulas of Graefe, Krummheuer, and Siejak [1990] are used. Blockage of venting area due to contact is considered. EQ. 999: Leakage formulas based on flow through a porous media are used. Blockage is not considered. EQ.-999: Leakage formulas based on flow through a porous media are used. Blockage of venting area due to contact is considered.	E10.0
31-40	Blank	E10.0
41-50	Optional gauge pressure when venting begins	E10.0
51-60	NGAS, number of gas species input, including initial air .	I10

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

Gas Card 1 (2I10,10X,3E10.0)

Columns	Quantity	Format
1-10	Load curve ID for inflator mass flow rate.	I10
11-20	Load curve ID for inflator gas temperature	I10
21-30	blank	10X
31-40	Molecular weight.	E10.0

Columns	Quantity	Format
41-50	Initial mass fraction of this gas component.	E10.0
51-60	Fraction of additional aspirated mass.	E10.0

Gas Card 2 (E10.0)

Columns	Quantity	Format
1-10	a, coefficient for the molar heat capacity at constant pressure.	E10.0
11-20	b, coefficient for the molar heat capacity at constant pressure.	E10.0
21-30	c, coefficient for the molar heat capacity at constant pressure.	E10.0

Card 4 (8E10.0)

Columns	Quantity	Format
1-10	x-coordinate of jet focal point EQ.0.0: node defintion is used on card 5 below.	E10.0
11-20	y-coordinate of jet focal point EQ.0.0: node defintion is used on card 5 below.	E10.0
21-30	z-coordinate of jet focal point EQ.0.0: node defintion is used on card 5 below.	E10.0
31-40	x-coordinate of jet vector head EQ.0.0: node defintion is used on card 5 below.	E10.0
41-50	y-coordinate of jet vector head EQ.0.0: node defintion is used on card 5 below.	E10.0
51-60	z-coordinate of jet vector head EQ.0.0: node defintion is used on card 5 below.	E10.0
61-70	Cone angle, α, defined in radians. LT.0.0: $\|\alpha\|$ is the load curve ID defining cone angle as a function of *time*	E10.0
71-80	Efficiency factor, β LT.0.0: $\|\beta\|$ is the load curve ID defining the efficiency factor as a function of *time*	E10.0

Card 5 (3E10.0,I5)

Columns	Quantity	Format
	For coordinate definition use the following input in columns 1-30:	
1-10	x-coordinate of secondary jet focal point, passenger side bag The coordinate option is okay if the bag is fixed in space. If the coordinate of the secondary point is (0,0,0) then a conical jet (drivers side airbag) is assumed.	E10.0
11-20	y-coordinate of secondary jet focal point	E10.0
21-30	z-coordinate of secondary jet focal point	E10.0

Columns	Quantity	Format
For node definition use the following input in columns 1-30:		
1-10	Node ID for node located at focal point. The nodal point option is recommended when the location of the airbag changes as a function of time. If the node ID of the secondary point is 0 then a conical jet (drivers side airbag) is assumed.	E10.0
11-20	Node ID for node along the axis of the jet	E10.0
21-30	Optional node ID for node located at secondary jet focal point.	E10.0
31-35	Number of materials to be included for jet interaction. If zero, then all airbag materials are included.	I5

Read the following card if and only if the number of materials in columns 31-35 above is nonzero.

Card 6 (8I10) [Use as many cards as necessary]

Columns	Quantity	Format
1-10	First material to be included	I10
11-20	Second material	I10
21-30	Third material	I10
31-40	Fourth material	I10
.		
.		
.		
71-80	Eighth material	I10

Define NICV Control Volume Interactions
(5X,2I5,2E10.0,2I10,I5)

This card defines two connected control volumes which vent into each other.

Columns	Quantity	Format
1-5	Blank	5X
6-10	First control Volume, ID	I5
11-15	Second control Volume, ID	I5
16-25	AREA, orifice area between connected bags: LT.0.0: |AREA| is the load curve ID defining orifice area as a function of absolute *pressure* EQ.0.0: AREA is taken as the surface area of the part ID defined below.	E10.0
26-35	SF, shape factor: LT.0.0: |SF| is the load curve ID defining vent orifice coefficient as a function of *relative time*	E10.0
36-45	Optional part ID of the partition between the interacting control volumes. AREA is based on this part ID.	I10
46-55	Load curve ID defining mass flow rate versus pressure difference. If this is defined then AREA, SF and part ID are ignored.	I10
56-60	IFLOW, flow direction: LT.0.0: One way flow from AB1 to AB2 only. EQ.0.0: Two way flow between AB1 and AB2 GT.0.0: One way flow from AB2 to AB1 only.	I5

The control volume ID's correspond to their order in the preceeding input. The ID of the first volume defined is 1, the second is 2, and so on to the last which has an ID of NCNV.

All input options are valid for control volume types 3, 5, 6, 8, and 10. Control volumes must contain the same gas. The flow between bags is governed by formulas which are similar to those of Wang-Nefske, except that choked flow is currently ignored. The load curve defining mass flow rate versus pressure difference may also be used with control volume types 7 and 9

59. Geometric Contact Entities

Geometric contact entities treat the impact between a deformable body defined as a list of slave nodes or shell materials 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 maximized within the constraint of the Courant criterion. This section is required if NGENT (Control Card 5, column 30) is nonzero. Define NGENT consecutive card sets which includes Cards 1 through 4 for each geometric contact entity which is defined. PLEASE NOTE THAT IN THE 910 VERSION OF LS-DYNA ONE ADDITIONAL CARD IS REQUIRED TO START THIS SECTION GIVING THE NUMBER OF CONTACT ENTITIES IN AN I5 FIELD.

Card 1 of Card Set 1 (4I5,3E10.0,I5,2E10.0,I5)
or (I10), (5X,3I5,3E10.0,I5,2E10.0,I5) if MLARG option.

Columns	Quantity	Format
	Card 1 if MLARG is active.	
1-10	Material number of rigid body to which geometric entity is attached	I10
	Card 2 if MLARG is active or Card 1 if MLARG is inactive	
1-5	Material number of rigid body to which geometric entity is attached. Leave blank if MLARG option is active.	I5
6-10	Type of geometric entity, IGTYP. For Types 8 and 9 the orientation of the surface segments is arbitrary. EQ.1: infinite plane EQ.2: sphere EQ.3: infinite cylinder EQ.4: hyperellipsoid EQ.5: torus EQ.6: CAL3D/MADYMO Plane EQ.7: CAL3D/MADYMO Ellipsoid EQ.8: VDA surface EQ.9: rigid body finite element mesh (shells only) EQ.10: finite plane EQ.11: load curve defining line	I5
11-15	Type of slaved item EQ.0: nodes	I5

Columns	Quantity	Format
	EQ.2: materials (shell and solid elements only not including VDA surface.)	
16-20	Number of slaved nodes or materials	I5
21-30	Penalty scale factor (default = 1.0)	E10.0
31-40	Damping option EQ..0: no damping GT..0: viscous damping in percent of critical EQ.-n: |n| is the load curve ID giving the damping force versus relative normal velocity	E10.0
41-50	Coulomb friction value	E10.0
51-55	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	I5
56-65	Birth time	E10.0
66-75	Death time (default = 1.0E+20)	E10.0
76-80	Flag for penalty stiffness 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	I5

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 of Card Set 1
(6E10.0)

Columns	Quantity	Format
1-10	x-center, x_c	E10.0
11-20	y-center, y_c	E10.0
21-30	z-center, z_c	E10.0
31-40	x-direction for local axis X′, A_x	E10.0
41-50	y-direction for local axis X′, A_y	E10.0
51-60	z-direction for local axis X′, A_z	E10.0

Card 3 of Card Set 1
(3E10.0)

Columns	Quantity	Format
1-10	x-direction for local axis Y′, B_x	E10.0
11-20	y-direction for local axis Y′, B_y	E10.0
21-30	z-direction for local axis Y′, B_z	E10.0

(x_c, y_c, z_c) positions the local origin of the geometric entity in global coordinates. The entity's local X′-axis is determined by the vector (A_x, A_y, A_z) and the local Y′-axis by the vector (B_x, B_y, B_z).

Cards 2 and 3 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), X′=(A_x, A_y, A_z)=(1,0,0), and Y′=(B_x, B_y, B_z)=(0,1,0).

Card 4 of Card Set 1
(I5,7E10.0)

Columns	Quantity	Format
1-5	In-out flag EQ.0: slave nodes exist outside of the entity EQ.1: slave nodes exist inside the entity	I5
6-15	Entity coefficient g_1 (CAL3D/MADYMO plane or ellipse number)	E10.0
16-25	Entity coefficient g_2	E10.0
26-35	Entity coefficient g_3	E10.0
36-45	Entity coefficient g_4	E10.0
46-55	Entity coefficient g_5	E10.0
56-65	Entity coefficient g_6	E10.0
66-75	Entity coefficient g_7	E10.0

Figure 59.1 shows the definitions of the geometric contact entities. The relationships between the entity coefficients and the Figure 59.1 variables are as follows for IGTYPE's 1-4. IGTYPES 8 and 9 are primarily for metalforming applications, however, type 9 has been used to model airbag containers. (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):

IGTYPE = 1: $g1 = P_x$ $\quad g4 = Q_x$
$g2 = P_y$ $\quad g5 = Q_y$
$g3 = P_z$ $\quad g6 = Q_z$

IGTYPE = 2: $g1 = P_x$ $\quad g4 = \mathrm{r}$
$g2 = P_y$
$g3 = P_z$

IGTYPE = 3: $g1 = P_x$ $\quad g4 = Q_x$
$g2 = P_y$ $\quad g5 = Q_y$
$g3 = P_z$ $\quad g6 = Q_z$
$g7 = r$

IGTYPE = 4: $g1 = P_x$ $g4$ = a

$g2 = P_y$ $g5$ = b

$g3 = P_z$ $g6$ = c

$g7 = n$ (order of the ellipsoid, default=2)

IGTYPE = 5: $g1$ = Radius of torus

$g2$ = r

IGTYPE = 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)

IGTYPE = 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.

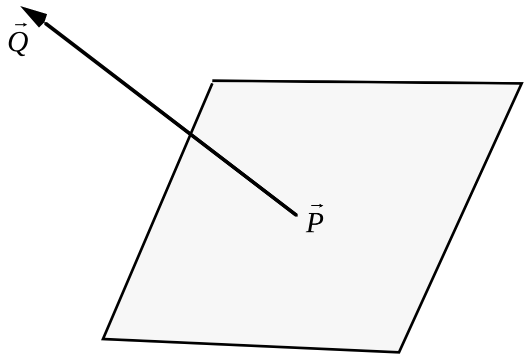
$g2$ = Scale factor for true thickness (optional)

$g3$ = Load curve ID defining thickness versus time. (optional)

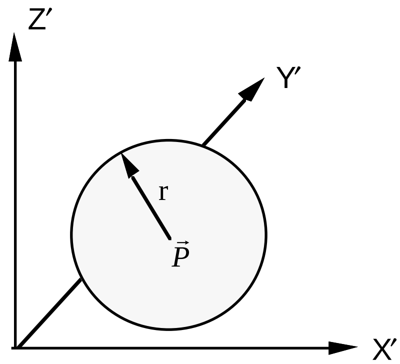
IGTYPE =10: $g1$ = Length of edge along X′ axis

$g2$ = Length of edge along Y′ axis

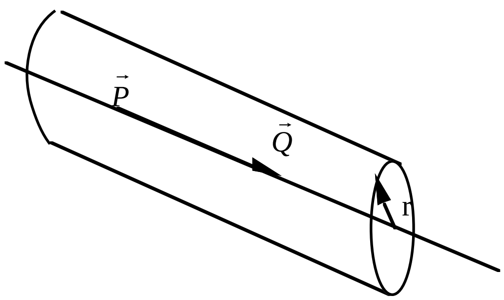
IGTYPE =11: $g1$ =Load curve ID defining axisymmetric surface profile about Z′-axis



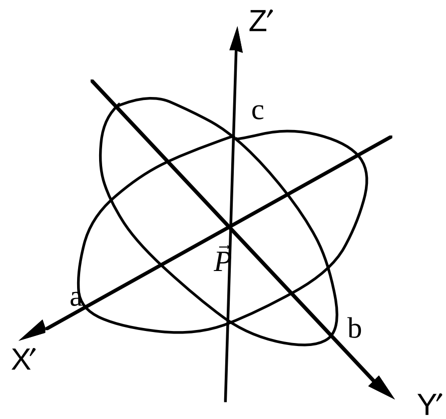
IGTYPE= 1: Infinite Plane



IGTYPE= 2: Sphere



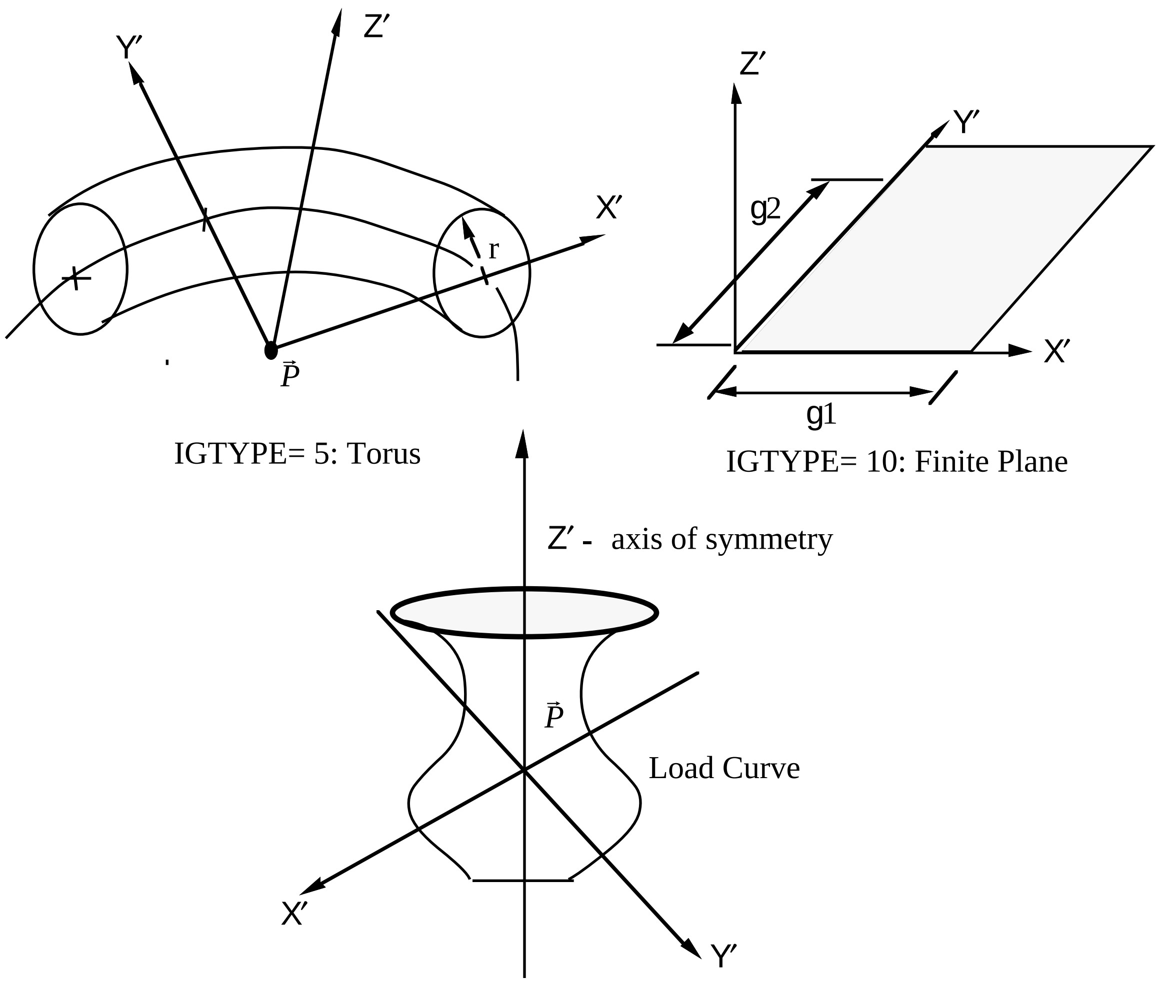
IGTYPE= 3: Infinite Cylinder



$$\left(\frac{X'}{a}\right)^n + \left(\frac{Y'}{b}\right)^n + \left(\frac{Z'}{c}\right)^n = 1$$

IGTYPE= 4: Hyperellipsoid

Figure 59.1a. Contact Entities.



IGTYPE= 5: Torus

IGTYPE= 10: Finite Plane

IGTYPE= 11:Load Curve

Figure 59.1b. Contact Entities.

Card Set 2
(16I5), or (8110) for LARGE option

Define NGENT card sets for each contact entity. These sets follow the NGENT sets defined above.

Columns		Quantity	Format	
1-5	(1-8)	Slave node n_1 or material m_1	I5	(I8)
6-10	(9-16)	Slave node n_2 or material m_2	I5	(I8)
.	.	.	.	
.	.	.	.	
.	.	.	.	

60. Rigid Body Stoppers

Rigid body stoppers provide a convenient way of controlling the motion of rigid tooling in metal forming 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.

Define NRBSTP control card sets here, see Columns 36-40 on Control Card 5. Optional slaved rigid bodies are defined after the control cards.

Control Card 1 (7I5,3E10.0) or (I10,6I5,3E10.0) for MLARG

Columns		Quantity	Format
1-5	(1-10)	Part ID of master rigid body.	I5 (I10)
6-10	(11-15)	LCMAX, load curve ID defining the maximum coordinate or displacement as a function of time. 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	I5 (I5)
11-15	(16-20)	LCMIN, load curve ID defining the minimum coordinate or displacement as a function of time. 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	I5 (I5)
16-20	(21-25)	NMXRB, number of rigid bodies that are slaved in the maximum coordinate direction to the master rigid body. This option requires additional input below.	I5 (I5)
21-25	(26-30)	NMNRB, number of rigid bodies that are slaved in the minimum coordinate direction to the master rigid body. This option requires additional input below.	I5 (I5)
26-30	(31-35)	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	I5 (I5)

Control Card 1 continued

Columns		Quantity	Format
31-35	(36-40)	Direction stopper acts in. EQ.1: x-translation EQ.2: y-translation EQ.3: z-translation EQ.4: arbitrary, defined by vector components rx, ry, rz EQ.5: x-axis rotation EQ.6: y-axis rotation EQ.7: z-axis rotation EQ.8: arbitrary, defined by vector components rx, ry, rz	I5 (I5)
36-45	(41-50)	rx-vector component	E10.0 (E10.0)
46-55	(51-60)	ry-vector component	E10.0 (E10.0)
56-65	(61-70)	rz-vector component	E10.0 (E10.0)

Control Card 2 (2E10.0)

Columns	Quantity	Format
1-10	Birth time	E10.0
11-20	Death time (default=10^{28})	E10.0

For each control card define NMXRB slave rigid bodies to the maximum coordinate followed by NMNRB slave rigid bodies for the minimum coordinate. Skip the input if no rigid bodies are slaves. See Figure 60.1.

Optional cards required if NMXRB > 0
Define NMXRB Cards with format (I10, E10.0)

Columns	Quantity	Format
1-10	Slave rigid body part ID, m_1	I10
11-20	Closure distance which activates 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. EQ.0.0: slaved rigid body stops when master stops	E10.0

Optional cards required if NMNRB > 0
Define NMNRB Cards with format (I10, E10.0)

Columns	Quantity	Format
1-10	Slave rigid body part ID, m_1	I10
11-20	Closure distance which activates 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. EQ.0.0: slaved rigid body stops when master stops	E10.0

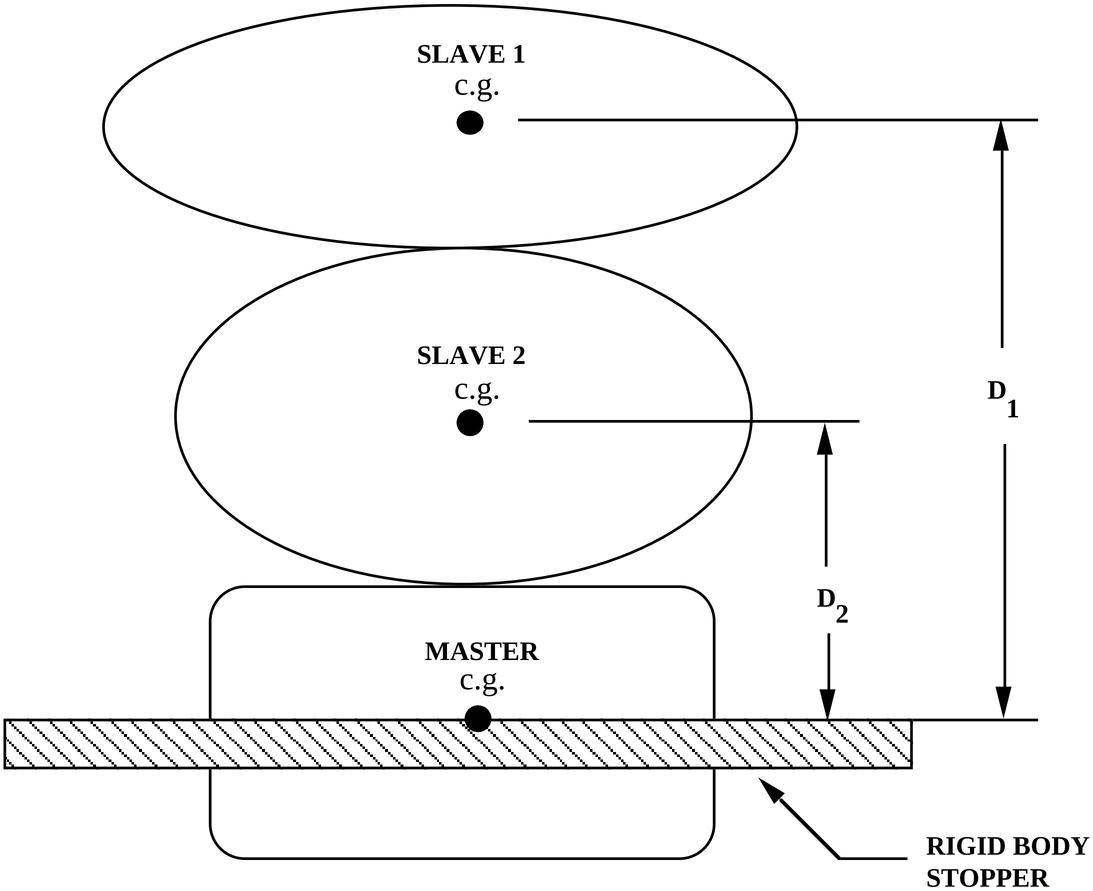


Figure 60.1. When the master rigid body reaches the rigid body stopper, the velocity component into the stopper is set to zero. Slave rigid bodies 1 and 2 also stop if the distance between their mass centers and the master rigid body is less than or equal to the input values D_1 and D_2, respectively.

61. AVS Database

Define the following information for the AVS Database if the output interval specified in columns 51-60 of the control card 23 is nonzero.

This is an ASCII database in a single file for displaying results with the LS-DYNA filter for the AVS software on STARDENT computers. It can be easily linked to any commercial post processor with little or no modification.

This 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.

Card 1 (I5)

Columns	Quantity	Format
1-5	Number of nodal vectors, NV	I5

Card 2,...NV+1 (2I5)

Columns	Quantity	Format
1-5	Nodal variable type, NT EQ.0: node EQ.1: brick EQ.2: beam EQ.3: shell	I5
6-10	Component number NT.EQ.0: Table 1 NT.EQ.1: Table 2 NT.EQ.2: not supported NT.EQ.3: Table 3	I5

Table 1
Nodal Quantities

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

Table 2
Brick Element Quantities Projected to Nodal Points

Component Number	Quantity
1	x-stress
2	y-stress
3	z-stress
4	xy-stress
5	yz-stress
6	zx-stress
7	effective plastic strain

Table 3

Shell Element Quantities Projected to Nodal Points

Component Number	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
21	outer surface effective plastic strain
22	bending moment-mxx
23	bending moment-myy
24	bending moment-mxy
25	shear resultant-qxx
26	shear resultant-qyy
27	normal resultant-nxx
28	normal resultant-nyy
29	normal resultant-nxy
30	thickness

Table 3. (Cont.)
Shell Element Quantities Projected to Nodal Points

Component Number	Quantity
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

62. MPGS Database

Define the following information for the MPGS Database if the output interval specified in columns 21-30 of control card 24 is nonzero. MPGS is a proprietary post-processor from Cray Research, Inc.

This database consists of a geometry file and multiple output files.

Card 1 (I5)

Columns	Quantity	Format
1-5	Number of nodal vectors, NV	I5

Card 2,...NV+1 (2I5)

Columns	Quantity	Format
1-5	Nodal variable type, NT EQ.0: node EQ.1: brick EQ.2: beam EQ.3: shell	I5
6-10	Component number NT.EQ.0: Table 1 NT.EQ.1: Table 2 NT.EQ.2: not supported NT.EQ.3: Table 3	I5

63. MOVIE Database

Define the following information for the MOVIE Database if the output interval specified in columns 31-40 of control card 24 is nonzero.

This database consists of a geometry file and multiple output files.

Card 1 (I5)

Columns	Quantity	Format
1-5	Number of nodal vectors, NV	I5

Card 2,...NV+1 (2I5)

Columns	Quantity	Format
1-5	Nodal variable type, NT EQ.0: node EQ.1: brick EQ.2: beam EQ.3: shell	I5
6-10	Component number NT.EQ.0: Table 1 NT.EQ.1: Table 2 NT.EQ.2: not supported NT.EQ.3: Table 3	I5

64. System Damping by Part ID

Skip this section if LCDAMP in columns 1-5 of Control Card 14 is greater than zero or equal to zero.

Card 1 (I5)

Columns	Quantity	Format
1-5	Number of parts for which system damping is defined, NMD	I5
6-10	IFLAG, flag to read directionality scale factors. EQ.0: skip EQ.1: read an additional card with the scale factors.	I5

Read 1 or 2 cards per part ID
Card 1 (2I5,E10.0) or (2I10,E10.0) for MLARG

Columns		Quantity	Format	
1-5	(1-10)	Part ID, N	I5	(I10)
6-10	(11-20)	Load curve number which specifies system damping for part N1	I5	(I10)
21-20	(21-30)	Scale factor for load curve (Default=1.0)	E10.0	(E10.0)

Optional Card 2 (6E10.0)
(Read if IFLAG=1)

Columns	Quantity	Format
1-10	Scale factor on global x translational damping forces.	E10.0
11-20	Scale factor on global y translational damping forces.	E10.0
21-30	Scale factor on global z translational damping forces.	E10.0
31-40	Scale factor on global x rotational damping moments.	E10.0

Columns	Quantity	Format
41-50	Scale factor on global y rotational damping moments.	E10.0
51-60	Scale factor on global z rotational damping moments.	E10.0

With 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:

$$F^n_{damp} = D_s mv$$

As seen from Figure 64.1 and the best damping constant for the system is usually based on the critical damping factor for the frequency or 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 (minimum) frequency of the structure. System damping is applied to both translational and rotational degrees of freedom.

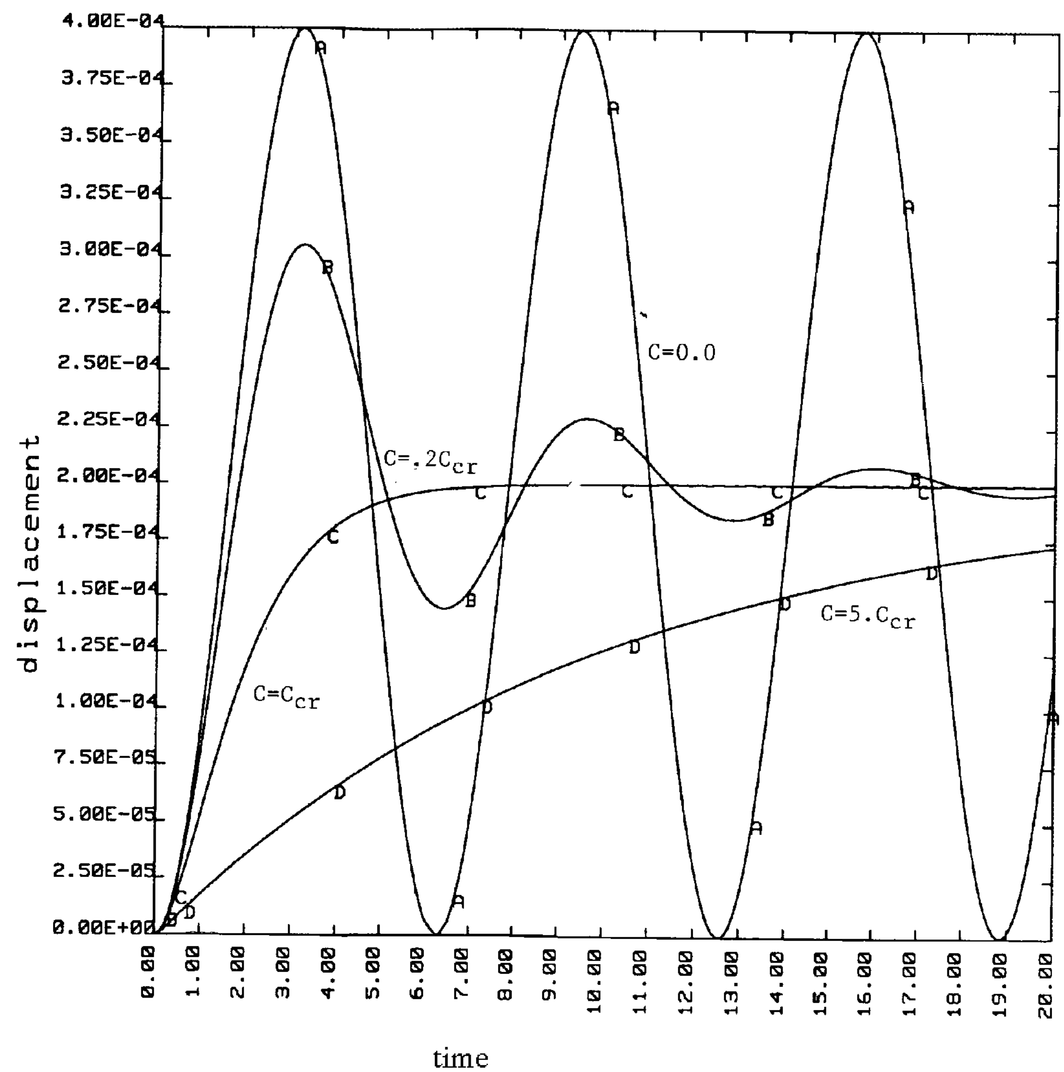


Figure 64.1 Displacement versus time curves with a variety of damping coefficients applied to a one degree-of-freedom oscillator.

65. Input Data For User Interface Control Subroutine

Define the input for this section if the number in column 46-50 of Control Card 15 is non-zero.

Card 1 (I5)

Columns	Quantity	Format
1-5	NUMINF, number of interfaces for which input data is provided.	I5

For each of the NUMINF interfaces define the following card sets:

Control Card (2I5)

Columns	Quantity	Format
1-5	Interface number to which this input data applies.	I5
6-10	Number of input constants, NUMCNS.	I5

Define (NUMCNS-1)/8+1 cards here:

Parameter Lists (8E10.0)

Columns	Quantity	Format
1-10	First input parameter	E10.0
11-20	Second input parameter	E10.0
21-30	Third input parameter	E10.0
31-40	Fourth input parameter	E10.0
.	.	.
.	.	.
.	.	.
71-80	Eighth input parameter	E10.0

66. Input Data For User Interface Friction Subroutine

Define the input for this section if the number in column 51-55 of Control Card 15 is nonzero.

Card 1 (I5)

Columns	Quantity	Format
1-5	NUMINF, number of interfaces for which input data is provided.	I5

For each of the NUMINF interfaces define the following card sets:

Control Card (2I5)

Columns	Quantity	Format
1-5	Interface number to which this input data applies	I5
6-10	Number of input constants, NUMCNS	I5

Define (NUMCNS-1)/8+1 cards here:

Parameter Lists (8E10.0)

Columns	Quantity	Format
1-10	First input parameter	E10.0
11-20	Second input parameter	E10.0
21-30	Third input parameter	E10.0
31-40	Fourth input parameter	E10.0
.	.	.
.	.	.
.	.	.
71-80	Eighth input parameter	E10.0

67. Linear Constraint Equations

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:

$$\underset{\sim}{\ddot{u}}_c = \left[\underset{\sim}{L}^t \underset{\sim}{M} \underset{\sim}{L} \right]^{-1} \underset{\sim}{L}^t \underset{\sim}{F}$$

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.

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.

Linear Constraint Equations

Define one input set for each constraint equation in this section for a total of NOCEQS (see Control Card 4, columns 36-40) sets.

Card 1 (I5)

Columns	Quantity	Format
1-5	Number of constrained degrees-of-freedom, *n*	I5

Cards 2,3,...,n+1
(I10,4x,6I1,E10.0)

Columns	Quantity	Format
1-10	Nodal point number, *k*	I10
11-14	Blank	4x

Define only one nonzero number in columns 15-20 below.

Columns	Quantity	Format
15	Insert 1 (0) for (no) translational constraint in global x-direction	I1
16	Insert 1 (0) for (no) translational constraint in global y-direction	I1
17	Insert 1 (0) for (no) translational constraint in global z-direction	I1
18	Insert 1 (0) for (no) rotational constraint about global x-axis	I1
19	Insert 1 (0) for (no) rotational constraint about global y-axis	I1
20	Insert 1 (0) for (no) rotational constraint about global z-axis	I1
21-30	Nonzero coefficient, C_k	E10.0

68. Cyclic Symmetry

These boundary conditons (Control Card 2, columns 36-40) can be used to model a segment of an object that has rotational symmetry such as an impeller. The segment boundaries, denoted as a side 1 and side 2, may be curved or planar. In this section a paired list of points are defined on the sides that are to be joined.

Card 1 (3E10.0)

Define a vector in the direction of the symmetry axis.

Columns	Quantity	Format
1-10	x-component	E10.0
11-20	y-component	E10.0
21-30	z-component	E10.0

Cards 2,3,4,...,NNCSYM+1
(3I5), or (3I8) for LARGE option

In the following input, the side 1 nodes lie on one side of the segment and the side 2 nodes lie on the opposite side. A one-to-one correspondence between the side 1 and side 2 nodes is necessary and assumed.

Columns		Quantity	Format
1-5	(1-8)	Interface nodal pair number EQ.0: increment nodal pair number by unity	I5 (I8)
6-10	(9-16)	Nodal point number of side 1 node	I5 (I8)
11-15	(17-24)	Nodal point number of side 2 node	I5 (I8)

Omitted data are automatically generated by incrementing the nodal point numbers by

$$\frac{\left(n_i - n_j\right)}{\left(sn_i - sn_j\right)}$$

where sn_i, sn_j are the node numbers on two successive cards and n_i and n_j are their corresponding nodal pair numbers. Side 1 and side 2 nodes are similarly generated.

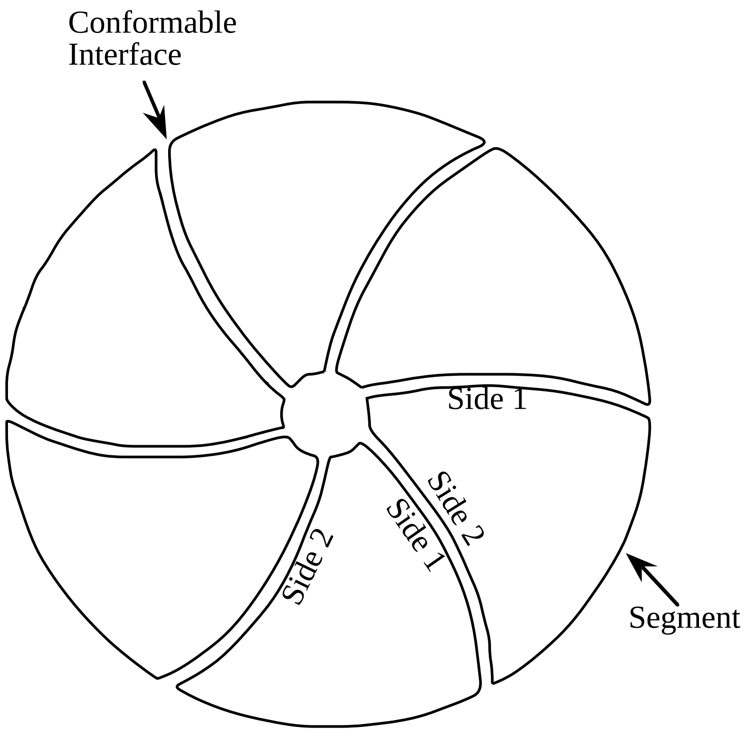


Figure 68.1. With cyclic symmetry only one segment is modeled.

69. Implicit Springback Solution

Card 1 (3I5)

Columns	Quantity	Format
1-5	Number of materials to be included in the springback calculation, NMSB	I5
6-10	Number of additional nodal constraints to be added, NANC	I5
11-15	Number of trim curves, NTC.	I5

Read the following card if and only if NMSB is nonzero.

Card 2,... (8I10) [Use as many cards as necessary]

Columns	Quantity	Format
1-10	First material to be included	I10
11-20	Second material	I10
21-30	Third material	I10
31-40	Fourth material	I10
.		
.		
.		
71-80	Eighth material	I10

Card 2+NMSB,... (I10,2E10.0) [Define NANC cards]

Columns	Quantity	Format
1-10	Node number	I10
11-20	Displacement boundary condition code 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	E10.0
21-30	Rotational boundary condition code 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	E10.0

Trim Curve Data Cards
Define NTC cards, i.e., one for each curve (2I5,6E10.0,E5.0).

Columns	Quantity	Format
1-5	Load curve number giving trim curve (general x-y data type 2)	I5
6-10	Trim flag EQ.-1: remove elements outside curve EQ. 1: remove elements inside curve	I5
11-20	Tx, the X-coordinate of orientation vector tail	E10.0
21-30	Ty, the Y-coordinate of orientation vector tail	E10.0
31-40	Tz, the Z-coordinate of orientation vector tail	E10.0
41-50	Hx, the X-coordinate of orientation vector head	E10.0
51-60	Hy, the Y-coordinate of orientation vector head	E10.0
61-70	Hz, the Z-coordinate of orientation vector head	E10.0
71-75	TMTOL, the tolerance for small element creation $0.0 \leq \text{TMTOL} \leq 1.0$, default = 0.25	E5.0

Trim curves must be closed loops. If the first and last points do not coincide, LS-DYNA will generate one additional segment to close the curve. Load curves must be type # 2, the general x-y data type.

The trimming orientation vector is used to define the local coordinate system for the trim curve, and the direction in which the trim curve is projected onto the deformed mesh to create the trim line (see figure 69.1). The default orientation vector is the global Z-axis, with the trim curve defined in the global X-Y plane. This default is used if all tail and head coordinates are entered as 0.0.

The trimming tolerance TMTOL limits the size of the smallest element created during trimming (see figure 69.2). 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.

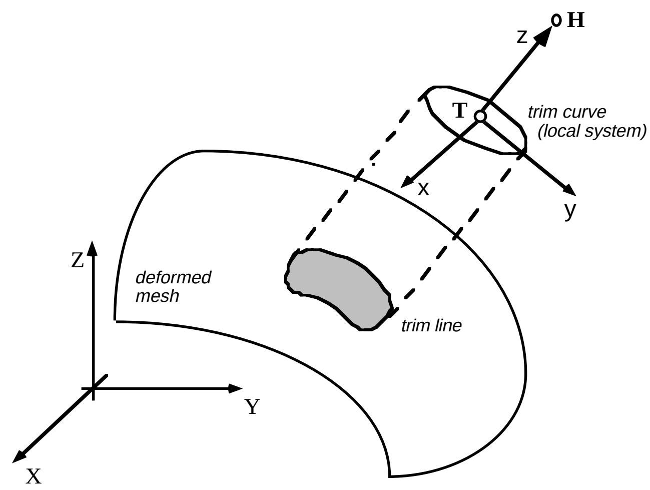


Figure 69.1 - Trimming Orientation Vector. The tail (**T**) and head (**H**) points define a local coordinate system (x,y,z). The local x-direction is constructed in the Xz-plane. Trim curve data is input in the x-y plane, and projected in the z-direction onto the deformed mesh to obtain the trim line.

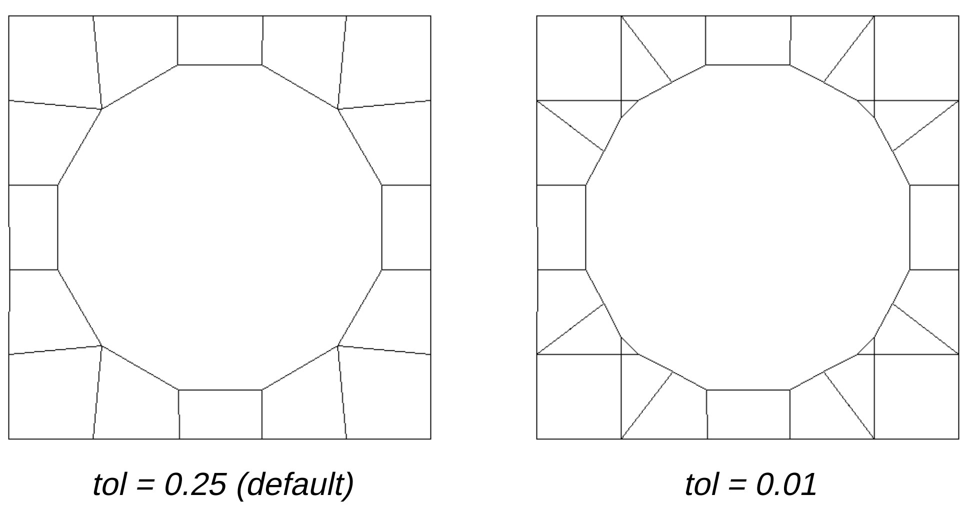


Figure 69.2 - Trimming Tolerance. The tolerance limits the size of the small elements generated during trimming. The default tolerance (left) produces large elements. Using a tolerance of 0.01 (right) allows smaller elements, and more detail in the trim line.

70. Superplastic Forming Option

Card 1 (4I5,5E10.0,I5)

This option must be used with material model 64 for strainrate sensitive, powerlaw plasticity.

Columns	Quantity	Format
1-5	Load curve number for Phase I pressure loading	I5
6-10	Interface number to determine completion of Phase I	I5
11-15	Load curve number for Phase II pressure loading (reverse) See comments below.	I5
16-20	Interface number to determine completion of Phase II	I5
21-30	Desired strain rate	E10.0
31-40	Percent of nodes in contact to terminate Phase I	E10.0
41-50	Percent of nodes in contact to terminate Phase II (calculation)	E10.0
51-60	Minimum allowable value for load curve scale factor	E10.0
61-70	Maximum allowable value for load curve scale factor	E10.0
71-75	Number of cycles for monotonic pressure after reversal	I5

Card 2 (E10.0)

Columns	Quantity	Format
1-10	Output intervals for files “pressure,” “curve1,” and “curve2”	E10.0

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.

The constraint method contact, type 18-nodes to surface, is recommended for superplastic forming simulations. 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.

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.

71. Material Repositioning Section

This section 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 input is provided here.

The flag to read this input appears on control card 17 in columns 31-35. Card 1 below gives the number of materials to be repositioned to make the model consistent with CAL3D/MADYMO initial geometry (NMOVCG). NMOVCG input card follow.

Card 1 (I10)

Columns	Quantity	Format
1-10	NMOVCG, number of materials to be repositioned. Also, with this option a merged rigid body can be fixed in space while the nodes and elements of the generated CAL3D/MADYMO parts are repositioned.	I10

Card 2,3,..., NMOVCG+1 (3I5) or (3I10) if MLARG

Columns		Quantity	Format
1-5	(1-10)	LS-DYNA material ID, MID, to be moved: GT.0: MID is moved LT.0: |MID| is not moved where |MID| is a material that is properly positioned which is merged to a CAL3D/MADYMO system in the rigid body merge section, Section 35.	I5 (I10)
6-10	(11-20)	CAL3D segment number/MADYMO system number	I5 (I10)
11-15	(21-30)	MADYMO ellipse/plane number GT.0: ellipse number LT.0: absolute value is plane number	I5 (I10)

72. Termination Criterion

In this section displacement termination is first defined for nodal points where the number of cards specified on Control Card 8, col. 51-60 are provided. In the second part of this section displacement termination is defined by a rigid body displacement where the number of cards specified on Control Card 8, col. 61-70 are provided. This second part is useful for rigid bodies with no user defined nodal points.

Caution: The inputs are different for the nodal and rigid body stop conditions. For the nodal stop condition the global coordinates are input, and for the rigid body stop condition the relative global translations are input.

Cards 1,...,NUMSTOP (2I5,2E10.0), or (I8,I5,2E10.0) for LARGE option

Columns		Quantity	Format	
1-5	(1-8)	Node ID	I5	(I8)
5-10	(9-13)	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	I5	(I5)
11-20	(14-23)	Maximum (most positive) coordinate, options 1, 2 and 3 above only	E10.0	E10.0
21-30	(24-33)	Minimum (most negative) coordinate, options 1, 2 and 3 above only	E10.0	E10.0

The analysis terminates when the current position of the node specified reaches either the maximum or minimum value (types 1, 2 or 3), or picks up force from any contact surface (type 4). If more than one condition is input, the analysis stops when any of the conditions is satisfied.

Termination

Termination by rigid body displacement is defined below.

Cards 1,...,NRBEND (2I5,2E10.0), or (I8,I5,2E10.0) for LARGE option

Columns		Quantity	Format	
1-5	(1-8)	Rigid body ID	I5	(I8)
5-10	(9-13)	Stop criterion EQ.1: global x displacement EQ.2: global y displacement EQ.3: global z displacement EQ.4: displacement magnitude	I5	(I5)
11-20	(14-23)	Maximum (most positive) displacement, options 1, 2, 3, and 4 above	E10.0	E10.0
21-30	(24-33)	Minimum (most negative) displacement, options 1, 2, and 3 above.	E10.0	E10.0

The analysis terminates when the current displacement of the rigid body specified reaches either the maximum or minimum value. If more than one condition is input, the analysis stops when any of the conditions is satisfied.

Termination by contact is defined below.

Cards 1,...,NCNEND (I10,2E10.0)

Columns	Quantity	Format
1-10	Contact ID	I10
11-20	Activation time.	E10.0
21-30	Time duration of null resultant force prior to termination. This time is tracked only after the activation time is reached. EQ.0.0: Immediate termination after null force is detected.	E10.0

The analysis terminates when the magnitude of the contact interface resultant force is zero. If more than one condition is input, the analysis stops when any of the conditions is satisfied.

73. ALE Smoothing Constraints

ALE smoothing constraints force a node to remain at its initial parametric location along a line between two other nodes at each mesh smoothing operation. Input NALESC constraint cards.

(4I5,3E10.0) or (3I8,I5,3E10.0) for LARGE Option

Columns		Quantity	Format	
1-5	(1-8)	Slave node	I5	(I8)
6-10	(9-16)	First node on line	I5	(I8)
11-15	(17-24)	Last node on line	I5	(I8)
16-20	(25-29)	IPRE EQ.0: smoothing constraints are performed after mesh relaxation. EQ.1: smoothing constraints are performed before mesh relaxation.	I5	(I5)
21-30	(30-39)	x-coordinate of constraint vector.	E10.0	E10.0
31-40	(40-49)	y-coordinate of constraint vector.	E10.0	E10.0
41-50	(50-59)	z-coordinate of constraint vector.	E10.0	E10.0

74. Tracer Particles

Define the NTRACE (Control Card 1) cards in this section. 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 data.

(E10.0,I5,3E10.0)

Columns	Quantity	Format
1-10	Start time for tracer particle	E10.0
11-15	Tracking option EQ.0: particle follows material EQ.1: particle is fixed in space	I5
16-25	Initial x-coordinate	E10.0
26-35	Initial y-coordinate	E10.0
36-45	Initial z-coordinate	E10.0

75. Parts Tied in Solid Parts

Define the NALTIE (Control Card 1) input sets. This section couples a Lagrange mesh to a solid mesh, see Figure 75.1. This capability allows Lagrangian shells or solids to be coupled to an Eulerian flow. This option may also be used to model rebar in concrete or tire cords in rubber.

The slave list must be a list of Lagrangian part ID's or a list of Lagrangian segments and the master list consists of solid element part ID's.

Define NALTIE Control Cards, one for each input set.

Control Card (4I5,I10)

Columns	Quantity	Format
1-5	Number of slave part ID's (Lagrangian) If the Lagrangian elements are to be defined by segments then input zero and define the number of slave segments in colums 21-30. Slaves cannot be defined by both parts and segments in the same set.	I5
6-10	Number of master part ID's	I5
11-15	Number of quadrature points	I5
16-20	Coupling type EQ.1: acceleration EQ.2: velocity (default) EQ.3: penalty	I5
21-30	Number of Lagrange slave segments. Must be zero if the number of slave parts is non zero.	I10

Parts Tied in Solid Parts

For each tied set define the slave and master sides of the set. The order is

slave side for set 1
master side for set 1
slave side for set 2
master side for set 2
. . . .
. . . .

Define the Lagrangian slaves by one of the following two methods.

By Segments

Slave Segments (5I8)

Columns	Quantity	Format
1-8	Segment number	I8
9-16	First node on segment, N1	I8
17-24	Second node on segment, N2	I8
25-32	Third node on segment, N3	I8
33-40	Fourth node on segment, N4	I8

By Parts

Slave ID's (16I5) or (8I10) if MLARG

Columns		Quantity	Format
1-5	(1-10)	First slave part ID	I5 (I10)
6-10	(11-20)	Second slave part ID	I5 (I10)
.			
.			
.			

Master ID'S (16I5) or (8I10) if MLARG. Insert as many cards as necessary.

Columns		Quantity	Format
1-5	(1-10)	First master part ID (solids)	I5 (I10)
6-10	(11-20)	Second master part ID	I5 (I10)
.			

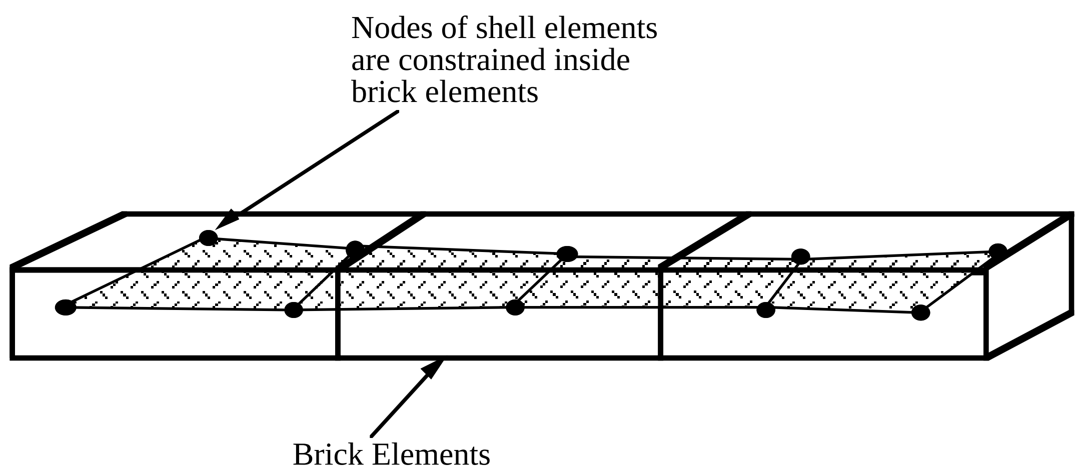


Figure 75.1. Nodes of Lagrange elements are constrained to move with the material points of the solid elements. The slave elements may be either Lagrangian shells, beams or bricks.

76. Multi-Material Euler

If NVOIDM (Control Card 26, Col 31-35) is nonzero define the input in this section. The simplest input allows the modelling of a void within a solid element type 12. In this latter case, the input which immediately follows is required. This capabliity allows the definition of void materials as fluids for convecting full Eulerian materials through free surfaces. Void materials are listed below.

For full mixed material Eulerian elements, type 11, NVOIDM is set to -1 and multi-material groups and mixture groups can then be defined.

The void option and multiple materials per element are incompatible and cannot be used together in the same run.

Define the following input if and only if NVOIDM>0.

If NVOIDM is greater than zero, define NVOIDM material ID's. Use as many cards as neccessary.

(10I8)

Columns	Quantity	Format
1-8	First void material	I8
9-16	Second void material	I8
17-24	Third void material	I8
.	.	.
.	.	.
.	.	.

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.

Define the following input if and only if NVOIDM<0.

Define the number of card sets, NALEGP=|NVOIDM|, specified on Control Card 26 in Columns 31-35. The following cards define the number of materials and the material ID's of each 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 follow cards. Define a total of NALEPG groups. NOTE THAT NALEPG MUST BE LESS THAN OR EQUAL TO THREE.

Card 1 (I8)

Columns	Quantity	Format
1-8	Number of materials in group	I8

Cards 2, 3, ... (10I8)

Columns	Quantity	Format
1-8	First material ID	I8
9-16	Second material ID	I8
17-24	Third material ID	I8
.	.	.
.	.	.
.	.	.

Example

OIL	WATER	AIR
GROUP 1	GROUP 2	GROUP 3
MAT ID 1, AND 2	MAT ID 3	MATI ID 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 materials, mat ID's 1 and 2.

The second group contains one material, mat ID 3.

The third group contains three materials, mat ID's 5, 6 and 7.

77. Pressure Outflow Boundary Conditions

Define the number of card sets, NOFLOW, specified on Control Card 3, columns 41-45.

Card (5I5) or (5I8) for LARGE option

Columns		Quantity	Format
1-5	(1-8)	Pressure card number	I5 (omit)
6-10	(9-16)	Nodal point n_1	I5 (I8)
11-15	(17-24)	Nodal point n_2	I5 (I8)
16-20	(25-32)	Nodal point n_3	I5 (I8)
21-25	(33-40)	Nodal point n_4	I5 (I8)

78. Reference Geometry for Airbag

(2I5,3'//NIF//',2I5), or (I8,I5,3'//NIF//',I5) for LARGE option
(NIF is defined on Control Card 11 Cols. 16-20)

Define the input in this section if and only if the parameter on Control Card 12, Columns 61-65 is nonzero.

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.

Card 1, (I10)

Columns	Quantity	Format
1-10	Number of nodal coordinates to be reset below, NUMPTS	I10

Cards 2,NUMPTS+1
(I5,5X,3'//NIF//',2I5), or (I8,5X,3'//NIF//',I5) for LARGE option
(NIF is defined on Control Card 11 Cols. 16-20)

Columns		Quantity	Format
1-5	(1-8)	Node number	I5 (I8)

Reference Geometry for Airbag

Columns		Quantity	Format	
6-10	(9-13)	Blank	5X	(5X)

If NIF = ‘E20.0’, otherwise subtract 10 spaces from the next 3 fields.

11-30	(14-33)	x-coordinate	E20.0	(E20.0)
31-50	(34-53)	y-coordinate	E20.0	(E20.0)
51-70	(54-73)	z-coordinate	E20.0	(E20.0)

79. USA Surface Boundary Condition Cards
(6I5), or (I5,4I8) for LARGE option

This option is for coupling with the Underwater Shock Analysis code USA which can be used for determining the transient response of totally or partially submerged structures to acoustic shock waves [DeRuntz, 1993]. USA is based on a boundary element formulation.

Card 1 (2I5)

Columns	Quantity	Format
1-5	Number of surface segments in the USA interface, NUSA	I5
6-10	Number of (wet) beam nodes in the USA interface, NUSAB	I5

Define the NUSA surface cards.

Cards 2,...NUSA+1 (6I5), or (I5,4I8) for LARGE option

Columns		Quantity	Format
1-5	(-)	Surface card number (Skip for Large option)	I5 (omit)
6-10	(1-5)	Wet or Dry Flag EQ.0: dry EQ.1: wet	I5 (I5)
11-15	(6-13)	Nodal point n_1	I5 (I8)
16-20	(14-21)	Nodal point n_2	I5 (I8)
21-25	(22-29)	Nodal point n_3	I5 (I8)
26-30	(30-37)	Nodal point n_4	I5 (I8)

The 4 node surface segment normals must point into the fluid.

When running a coupled problem with USA the procedure involves several steps. First, LS-DYNA is executed to create a linking file "dyna.pre" used by USA and a dump file "d3dump". The execution lines are:

LS-DYNA > outputfilename0 <cr>
i=inputfilename <cr>

Where we note that no prompt is provided for the second line of the input and that <cr> means that the **c**arriage **r**eturn key should be pressed. Then, it is necessary to create the fluid mass matrix by running the code FLUMAS:

FLUMAS < flumasinputfilename > flumasoutputfilename

The ouput file from the LS-DYNA run, dyna.pre, is referenced in the input file to FLUMAS. Next, the code AUGMAT which initializes constants and arrays for the staggered solution procedure for the transient analysis is executed:

AUGMAT < augmatinputfilename > augmatoutputfilename

Finally, the coupled solution can begin by again executing LS-DYNA:

LS-DYNA > outputfilename <cr>
r=d3dump <cr>
*add usainputfilename <cr>

We note that no prompts are provide for the second and third lines of input. 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.

80. MCOL Input Cards

(I10,2E10.0)

This section is input if NMCOL is nonzero on control card 17, columns 66-70.

Columns	Quantity	Format
1-10	Maximum number of time steps in MCOL calculation, MXSTEP. If the number of MCOL time steps exceeds MXSTEP, then LS-DYNA will terminate.	I10
11-20	Time interval for MCOl subcycling EQ.0.0: no subcycling	E10.0
21-30	Time interval for output of MCOL rigid body data	E10.0

Card 2...Card 1+NMCOL

Input NMCOL Ship Definition (I5,A60) or (I10,A60) if MLARG

Columns		Quantity	Format
1-5	(1-10)	LS-DYNA rigid body material assignment for ship	I5 (I10)
6-65	(11-70)	File name containing MCOL input parameters for this ship.	A60 (A60)

The MCOL output is sent to the files, MCOLOUT (ship positions) and MCOLENERGY (energy breakdown). In TAURUS, MCOLOUT can be plotted through the rigid body time history option and MCOLENERGY must be plotted with the MADYMO option.

81. Temperature Initial Condition Cards
(I8,E10.0)

Define the number of nodes with temperature initial conditions as specified on control card 27.

Columns	Quantity	Format
1-8	Node number	I8
9-18	Temperature initial condition	E10.0

82. Element Heat Generation Cards
(I8,I5,E10.0)

Define the number of brick and shell elements with heat generation as specified on control card 27.

Columns	Quantity	Format
1-8	Define the number of brick elements with thermal generation	I8
9-16	Define the number of shell elements with thermal generation	I5

Define the number of brick elements with thermal generation. Include nbrick cards.

Columns	Quantity	Format
1-8	Brick Element number	I8
9-13	Load curve ID for volumetric heat generation rate, $\dot{q}'''$ GT.0: function versus time EQ.0: use multiplier value only LT.0: function versus temperature	I5
14-23	Curve multiplier for $\dot{q}'''$	E10.0

Define the number of shell elements with thermal generation, Include nshell cards.

Columns	Quantity	Format
1-8	Shell Element number	I8
9-13	Load curve ID for volumetric heat generation rate, $\dot{q}'''$ GT.0: function versus time EQ.0: use multiplier value only LT.0: function versus temperature	I5
14-23	Curve multiplier for $\dot{q}'''$	E10.0

83. Temperature Boundary Condition Cards (I8,I5,E10.0)

Define the number of nodes with temperature boundary conditions as specified on control card 27.

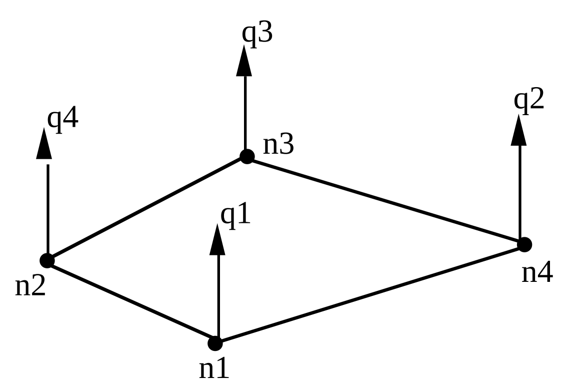
Columns	Quantity	Format
1-8	Node number	I8
9-13	Load curve ID for temperature versus time EQ.0: use multiplier value only	I5
14-23	Curve multiplier for temperature	E10.0

84. Flux Boundary Condition Cards (4I8,I5,4E10.0)

Define the number of flux boundary condition surfaces as specified on control card 27.

Columns	Quantity	Format
1-32	Node numbers N_1 - N_4 defining surface	4I8
33-37	Load curve ID for heat flux GT.0: function versus time EQ.0: use multiplier values only LT.0: function versus temperature	I5
38-77	Curve multipliers for heat flux at nodes N_1 - N_4	4E10.0

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 below.



85. Convection Boundary Condition Cards (4I8,I5,E10.0,I5,2E10.0)

Define the number of convection boundary condition surfaces as specified on control card 27.

Columns	Quantity	Format
1-32	Node numbers N_1 - N_4 defining surface	4I8
33-37	Load curve ID for heat transfer coefficient, h GT.0: function versus time EQ.0: use multiplier value only LT.0: function versus temperature	I5
38-47	Curve multiplier for h	E10.0
48-52	Load curve ID for T_∞ versus time EQ.0: use multiplier value only	I5
53-62	Curve multiplier for T_∞	E10.0

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

h	heat transfer coefficient;
$(T - T_\infty)$	temperature potential.

86. Radiation Boundary Condition Cards

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

1. The first type, specified by the "number of radiation boundary condition surfaces" on control card 27, 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, specified by the "number of enclosure radiation surfaces" on control card 27, 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**.

1. Define the number of radiation boundary condition surfaces

Columns	Quantity	Format
1-32	Node numbers N_1 - N_4 defining surface note: use only N_1 - N_2 to define a 2D surface	4I8
33-37	Load curve ID for radiation factor, *f* GT.0: function versus time EQ.0: use multiplier value only LT.0: function versus temperature	I5
38-47	Curve multiplier for *f*	E10.0
48-52	Load curve ID for T_∞ versus time EQ.0: use multiplier value only	I5
53-62	Curve multiplier for T_∞	E10.0

The radiation factor is defined as $f=\sigma\varepsilon F$, where σ is the Stefan Boltzmann constant, ε is the surface emissivity, and F is the view factor between the surface and the environment (usually F=1).

2. Define the number of enclosure radiation surfaces

Card 1

Columns	Quantity	Format
1-10	Stefan Boltzmann constant	I10

Card 2,...., number of surfaces

Columns	Quantity	Format
1-32	Node numbers N_1 - N_4 defining surface note: use only N_1 - N_2 to define a 2D surface	4I8
33-37	Load curve ID for surface emissivity GT.0: function versus time EQ.0: use multiplier value only LT.0: function versus temperature	I5
38-47	Curve multiplier for emissivity	E10.0

Surface-to-surface (area*view factor) file - viewfl

A file, with the name **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}

87. Boundary Element Method for Fluid Dynamics

Define the input for this section if ICFD in Columns 76-80 of Control Card 17 equals 2. Otherwise skip this section.

Boundary Element Method Control
Card 1 (3I8, E10.0, I8, E10.0)

Columns	Quantity	Format
1-8	NBEMSG, number of boundary element method segments.	I8
9-16	NWAKES, number of boundary element method wakes.	I8
17-24	LWAKES, number of elements in each wake (length of the wakes).	I8
25-34	DTBEM, time increment between calls to the boundary element method routines. Fluid dynamic pressures on the structure are held constant until they are updated by the BEM routines.	E10.0
35-42	IUPBEM, number of times the fluid dynamic pressures are computed before the boundary element method matrix of influence coefficients is recomputed and factored.	I8
43-52	FARBEM, Nondimensional distance for which a point in the fluid flow is considered to be in the far-field of a boundary element segment.	E10.0

Specification of Onset Flow
Card 2 (6E10.0)

Columns	Quantity	Format
1-10	VINF(1), x component of fluid velocity vector.	E10.0
11-20	VINF(2), y component of fluid velocity vector.	E10.0
21-30	VINF(3), z component of fluid velocity vector.	E10.0
31-40	RHOBEM, fluid density.	E10.0
41-50	PRESBEM, fluid static pressure.	E10.0
51-60	AIRMACH, Mach number for Prandtl-Glauert compressibility correction. Cannot be used if shocks are present in flow.	E10.0

Body Surface Definition
Cards 3,...2+NBEMSG (5I8)

Body surface definition cards. NBEMSG cards (one for each segment). These cards specifiy the nodes used to define each boundary element segment. For triangular segments the 4th node number should be the same as the 3rd node number. It is recommended that the boundary element segments use the same nodes and be coincident with the structural shell segments (or the outer face of brick 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. The boundary element segements can be easily made coincident with structural thin shell segments by using a negative number for the segment identification number. If the segment identification number is less than zero then the structural thin shell element whose identification number is its negative will be used to define the boundary element segment nodes.

Columns	Quantity	Format
1-8	The identification number of a boundary element segment.	I8
	If less than zero the structural thin shell element with the negative	

identification number will be used to define the boundary element segment nodes.

9-16	The node number of the 1st corner node of the segment.	I8
17-24	The node number of the 2nd corner node of the segment.	I8
25-32	The node number of the 3rd corner node of the segment.	I8
33-40	The node number of the 4th corner node of the segment.	I8

Cards 3+NBEMSG,...2+2*NBEMSG (5I8)

Segment neighbor cards. NBEMSG cards (one for each segment). Neighbors are used for finite-difference computations of the gradient of the boundary element singularity strengths. The surface pressures are computed from these gradients. The neighbors must be specified with care to obtain accurate computations of fluid pressures, and special rules must be followed at junctions between BEM wakes and the body, and near body surface slope discontinuities. Please see the detailed discussion in the Theory Manual for specific guidance on the definition of neighbors.

Columns	Quantity	Format
1-8	The identification number of a boundary element segment.	I8
9-16	The number of the boundary element segment which is the neighbor on the 1st side.	I8
17-24	The number of the boundary element segment which is the neighbor on the 2nd side.	I8
25-32	The number of the boundary element segment which is the neighbor on the 3rd side.	I8
33-40	The number of the boundary element segment which is the neighbor on the 4th side.	I8

Cards 3+2*NBEMSG,...2+2*NBEMSG+NWAKES (4I8)

Wake cards. NWAKES cards (one for each wake).

Columns	Quantity	Format
1-8	The wake number (for file readability: not used by LS-DYNA).	I8
9-16	The “upper” segment to which the wake is attached. The “upper” direction is arbitrary.	I8
17-24	The edge of the upper segment to which the wake is attached.	I8
25-32	The “lower” segment to which the wake is attached. The “lower” direction is arbitrary; this segment is the boundary element segment which shares the side with the “upper” segment to which the wake is attached.	I8

88. User Defined Loading and Sub-Sea Structural Loading

User Defined Loading
(8E10.0)

Define the number of input parameters specified on Control Card 10, columns 31-35. Use as many cards as necessary. Skip this section if no parameters are to be input, i.e., the flag for the user defined loading subroutine is greater than or equal to zero.

Columns	Quantity	Format
1-10	First parameter	E10.0
11-20	Second parameter	E10.0
21-30	Third parameter	E10.0
31-40	Fourth parameter	E10.0
.		
.		
.		
71-80	Eighth parameter	E10.0

Sub-Sea Structural Loading

Define the input in this section if SSA on Control Card 10, columns 46-50, is defined as a nonzero number; otherwise, skip this section. This model allows a simple way of loading the structure to account for the effects of the primary explosion and the subsequent bubble oscillations. The unit conversion factors, defined on Card 17 for the MADYMO3D/GM-CAL3D length conversion, should be specified to convert from the sub-sea structural loading input units to a kilogram-meter-second set of units.

Control Card (I10,5E10.0,I10)

Columns	Quantity	Format
1-10	NS, number of charges	I10
11-20	NE, number of explosive materials	E10.0
21-30	VS, sound speed in fluid	E10.0
31-40	DS, density of fluid	E10.0
41-50	IREF, consider reflections from sea floor. EQ.0: off EQ.1: on	E10.0
51-60	ZB, z coordinate of sea floor if IREF=1, otherwise, not used.	E10.0
61-70	ZSURF, z coordinate of sea surface	E10.0
71-80	NPIDS, number of parts defining wet surface EQ.0: all parts are included. GT.0: define NPIDS part ID's below.	I10

Control Card (I10)

Columns	Quantity	Format
1-10	NFLD, number of flooding control cards	I10

Define Flooding Controls if NFLD>0 (2I10,E10.0)

Columns	Quantity	Format
1-10	Material or part ID subject to flood control	I10
11-20	Flooding status EQ.1: Fluid on both sides. EQ.2: Fluid outside, air inside. EQ.3: Air outside, fluid inside. EQ.4: Material or part is ignored.	I10
21-30	Tubular outer diameter of beam elements. For shell elements this input must be greater than zero for loading.	E10.0

Define Part ID's if NPIDS>0 (8I10)

Skip this section if NPIDS equals 0. Use as many cards as necessary.

Columns	Quantity	Format
1-10	First material or part ID to be included	I10
11-20	Second material or part ID	I10
21-30	Third material or part ID	I10
31-40	Fourth material or part ID	I10

Columns	Quantity	Format
.		
.		
.		
71-80	Eighth material or part ID	I10

Define NE explosive property sets. (8E10.0)

Columns	Quantity	Format
11-20	NE, explosive material ID number	E10.0
21-30	A, shock pressure parameter	E10.0
31-40	α, shock pressure parameter	E10.0
41-50	γ, time constant parameter	E10.0
51-60	K_θ, time constant parameter	E10.0
61-70	κ, ratio of specific heat capacities	E10.0

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.

Define NS explosive charges. (6E10.0, I10, E10.0)

Columns	Quantity	Format
1-10	XS, X coordinate of charge	E10.0
11-20	YS, Y coordinate of charge	E10.0
21-30	ZS, Z coordinate of charge	E10.0
31-40	W, weight of charge	E10.0
41-50	TDELY, time delay before charge detonates	E10.0
51-60	RAD, charge radius	E10.0
61-70	MEID, explosive material ID	I10
71-80	CZ, water depth	E10.0

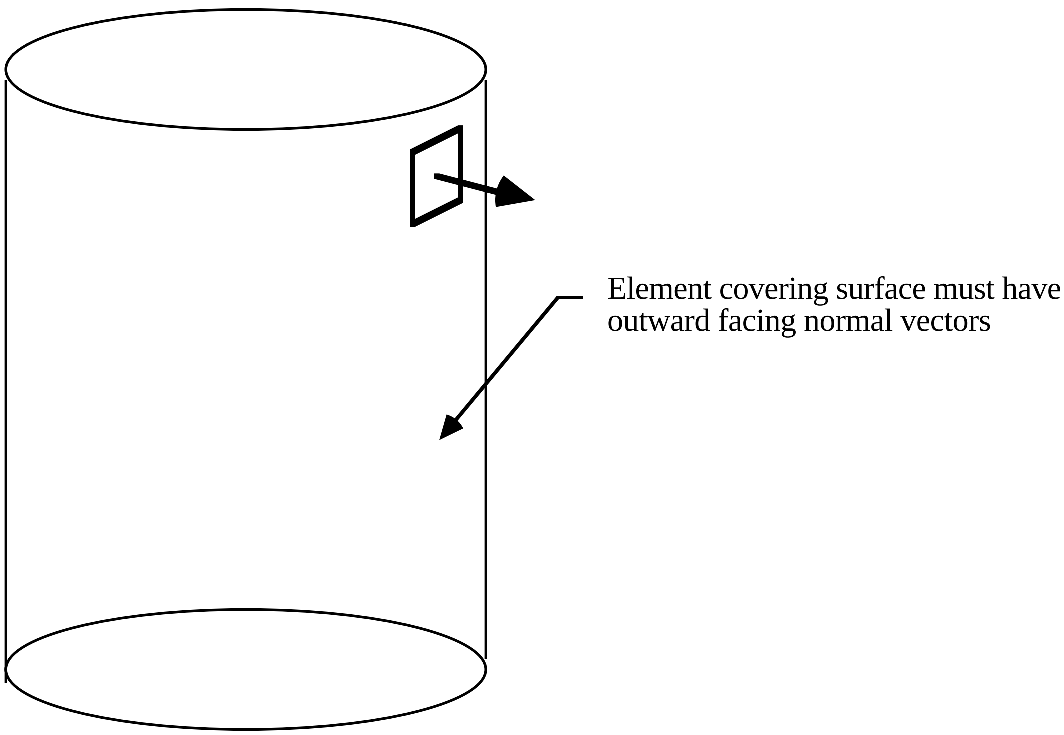


Figure 87.1. The shell elements interacting with the fluid must be numbered such that their outward normal vector points into the fluid media.

89. Subsystem Definitions for the SSSTAT File
(8I10)

Define NSS (Control Card 20, Columns 76-80) subsystems sets for output of the subsystem energy breakdown in the SSSTAT file. The SSSTAT is output frequency is identical to that of the GLSTAT file.

For each substem defined the follow 2 or more cards.

Columns	Quantity	Format
1-10	NUM, number of part IDs in the subsystem.	I10

Define NUM part ID's. Use as many cards as necessary.

Columns	Quantity	Format
1-10	First part ID to be included	I10
11-20	Second part ID	I10
21-30	Third part ID	I10
31-40	Fourth part ID	I10
.		
.		
.		
71-80	Eighth ID	I10

90. Feedback Control for Load Curve Adjustment
(8I10)

Define NFDBCK (Control Card 3, Columns 46-50) control sets. Each set provides information that is used as the solution evolves to scale the ordinate values of the specified load curve ID.

Define NFDBCK load sets. Each set contains 3 cards.

Card 1 (3I10)

Columns	Quantity	Format
1-10	Load curve ID to control.	I10
11-20	Active part ID for load curve control.	I10
21-30	Load curve ID defining flow limit diagram. If the ordinate value, of the maximum principal strain is exceeded the scale factor for flow, SF, is active. See Figure 3.15 for an explanation of the flow limit diagram.	I10

Card 2 (3E10.0)

Columns	Quantity	Format
1-10	If the strain ratio, $\varepsilon_{major_{fld}} / \varepsilon_{major_{workpiece}}$, is exceeded the scale factor for flow, SF, is active.	E10.0
11-20	If the thickness strain limit is exceeded the scale factor for thickening, ST, is active.	E10.0
21-30	Scale factor for flow, SF (Default=1.0)	E10.0
31-40	Scale factor for thickening, ST (Default=1.0)	E10.0
41-50	Bias for combined flow and thickening, S, $-1 \le S \le 1$.	E10.0

The bias value determines the final scale factor, S_{final}, in the event that the thickness and flow limit diagram criteria are satisfied. In this case the scale factor for the load curve is given by:

$$S_{final} = \frac{1}{2}(1-S)\cdot SF + \frac{1}{2}(1+S)ST$$

Card 3 (6E10.0)

Columns	Quantity	Format
1-10	Minimum x-coordinate (Default=-1.E+20)	E10.0
11-20	Maximum x-coordinate (Default= 1.E+20)	E10.0
21-30	Minimum y-coordinate (Default=-1.E+20)	E10.0
31-40	Maximum y-coordinate (Default= 1.E+20)	E10.0
41-50	Minimum z-coordinate (Default=-1.E+20)	E10.0
51-60	Maximum z-coordinate (Default= 1.E+20)	E10.0

91. Beam Force and Moment Release Cards

Define the number of nodal constraint sets, NBRC (Control Card 4, columns 71-75). The number of nodes in any set must not exceed the maximum number of nodes, MXN (Control Card 4, columns 76-80) and all nodes in the set must be coincident. A unique nodal point ID must be defined for the beam at the location where the beam force and moment resultants are released. The degrees-of-freedom which are not released are constrained to the other nodes in the set. The released degree-of-freedom can be either global or local relative to the local beam coordinate system which is stored for the nodal point with the beam data.

Constraint equations are used to join the nodal points together with the proper release conditions imposed. **Consequently, nodal points 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.

Please note that this option may lead to nonphysical constraints if the translational degrees-of-freedom are released.

Card 1
(I5)

Columns	Quantity	Format
1-5	Number of nodes in set. These nodes must be coincident.	I5

Card 2 (Repeat for Each Node in Set)
(I10,4I5)

Columns	Quantity	Format
1-10	Node ID.	I10

Beam Force and Moment Release

Columns	Quantity	Format
11-15	Release conditions for translations 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)	I5
16-20	Translational coordinate system EQ.0: defaults to global coordinate system EQ.1: global coordinate system EQ.2: local coordinate system	I5
21-25	Release conditions for rotations 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)	I5
26-30	Rotational coordinate system EQ.1: global coordinate system EQ.2: local coordinate system (default)	I5

92. Rigid/Deformable Material Switching

Define the input for this section if IRDMS in Columns 46-50 of Control Card 13 equals 2 or 3. Materials that are switched from deformable to rigid may later be changed back to deformable. If IRDMS equals 3 then define sets of materials for automatic material switching.

Card 1 (I5)

Columns	Quantity	Format
1-5	NDEFR, number of materials to switch from deformable to rigid plus the number of rigid materials for which a new master rigid body is defined.	I5

Cards 2,...NDEFR+1 (2I5) or (2I10) if MLARG

Skip this card if NDEFR=0.

Columns		Quantity	Format
1-5	(1-10)	The identification number of a material which is switched to a rigid material.	I5 (I10)
6-10	(11-20)	The identification number of the master rigid body to which the material is merged. If zero, the material becomes either an independent or master rigid body.	I5 (I10)

Card NDEFR+2 (I5)

Insert a blank card here if no inertial properties are to be defined.

Columns	Quantity	Format
1-5	NRBIPS, number of inertial property sets to be stored with specified rigid (not deformable) bodies and used 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. The properties which are input in this section are continuously updated to account for rigid body rotations for use when LS-DYNA is restarted. **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.**	I5

Rigid/Deformable Material Switching

Define the NRBIPS sets below. Skip the following two cards if NRBIPS=0.

Card NDEFR+3+... (5X,4E10.0,I10)

Note: All data must be provided.

Columns	Quantity	Format
1-5	Blank	5X
6-15	x-coordinate of center of mass	E10.0
16-25	y-coordinate of center of mass	E10.0
26-35	z-coordinate of center of mass	E10.0
36-45	Translational mass	E10.0
46-55	Material number of rigid body	I10

Card 2 (6E10.0)

Columns	Quantity	Format
1-10	I_{xx}, xx component of inertia tensor	E10.0
11-20	I_{xy}	E10.0
21-30	I_{xz}	E10.0
31-40	I_{yy}	E10.0
41-50	I_{yz}	E10.0
51-60	I_{zz}	E10.0

The inertia tensor is defined in the global coordinate system.

Define the following sets of cards if IRDMS (see Control Card 13, Cols. 46-50) equals 3. This option allows automatic material switching to take place without a restart.

Automatic Material Switching Control Card 1 (I10)

Columns	Quantity	Format
1-10	NSETS, number of sets for automatic material switching.	I10

Define NSETS of materials for rigid/deformable switching.

Automatic Material Switching Control Card 2 (2I5,3E10.0,2I5)

Columns	Quantity	Format
1-5	Set number for this switch	I5
6-10	Activation switch code. Define the test to activate the automatic material switch. EQ.0: switch takes place at time1 EQ.1: switch takes place between time 1 and time 2 if rigid wall (specified below) force is zero. EQ.2: switch takes place between time 1 and time 2 if contact surface (specified below) force is zero. EQ.3: switch takes place between time 1 and time 2 if rigid wall (specified below) force is non-zero. EQ.4: switch takes place between time 1 and time 2 if contact surface (specified below) force is non-zero.	I5
11-20	Time 1. Switch will not take place before this time	E10.0
21-30	Time 2. Switch will not take place after this time EQ.0.0: Time 2 set to 1.0e20	E10.0
31-40	Time 3. Period of time after activation of this switch for which another automatic material switch may not take place. If set to zero a material switch may take place immediately after this switch.	E10.0
41-45	Rigid wall / Contact surface number for switch codes 1,2,3,4	I5
46-50	Related switch set If specified the material switching being defined cannot be activated until after the related switch set has been activated and switched.	I5

51-55	Define a pair of related switches. EQ. 0: not paired EQ. 1: Master switch paired with switch RELSW. EQ.-1: Slave switch paired with switch RELSW.	I5

Notes:

Only surface to surface type contacts can be used to activate an automatic material switch. Contact surface and rigid wall numbers refer to the position they are defined within the input file.

Automatic Material Switching Control Card 3 (3I5,E10.0)

Columns	Quantity	Format
1-5	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. If this flag is set to zero, the nodal rigid bodies and generalized welds which become part of a rigid body after switching occurs are automatically turned off. This may lead to problems if the nodal rigid body has nodes that belong to a rigid body. EQ.0: automatic as described above. EQ.1: delete EQ.2: activate	I5
6-10	Flag to delete or activate nodal constraint sets If nodal constraint/spot weld definitions are active in the deformable bodies that are switched to rigid, then the definitions should be deleted to avoid instabilities. EQ.0: no change EQ.1: delete EQ.2: activate	I5
11-15	Flag to delete or activate rigid walls EQ.0: no change EQ.1: delete EQ.2: activate	I5
16-25	Maximum permitted time step size after restart	E10.0

Automatic Material Switching Control Card 4 (2I5)

Columns	Quantity	Format
1-5	NDEFR, number of materials to switch from deformable to rigid plus the number of rigid bodies for which new master bodies are defined.	I5
6-10	Number of materials to switch from rigid to deformable, NRDEF	I5

NDEFR Deformable to Rigid Cards (2I5) or (2I10) if MLARG

Define NDEFR cards here. Skip these cards if NDEFR=0.

Columns		Quantity	Format
1-5	(1-10)	The identification number of either a material which is switched to a rigid material or a rigid material which will become a slave to a master rigid body.	I5 (I10)
6-10	(11-20)	The identification number of the master rigid body to which the material is merged. If zero, the material becomes either an independent or master rigid body.	I5 (I10)

NRDEF Rigid to Deformable Cards (I5) or (I10) if MLARG

Define NRDEF cards here. Skip these cards if NRDEF=0.

Columns		Quantity	Format
1-5	(1-10)	The identification number of a material which is switched from a rigid to a deformable material. This must be a material which was switched to a rigid material in the initial input or in a previous restart . When a master rigid body is switched to a deformable material, its slaves become independent rigid bodies.	I5 (I10)

93. Velocity Reinitialization After DR

Define this section if and only if the initial condition parameter, INITV, on Control Card 11, columns 1-5 is equal to 7. The velocities are initialized in the order the cards are defined. Later cards may overwrite the velocities previously set. This section may be used to reset the velocites of nodes attached to deformable materials. *Rigid body nodal velocities must not be reset in this section.*

After a spinning body has completed dynamic relaxation for its static stresses due to centrifugal forces, the velocity field defined in Section 30 is reimposed. However, this field is no longer correct since the body has stretched during the DR process. The differences between the velocity field based on the initial geometry and the deformed geometry may create significant stress oscillations. To overcome this problem, the rotational velocity field defined in this section is based on the deformed geometry.

(INITV=7) Card 1 (I5)

Columns	Quantity	Format
1-5	Number of material card sets to be read, NMSETS	I5
6-10	Number of nodal card sets to be read, NNSETS	I5

(INITV=7) Define NMSETS Card Sets Below

Card 1 (I10,7E10.0)

Columns	Quantity	Format
1-10	Material ID EQ.0: Set all nodal velocities EQ.n: Initialize velocities of material n only	I10
11-20	Angular velocity	E10.0
21-30	x-coordinate on rotational axis	E10.0
31-40	y-coordinate on rotational axis	E10.0
41-50	z-coordinate on rotational axis	E10.0

Columns	Quantity	Format
51-60	x-direction cosine	E10.0
61-70	y-direction cosine	E10.0
71-80	z-direction cosine	E10.0

Card 2 (3E10.0)

Columns	Quantity	Format
1-10	x-rigid body velocity	E10.0
11-20	y-rigid body velocity	E10.0
21-30	z-rigid body velocity	E10.0

(INITV=7) Define NNSETS Card Sets Below

Card 1 (I10,7E10.0)

Columns	Quantity	Format
1-10	Number of nodal points to be defined below.	I10
11-20	Angular velocity	E10.0
21-30	x-coordinate on rotational axis	E10.0
31-40	y-coordinate on rotational axis	E10.0
41-50	z-coordinate on rotational axis	E10.0
51-60	x-direction cosine	E10.0
61-70	y-direction cosine	E10.0
71-80	z-direction cosine	E10.0

Card 2 (3E10.0)

Columns	Quantity	Format
1-10	global x-rigid body velocity	E10.0
11-20	global y-rigid body velocity	E10.0
21-30	global z-rigid body velocity	E10.0

Nodal Point ID's (10I8)

Define NNOD nodal points with ten nodal ID's per card.

Columns	Quantity	Format
1-8	Slave node n_1	I8
9-16	Slave node n_2	I8
. .	.	.
. .	.	.
. .	.	.

RESTART INPUT DECK (Optional)

A complete input deck is generally not needed to restart LS-DYNA; however, a smaller optional restart input deck may be used to reset:

- termination time,
- output printing interval,
- output plotting interval.

All changes made when restarting will be reflected in subsequent restart dumps.

If a restart is made using a complete LS-DYNA input file, the resulting families files will be re-initialized (D3PLOT01, etc.) and a new sequence of files will be created.

1. Title Card
(12A6,A2,A1,A5)

Columns	Quantity	Format
1-72	Heading to appear on output	12A6
73-74	Input code version for full deck restart EQ.87: input follows manual published in 1987 EQ.88: input follows manual published in 1989 EQ.90: input follows manual published in 1990 EQ.91: input follows manual published in 1991 EQ.92: input follows manual published in 1992 EQ.93: input follows this manual	A2
75	Version number EQ.0: for versions 87, 88, 89, 91, 92 and 93 EQ.4: for versions 903, 904, 905 and 906 EQ. i: activates reading of Implicit Control Cards	A1
76-80	Input format "LARGE" - Large input format node and element numbers upto 99999999 may be used. "MLARG" - As "LARGE" but material (part) numbers upto 99999999 may be used.	A5

2. Control Cards

Control Card 1

(3E10.0,I2,I3,6I5,E10.0,I5)

Columns	Quantity	Format
1-10	New termination time EQ.0.0: termination time remains unchanged	E10.0
11-20	New output printing interval EQ.0.0: output printing interval remains unchanged for timehistory data	E10.0
21-30	New output plotting interval EQ.0.0: output plotting interval remains unchanged	E10.0
31-32	ND2ASL, number of *2D automatic* contact interfaces to be eliminated or activated (≤ 48). (This option is for two-dimensional problems only.)	I5
33-35	NDSL, number of sliding interfaces to be eliminated or activated (≤ 48)	I5
36-40	Number of materials to be eliminated	I5
41-45	Number of solid element blocks to be eliminated	I5
46-50	Number of beam element blocks to be eliminated	I5
51-55	Number of 4-node shell element blocks to be eliminated	I5
56-60	Number of 8-node thick shell element blocks to be eliminated	I5
61-65	Number of load curves to be redefined, NRDLC	I5
66-75	New time step size for mass scaled calculation. Mass scaling must be active in the time zero run.	E10.0
76-80	Basis of time step 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/(longest side) EQ.1: characteristic length=area/(longest diagonal) EQ.2: based on bar wave speed and max (shortest side,area/longest side)	I5

Control Card 2
(E10.0,5I5,I10,5I5)

Columns	Quantity	Format
1-10	New time step scale factor EQ.0.0: time step scale factor remains unchanged	E10.0
11-15	Number of changed translational boundary condition codes	I5
16-20	Number of materials for stress initialization. With this option an entire LS-DYNA input deck is required as part of the restart input. Materials and sliding interfaces can be added to or deleted from the calculation by using this option. Although many parameters can be changed, there are some restrictions. EQ.-1: all materials will be initialized.	I5
21-25	Number of cycles between restart dumps. EQ.0: no change	I5
26-30	Number of cycles between running restart dumps EQ.0: no change	I5
31-35	General printout option flag EQ.0: no change EQ.1: read in three cards below defining printout intervals for various databases	I5
36-45	Termination cycle	I10
46-50	Flag for rigid/deformable material switching, IDRMS EQ.1: off EQ.2: on, read switched materials below EQ.3: on, read switched materials below with supplemental input	I5
51-55	Load curve number that limits the maximum time step size LT.0: discontinue using load curve EQ.0: no change EQ.n: use load curve number n	I5
56-60	Number of interfaces to set the small penetration search option, NSPSO	I5
61-65	Flag to read dynamic relaxation data (See Section 15 below), IDRA EQ.0: off EQ.1: on	I5
66-70	Number of geometric contact entities to delete	I5

Columns	Quantity	Format
71-75	Modify velocity flag, MNVF EQ.0: no change EQ.1: zero velocity after restart EQ.2: new velocities are read in EQ.3: all nodes are given the same velocity EQ.4: as 2 but exempted nodes are defined EQ.5: Box option EQ.6: generation with arbitrary node numbering (not recommended) EQ.7: rotational and translational via material/node ID	I5
76-80	Implicit coupling for springback calculations EQ.0: off EQ.1: on EQ.2: on, activate seamless switch to implicit LS-DYNA	I5

When initializing stresses there are some limitations. Initial velocities in the new input deck are ignored. Instead the velocity field in the restart dump file is used. Element formulations cannot be changed in a full start restart, i.e., beams cannot be changed to truss elements, shells to membranes, or Hughes-Liu formulations to that of Belytschko-Tsay. Furthermore, the number of integration points in the element formulations must also remain the same as well as other section properties. The addition of lumped masses or section areas will result in a sudden jump in kinetic energy. Material types must remain the same after a restart; however, the material constants can usually be altered without causing major problems.

Control Card 3
(9I5,E10.0,I5)

Columns	Quantity	Format
1-5	Flag for discrete element stress initialization. All discrete elements will be initialized if this flag is nonzero. This option is ignored if the flag in column 20 of the second control card is set to a -1. EQ.0: not initialized NE.0: initialized	I5
6-10	Flag for seat belt element stress initialization. All belts, retractors, etc. will be initialized if this flag is nonzero. This option is ignored if the flag in column 20 of the second control card is set to a -1. EQ.0: not initialized NE.0: initialized	I5
11-15	Number of rigid bodies with modified restraint codes	I5
16-20	Number of cycles between printing information to the D3HSP file. EQ.0: no change	I5
21-25	Number of rigid body stoppers to be redefined, NCRBSTP. EQ.0: no change	I5
26-30	Number of nodal displacement termination conditions to be read EQ.0: no change EQ.-1: turn this feature off	I5
31-35	Number of displacement termination conditions to be read for rigid bodies EQ.0: no change EQ.-1: turn this feature off	I5
36-40	Number of rigid bodies with modified velocities.	I5
41-45	Flag to read in modified thermal parameters. EQ.0: do not read in. NE.0: read in parameters on two extra control cards.	I5
46-55	New output plotting interval to contact force file EQ.0.0: output printing interval remains unchanged.	E10.0
56-60	Number of contact termination conditions to be read. EQ.0: no change EQ.-1: turn this feature off	I5

Cards 4, 5, and 6 (8E10.0)

Define these cards only if printout option on Card 2 above is flagged. If the new output intervals are defined as 0.0 then the output interval is unchanged. To stop output to one of these files set the output interval to a high value.

Columns		Quantity	Format
1-10	Card 4	Output interval for cross-section forces	E10.0
11-20		Output interval for rigid wall forces	E10.0
21-30		Output interval for nodal point data	E10.0
31-40		Output interval for element data	E10.0
41-50		Output interval for global data	E10.0
51-60		Output interval for discrete elements	E10.0
61-70		Output interval for material energies	E10.0
71-80		Output interval for nodal interface forces	E10.0
1-10	Card 5	Output interval for resultant interface forces	E10.0
11-20		Output interval for smug animator instant	E10.0
21-30		Output interval for spc reaction forces	E10.0
31-40		Output for nodal constraint resultants (spotwelds and rivets)	E10.0
41-50		Output interval for airbag statistics	E10.0
51-60		Output interval for AVS database	E10.0
61-70		Output interval for nodal force groups	E10.0
71-80		Output interval for boundary condition forces and energy on nodal points with discrete forces, pressures, or designated velocities	E10.0

Columns	Quantity		Format
1-10	Card 6	Output interval for rigid body data	E10.0
11-20		Output interval for geometric contact entities	E10.0
21-30		Output interval for MPGS database	E10.0
31-40		Output interval for MOVIE database	E10.0
41-50		Output interval for sliding interface database	E10.0
51-60		Output interval for seat belt database	E10.0
61-70		Output interval for joint forces	E10.0
71-80		Output interval for TRHIST data	E10.0

By defining the output interval a file is created for the output. Each output type is placed into a separate file. Normally these names are assigned by LS-DYNA. Using the "W=" option on startup, a root name can be specified. Extensions are then added to this root to form the output file names. The file names and corresponding unit numbers are:

	I/O UNIT #	FILE NAME
Cross-section forces	i/o unit#31	SECFORC
Rigidwall forces	i/o unit#32	RWFORC
Nodal point data	i/o unit#33	NODOUT
Element data	i/o unit#34	ELOUT
Global data	i/o unit#35	GLSTAT
Discrete elements	i/o unit#36	DEFORC
Material energies	i/o unit#37	MATSUM
Nodal interface forces	i/o unit#38	NCFORC
Resultant interface forces	i/o unit#39	RCFORC
Smug animator database	i/o unit#40	DEFGEO
Nastran/BDF file	i/o unit#49	NASBDF (see comment below)
SPC reaction forces	i/o unit#41	SPCFORC
Nodal constraint resultants (spotwelds/rivets)	i/o unit #42	SWFORC
Airbag statistics	i/o unit #43	ABSTAT
ASCII database	i/o unit #44	AVSFLT
Nodal force group	i/o unit #45	NODFOR
Boundary conditions nodal forces and energies	i/o unit #46	BNDOUT
Rigid body data	i/o unit #47	RBDOUT
Contact entities	i/o unit #48	GCEOUT
MPGS file family	i/o unit #50	MPGSnnn.xxx where nnn=001-999
MOVIE file family	i/o unit #50	MOVIEnnn.xxx where.nnn=001-999
Interface energies	i/o unit #51	SLEOUT
Seat belts	i/o unit #52	SBTOUT
Joint forces	i/o unit #53	JNTFORC

Upon restart these files are recreated if and only if they are not on disk. Otherwise, they are opened, read, and the new output continues at the bottom of the file.

(Restart) Control Cards

Define the following two control cards if the flag to read in modified thermal control parameters on restart control card 3 is non zero. The following parameters apply to a coupled structural/thermal or a thermal only analysis.

Thermal Control Card 1 (I5,5E10.0)

Columns	Quantity	Format
1-5	Time step code EQ.0: No change EQ.1: Fixed timestep EQ.2: variable timestep	I5
6-15	Thermal time step on restart EQ.0.0: No change	E10.0
16-25	Minimum thermal timestep EQ.0.0: No change	E10.0
26-35	Maximum thermal timestep EQ.0.0: No change	E10.0
36-45	Maximum temperature change in each timestep EQ.0.0: No change	E10.0
46-55	Time step control Parameter (p) EQ.0.0: No change $0.0 < p < 1$	E10.0

Thermal Control Card 2 (I5,2E10.0)

Columns	Quantity	Format
1-5	Maximum number of reformations per time step.	I5
6-15	Non-linear convergence tolerance EQ.0.0: No change	E10.0
16-25	Print interval to TPRINT file EQ.0.0: No change	E10.0

Implicit Control Card 1: General Data
(Input for Implicit Analysis only, code version "i" on title card)
(I5,E10.0)

Columns	Quantity	Format
1-5	Implicit/Explicit switching flag EQ. 0: explicit analysis EQ. 1: implicit analysis EQ. 2: explicit followed by one implicit step (*"springback analysis"*)	I5
6-15	Initial time step size for implicit solution	E10.0
16-20	Element formulation switching flag EQ. 1: switch to fully integrated formulation for implicit phase of springback analysis (DEFAULT) EQ. 2: retain original element formulation	I5
21-25	Number of steps for nonlinear springback EQ. 0: DEFAULT = 1	I5
26-30	Geometric (initial stress) stiffness flag EQ. 1: include EQ. 2: ignore (DEFAULT)	I5

Implicit Control Card 2: Nonlinear Solver
(Input for Implicit Analysis only, code version "i" on title card)
(3I5,4E10.0,3I5)

Columns	Quantity	Format
1-5	Nonlinear solution method for implicit analysis EQ.1: linear EQ.2: nonlinear with BFGS updates (DEFAULT)	I5
6-10	Iteration limit between automatic stiffness reformations EQ.0: DEFAULT = 11	I5
11-15	Stiffness reformation limit per time step EQ.0: DEFAULT = 15	I5
16-25	Displacement convergence tolerance EQ.0: DEFAULT = 0.001	E10.0
26-35	Energy convergence tolerance EQ.0: DEFAULT = 0.01	E10.0
36-45	(blank)	E10.0
46-55	Line Search convergence tolerance EQ.0: DEFAULT = 0.90	E10.0
56-60	Displacement norm for convergence test EQ.1: increment vs. displacement over current step EQ.2: increment vs. total displacement (DEFAULT)	I5
61-65	Divergence flag EQ.1: reform stiffness if divergence detected (DEFAULT) EQ.2: ignore divergence during equilibrium iterations	I5
66-70	Initial stiffness formation flag EQ.1: reform stiffness at start of each step (DEFAULT) EQ.n: reform at start of every "n"th step	I5

Implicit Control Card 3: Linear Solver
(Input for Implicit Analysis only, code version "i" on title card)
(3I5)

Columns	Quantity	Format
1-5	Linear equation solver EQ.1: sparse, direct, automatic out-of-core (DEFAULT) EQ.2: sparse, direct, incore	I5
6-10	Linear solver print flag EQ.1: timing summary at end of output file (DEFAULT) EQ.2: timing, storage information to screen, output file	I5
11-15	Negative eigenvalue flag EQ.1: stop or retry step if negative eigenvalues detected EQ.2: print warning message, try to continue (DEFAULT)	I5

Implicit Control Card 4: Auto Time Step Control
(Input for Implicit Analysis only, code version "i" on title card)
(3I5,2E10.0)

Columns	Quantity	Format
1-5	Auto time step control flag EQ.0: constant time step size EQ.1: automatically adjusted step size	I5
6-10	Optimum iteration count per time step EQ.0: DEFAULT = 11	I5
11-15	Allowable iteration window EQ.0: DEFAULT = 5	I5
16-25	Minimum time step size EQ.0: DEFAULT = 0.001 * DT	E10.0
26-35	Maximum time step size EQ.0: DEFAULT = 10 * DT	E10.0
36-45	Blank	10X
46-50	Artificial Stabilization Flag, ASFLAG EQ.1: active for all deformable shell elements (DEFAULT for *springback analysis*) EQ.2: inactive (DEFAULT for *standard analysis*)	I5
51-60	Scale factor for Artificial Stabilization LT.0.0: absolute value gives load curve for scale factor vs. time EQ.0.0: DEFAULT = 1.0	E10.0
61-70	Time when Artificial Stabilization begins EQ.0.0: DEFAULT = immediately upon entering IMPLICIT mode	E10.0
71-80	Time when Artificial Stabilization ends EQ.0.0: DEFAULT = termination time	E10.0

Implicit Control Card 5. (blank)
(Input for Implicit Analysis only, code version "i" on title card)

Columns	Quantity	Format
	(this card entirely blank)	

Implicit Control Card 6. (blank)
(Input for Implicit Analysis only, code version "i" on title card)

Columns	Quantity	Format
	(this card entirely blank)	

3. Sliding Interfaces Small Penetration Option
(16I5), or (10I8) for LARGE option

Skip this section if there are no sliding interfaces flagged.

Columns		Quantity	Format
1-5	(1-8)	Number of first sliding interface to flag	I5 (I8)
6-10	(9-16)	Number of second sliding interface to flag	I5 (I8)
10-15	(17-24)	Number of third sliding interface to be flag	I5 (I8)
.	.	.	.
.	.	.	.
.	.	.	.

4. Deleted/Activated Sliding Interfaces
(16I5), or (10I8) for LARGE option

Skip this section if ND2DASL=0 in columns 32-32 of the first restart control card. This input applies only to 2D automatic contact definitions only.

Columns		Quantity	Format
1-5	(1-8)	Number of first sliding interface to be deleted or activated. For activation input the interface identification as a negative number.	I5 (I8)
6-10	(9-16)	Number of second sliding interface to be deleted or activated. For activation input the interface identification as a negative number.	I5 (I8)
10-15	(17-24)	Number of third sliding interface to be deleted or activated. For activation input the interface identification as a negative number.	I5 (I8)
.	.	.	.
.	.	.	.
.	.	.	.

Skip this section if NDSL=0 in columns 33-35 of the first restart control card. This input applies to 3D contact interfaces and to the non automatic 2D contact definitions.

Columns		Quantity	Format
1-5	(1-8)	Number of first sliding interface to be deleted or activated. For activation input the interface identification as a negative number.	I5 (I8)
6-10	(9-16)	Number of second sliding interface to be deleted or activated. For activation input the interface identification as a negative number.	I5 (I8)
10-15	(17-24)	Number of third sliding interface to be deleted or activated. For activation input the interface identification as a negative number.	I5 (I8)
.	.	.	.
.	.	.	.
.	.	.	.

5. Deleted Contact Entities
(16I5), or (10I8) for LARGE option

Skip this section if there are no deleted contact entities.

Columns		Quantity	Format
1-5	(1-8)	Number of first contact entity to be deleted	I5 (I8)
6-10	(9-16)	Number of second contact entity to be deleted	I5 (I8)
10-15	(17-24)	Number of third contact entity to be deleted	I5 (I8)
.	.	.	.
.	.	.	.
.	.	.	.

6. Deleted Materials
(16I5), or (10I8) for LARGE option

Skip this section if there are no deleted material blocks.

Columns		Quantity	Format
1-5	(1-8)	Number of first material to be deleted	I5 (I8)
6-10	(9-16)	Number of second material to be deleted	I5 (I8)
10-15	(17-24)	Number of third material to be deleted	I5 (I8)
.	.	.	.
.	.	.	.
.	.	.	.

7. Deleted Solid Element Blocks
(16I5), or (10I8) for LARGE option

Skip this section if there are no deleted solid elements.

Columns		Quantity	Format
1-5	(1-8)	First element of first block to be deleted	I5 (I8)
6-10	(9-16)	Last element of first block to be deleted	I5 (I8)
10-15	(17-24)	First element of second block to be deleted	I5 (I8)
16-20	(25-32)	Last element of second block to be deleted	I5 (I8)
.	.	.	.
.	.	.	.
.	.	.	.

8. Deleted Beam Element Blocks
(16I5), or (10I8) for LARGE option

Skip this section if there are no deleted beam elements.

Columns		Quantity	Format
1-5	(1-8)	First element of first block to be deleted	I5 (I8)
6-10	(9-16)	Last element of first block to be deleted	I5 (I8)
10-15	(17-24)	First element of second block to be deleted	I5 (I8)
16-20	(25-32)	Last element of second block to be deleted	I5 (I8)
.	.	.	.
.	.	.	.
.	.	.	.

9. Deleted Shell Element Blocks

(16I5), or (10I8) for LARGE option

Skip this section if there are no deleted shell elements.

Columns		Quantity	Format
1-5	(1-8)	First element of first block to be deleted	I5 (I8)
6-10	(9-16)	Last element of first block to be deleted	I5 (I8)
10-15	(17-24)	First element of second block to be deleted	I5 (I8)
16-20	(25-32)	Last element of second block to be deleted	I5 (I8)
.	.	.	.
.	.	.	.
.	.	.	.

10. Deleted Thick Shell Element Blocks
(16I5), or (10I8) for LARGE option

Skip this section if there are no deleted thick shell elements.

Columns		Quantity	Format
1-5	(1-8)	First element of first block to be deleted	I5 (I8)
6-10	(9-16)	Last element of first block to be deleted	I5 (I8)
10-15	(17-24)	First element of second block to be deleted	I5 (I8)
16-20	(25-32)	Last element of second block to be deleted	I5 (I8)
.	.	.	.
.	.	.	.
.	.	.	.

11. Changed Boundary Condition Cards
(I5,F5.0), or (I8,F5.0) for LARGE option

Skip this section if there are no changed translational boundary condition codes

Columns		Quantity	Format	
1-5	(1-8)	Nodal point number	I5	(I8)
6-10	(9-16)	New boundary condition code EQ.0: no constraints EQ.1: constrained x EQ.2: constrained y EQ.3: constrained z EQ.4: constrained x and y EQ.5: constrained y and z EQ.6: constrained z and x EQ.7: constrained x, y, and z	F5.0	(F5.0)

12. Changed Rigid Body Constraints
(3I5), or (3I10) for MLARG option

Skip this section if the number of rigid bodies with changed restraints is zero on Control Card 3.

Columns		Quantity	Format
1-5	(1-10)	Rigid body part ID	I5 (I10)
6-10	(11-20)	Type of restraint EQ.1: translational EQ.2: rotational	I5 (I10)
10-15	(21-30)	Direction of restraint EQ.0: no constraints EQ.1: constrained x EQ.2: constrained y EQ.3: constrained z EQ.4: constrained x and y EQ.5: constrained y and z EQ.6: constrained z and x EQ.7: constrained x, y, and z	I5 (I10)

13. Material Initialization
(2I5)

Skip this section if the number of materials to be initialized is zero or equal to -1. This section tells LS-DYNA how to remap stresses and deformations from materials in the restart file to the model described in the new input deck. LS-DYNA assumes a one-to-one correlation between the nodes and elements of the materials which are to be remapped from the restart to the new run.

Columns		Quantity	Format
1-5	(1-10)	Material identification number used in dump file	I5 (I10)
6-10	(11-20)	Material identification number used in new input file	I5 (I10)

Repeat as above for each material to be initialized. Include an entire LS-DYNA input description file appended after the restart input (omit only the Title Card), which contains all the changes for the restarted problem. Note that values in this portion of the restart file override previously specified values, i.e., termination time on Card 2 above is overridden by the value on the normal LS-DYNA control cards defined below.

14. Load Curve Cards

Redefine the number, NRDLC, of load curve sets specified. Repeat the following cards for redefined each set:

Card 1 (2I5)

Columns	Quantity	Format
1-5	Load curve number of redefined curve	I5
6-10	Number of points in load curve (must not change), NPTS	I5

Card 2,...,NPTS+1 (2E10.0)

Columns	Quantity	Format
1-10	Time	E10.0
11-20	Load or function value	E10.0

15. Damping/Dynamic Relaxation

Define this card if and only if IDRA is set to 1.

Columns	Quantity	Format
1-5	Load curve number which specifies system damping constant, LCDAMP EQ.0: no damping EQ.-1: system damping is defined for each material by load curves. The data are defined in a separate input section called “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.	I5
6-15	System damping constant, d (this option is bypassed if the load curve number defined above is nonzero), VALDMP	E10.0
16-20	Dynamic relaxation flag for stress initialization, IDRFLG EQ.0: inactive EQ.1: dynamic relaxation is activated	I5
21-25	Number of iterations between convergence checks, for dynamic relaxation option (default = 250), NRCYCK	I5
26-35	Convergence tolerance for dynamic relaxation option, DRTOL (default = 0.001)	E10.0
36-45	Dynamic relaxation factor (default = .995), DRFCTR	E10.0
46-55	Optional termination time for dynamic relaxation, DRTERM. Termination occurs at this time or when convergence is attained. (Default = infinity)	E10.0
56-65	Scale factor for computed time step during dynamic relaxation, TSSFDR. If zero, the value is set to TSSFAC defined on Card 8. After converging, the scale factor is reset to TSSFAC.	E10.0
66-70	Automatic control for dynamic relaxation option based on algorithm of Papadrakakis, IRELAL [Papadrakakis 1981] EQ.0: inactive EQ.1: active	I5
71-80	Convergence tolerance on automatic control of dynamic relaxation, EDTTL	E10.0

(Restart) Damping/Dynamic Relaxation

After restart LS-DYNA continues the normal output of data as it would without the dynamic relaxation being active. Unlike the dynamic relaxation phase at the beginning of the calculation, a separate database is not used. Only load curves that are flagged for dynamic relaxation are applied after restarting.

16. Implicit Springback Solution

Card 1 (2I5)

Columns	Quantity	Format
1-5	Number of materials to be included in the springback calculation, NMSB	I5
6-10	Number of additional nodal constraints to be added, NANC	I5
11-15	Number of trim curves, NTC.	I5

Read the following card if and only if NMSB is nonzero.

Card 2,... (8I10) [Use as many cards as necessary]

Columns	Quantity	Format
1-10	First material to be included	I10
11-20	Second material	I10
21-30	Third material	I10
31-40	Fourth material	I10
.		
.		
.		
71-80	Eighth material	I10

Card 2+NMSB,... (I10,2E10.0) [Define NANC cards]

Columns	Quantity	Format
1-10	Node number	I10
11-20	Displacement boundary condition code 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	E10.0
21-30	Rotational boundary condition code 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	E10.0

Trim Cruve Data Cards

Define NTC cards, i.e., one for each curve (2I5).

Columns	Quantity	Format
1-5	Load curve number giving trim curve (general x-y data type 2)	I5
6-10	Trim flag EQ.-1: remove elements inside curve EQ. 1: remove elements outside curve	I5

Trim curves must be closed loops. If the first and last points do not coincide, LS-DYNA will generate one additional segment to close the curve. Load curves must be type # 2, the general x-y data type.

17. Changed Rigid Body Stoppers

Redefine NCRBSTP card sets here, see Columns 31-35 on Control Card 3. Optional slaved rigid bodies are defined within the card set. New stopper definitions cannot be introduced in this section. Existing stoppers can be modified.

Card 1 (7I5,3E10.0) or (I10,6I5,3E10.0) if MLARG

Columns		Quantity	Format
1-5	(1-10)	Part ID of master rigid body.	I5 (I10)
6-10	(11-15)	LCMAX, load curve ID defining the maximum coordinate as a function of time. EQ.0: no limitation of the maximum displacement	I5 (I5)
11-15	(16-20)	LCMIN, load curve ID defining the minimum coordinate as a function of time. EQ.0: no limitation of the minimum displacement	I5 (I5)
16-20	(21-25)	NMXRB, number of rigid bodies that are slaved in the maximum coordinate direction to the master rigid body. This option requires additional input below. This number should not exceed the maximum used in any stopper definition in the first run.	I5 (I5)
21-25	(26-30)	NMNRB, number of rigid bodies that are slaved in the minimum coordinate direction to the master rigid body. This option requires additional input below. This number should not exceed the maximum used in any stopper definition in the first run.	I5 (I5)
26-30	(31-35)	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	I5 (I5)
31-35	(36-40)	Direction stopper acts in EQ.1: x-translation EQ.2: y-translation EQ.3: z-translation EQ.4: v-translation (v is defined by vx,vy,vz) EQ.5: x-axis rotation EQ.6: y-axis rotation EQ.7: z-axis rotation EQ.8: v-axis rotation (v is defined by vx,vy,vz)	I5 (I5)

Card 1 continued

Columns		Quantity	Format	
36-45	(41-50)	x-component of v-vector	E10.0	(E10.0)
46-55	(41-50)	y-component of v-vector	E10.0	(E10.0)
56-65	(41-50)	z-component of v-vector	E10.0	(E10.0)

Card 2 (2E10.0)

Columns	Quantity	Format
1-10	Birth time	E10.0
11-20	Death time (default=10^{28})	E10.0

Immediately following the two cards above define NMXRB slaved rigid bodies to the maximum coordinate followed by NMNRB slaved rigid bodies for the minimum coordinate. Skip the input if no rigid bodies are slaved.

Optional cards required if NMXRB > 0
Define NMXRB Cards with format (I10, E10.0)

Rigid bodies that are slaved in the maximum coordinate direction of the master rigid body.

Columns	Quantity	Format
1-10	Slave rigid body part ID, m_1	I10
11-20	Closure distance which activates 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 also stops. However, the slaved rigid body is free to move away from the master. EQ.0.0: slaved rigid body stops when master stops	E10.0

Optional cards required if NMNRB > 0
Define NMNRB Cards with format (I10, E10.0)

Rigid bodies that are slaved in the minimum coordinate direction of the master rigid body.

Columns	Quantity	Format
1-10	Slave rigid body part ID, m_1	I10
11-20	Closure distance which activates 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 also stops. However, the slaved rigid body is free to move away from the master. EQ.0.0: slaved rigid body stops when master stops	E10.0

18. Displacement Termination

In the first part of this section displacement termination is defined for nodal points where the number of cards specified on Control Card 3, col. 26-30 are provided. In the second part displacement termination is defined by a rigid body displacement where the number of cards specified on Control Card 3, col. 31-35 are provided. This second part is useful for rigid bodies with no user defined nodal points. In the third part contact termination is defined where the number of cards is specified on Control Card 3 in columns 56-60. This input completely overrides existing termination conditions defined in the time zero run.

Caution: The inputs are different for the nodal and rigid body stop conditions. For the nodal stop condition the global coordinates are input, and for the rigid body stop condition the relative global translations are input.

Cards 1,...,NUMSTOP (2I5,2E10.0), or (I8,I5,2E10.0) for LARGE option

Columns		Quantity	Format	
1-5	(1-8)	Node ID	I5	(I8)
5-10	(9-13)	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	I5	(I5)
11-20	(14-23)	Maximum (most positive) coordinate, options 1, 2 and 3 above only	E10.0	E10.0
21-30	(24-33)	Minimum (most negative) coordinate, options 1, 2 and 3 above only	E10.0	E10.0

The analysis terminates when the current position of the node specified reaches either the maximum or minimum value (types 1, 2 or 3), or picks up force from any contact surface (type 4). If more than one condition is input, the analysis stops when any of the conditions is satisfied.

(Restart) Displacement Termination

Termination by rigid body displacement is defined below.

Cards 1,...,NRBEND (2I5,2E10.0), or (I8,I5,2E10.0) for LARGE option

Columns		Quantity	Format	
1-5	(1-8)	Rigid body ID	I5	(I8)
5-10	(9-13)	Stop criterion EQ.1: global x displacement EQ.2: global y displacement EQ.3: global z displacement EQ.4: displacement magnitude	I5	(I5)
11-20	(14-23)	Maximum (most positive) displacement, options 1, 2, 3, and 4 above	E10.0	E10.0
21-30	(24-33)	Minimum (most negative) displacement, options 1, 2, and 3 above.	E10.0	E10.0

The analysis terminates when the current displacement of the rigid body specified reaches either the maximum or minimum value. If more than one condition is input, the analysis stops when any of the conditions is satisfied.

Termination by contact is defined below.

Cards 1,...,NCNEND (I10,2E10.0)

Columns	Quantity	Format
1-10	Contact ID	I10
11-20	Activation time.	E10.0
21-30	Time duration of null resultant force prior to termination. This time is tracked only after the activation time is reached. EQ.0.0: Immediate termination after null force is detected.	E10.0

The analysis terminates when the magnitude of the contact interface resultant force is zero. If more than one condition is input, the analysis stops when any of the conditions is satisfied.

19. Modified Nodal Velocities

Skip this section if the modify nodal velocity flag (MNVF) on Control Card 2 is 0 or 1.

MNVF = 2 : New Velocities
MNVF = 3 : All nodes have same velocity
MNVF = 4 : As 3 but exempted nodes are defined
MNVF = 5 : Box option
MNVF = 6 : Generation with arbitrary node numbering
MNVF = 7 : Rotational and translational via material/node ID

(MNVF=2 and 6)
(I5,3E10.0,I5), or (I8,3E10.0,I5) for LARGE option

Columns		Quantity	Format	
1-5	(1-8)	Nodal point number	I5	(I8)
6-15	(9-18)	Initial velocity in x-direction	E10.0	(E10.0)
16-25	(19-28)	Initial velocity in y-direction	E10.0	(E10.0)
26-35	(29-38)	Initial velocity in z-direction	E10.0	(E10.0)
36-40	(39-43)	Node increment (no sorting) EQ.0: default set to 1 if MNVF=2, if MNVF=6 any missing nodal ID's between two consecutively defined nodes will have their velocities set.	I5	(I5)
41-50	(44-53)	x-angular velocity	E10.0	
51-60	(54-63)	y-angular velocity	E10.0	
61-70	(64-73)	z-angular velocity	E10.0	

If arbitrary node and element numbers are assumed (IARB=1) then each node must be defined for the case where MNVF=2.

(Restart) Modified Nodal Velocities

(MNVF=3 or 4)

Columns	Quantity	Format
1-10	x-velocity	E10.0
11-20	y-velocity	E10.0
21-30	z-velocity	E10.0
31-40	x-angular velocity	E10.0
41-50	y-angular velocity	E10.0
51-60	z-angular velocity	E10.0

The above velocity applies to all nodes if MNVF=3. If MNVF=4 then the nodes defined below are assigned unique velocities.

Define the following cards if and only if MNVF=4.

Cards 1 (I5)

Columns	Quantity	Format
1-5	NUMNOD, the number of nodes that are assigned velocities other than the default.	I5

Cards 2,...,NUMNOD+1 (I5,6E10.0)

Columns		Quantity	Format
1-5	(1-8)	Node numbers	I5 (I8)
6-15	(9-18)	x-velocity	E10.0
16-25	(19-28)	y-velocity	E10.0
26-35	(29-38)	z-velocity	E10.0
36-45	(39-48)	x-angular velocity	E10.0
46-55	(49-58)	y-angular velocity	E10.0
56-65	(59-68)	z-angular velocity	E10.0

Define the following cards if and only if MNVF=5.

Cards 1 (I5)

Columns	Quantity	Format
1-5	Number of boxes, NBOX	I5

Define the following NBOX card sets to specify the box dimensions and the velocity of nodal points lying within the box.

Cards 2,4,.... (6E10.0)

Columns	Quantity	Format
1-10	Xmin	E10.0
11-20	Xmax	E10.0
21-30	Ymin	E10.0
31-40	Ymax	E10.0
41-50	Zmin	E10.0
51-60	Zmax	E10.0

Cards 3,5,.... (3E10.0)

Columns	Quantity	Format
1-10	x-translational velocity	E10.0
11-20	y-translational velocity	E10.0
21-30	z-translational velocity	E10.0
31-40	x-angular velocity	E10.0
41-50	y-angular velocity	E10.0
51-60	z-angular velocity	E10.0

(Restart) Modified Nodal Velocities

(MNVF=7) Card 1 (I5)

Columns	Quantity	Format
1-5	Number of material card sets to be read, NMSETS	I5
6-10	Number of nodal card sets to be read, NNSETS	I5

(MNVF=7) Define NMSETS Card Sets Below

Card 1 (I10,7E10.0)

Columns	Quantity	Format
1-10	Material ID EQ.0: Set all nodal velocities EQ.n: Initialize velocities of material n only	I10
11-20	Angular velocity	E10.0
21-30	x-coordinate on rotational axis	E10.0
31-40	y-coordinate on rotational axis	E10.0
41-50	z-coordinate on rotational axis	E10.0
51-60	x-direction cosine	E10.0
61-70	y-direction cosine	E10.0
71-80	z-direction cosine	E10.0

Card 2 (3E10.0)

Columns	Quantity	Format
1-10	x-rigid body velocity	E10.0
11-20	y-rigid body velocity	E10.0
21-30	z-rigid body velocity	E10.0

(MNVF=7) Define NNSETS Card Sets Below

Card 1 (I10,7E10.0)

Columns	Quantity	Format
1-10	Number of nodal points to be defined below.	I10
11-20	Angular velocity	E10.0
21-30	x-coordinate on rotational axis	E10.0
31-40	y-coordinate on rotational axis	E10.0
41-50	z-coordinate on rotational axis	E10.0
51-60	x-direction cosine	E10.0
61-70	y-direction cosine	E10.0
71-80	z-direction cosine	E10.0

Card 2 (3E10.0)

Columns	Quantity	Format
1-10	global x-rigid body velocity	E10.0
11-20	global y-rigid body velocity	E10.0
21-30	global z-rigid body velocity	E10.0

Nodal Point ID's (10I8)

Define NNOD nodal points with ten nodal ID's per card.

Columns	Quantity	Format
1-8	Slave node n_1	I8
9-16	Slave node n_2	I8
. .	.	.
. .	.	.

20. Modified Rigid Body Velocities
(I5, 6E10.0)

Skip this section if the number of rigid bodies with modified velocities is 0 (Control Card 3).

Columns	Quantity	Format
1-5	Nodal point number	I5
6-15	x-velocity	E10.0
16-25	y-velocity	E10.0
26-35	z-velocity	E10.0
36-45	x-angular velocity	E10.0
46-55	y-angular velocity	E10.0
56-65	z-angular velocity	E10.0

21. Rigid/Deformable Material Switching

Define the materials for rigid/deformable switching below if IDRMS on control card 2 is set to 2 or 3.

Optional Card defined if IRDMS=3 only
(3I5,E10.0)

Columns	Quantity	Format
1-5	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	I5
6-10	Flag to delete or activate nodal constraint sets 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	I5
11-15	Flag to delete or activate rigid walls EQ.0: no change EQ.1: delete EQ.2: activate	I5
16-25	Maximum permitted time step size after restart	E10.0

Card 1 (2I5)

Columns	Quantity	Format
1-5	NDEFR, number of materials to switch from deformable to rigid plus the number of rigid bodies for which new master bodies are defined.	I5
6-10	Number of materials to switch from rigid to deformable, NRDEF	I5

(Restart) Rigid/Deformable Material Switching

Cards 2,...NDEFR+1 (2I5) or (2I10) if MLARG

Columns		Quantity	Format
1-5	(1-10)	The identification number of a material which is switched to a rigid material or a rigid material which will become a slave to a master rigid body.	I5 (I10)
6-10	(11-20)	The identification number of the master rigid body to which the material is merged. If zero, the material becomes either an independent or master rigid body.	I5 (I10)

Cards 2+NDEFR,...NDEFR+NRDEF+1 (I5) or (I10) if MLARG

Columns		Quantity	Format
1-5	(1-10)	The identification number of a material which is switched from a rigid to a deformable material. This must be a material which was switched to a rigid material in the initial input or in a previous restart . When a master rigid body is switched to a deformable material, its slaves are become independent rigid bodies.	I5 (I10)

REFERENCES

Arruda, E. and M. Boyce, "A Three-Dimensional Constitutive Model for the Large Stretch Behavior of Rubber Elastic Materials," published in the Journal of the Mechanics and Physics of Solids, Vol. 41, No. 2, pp. 389-412, (1993).

Baker, E.L., "An Explosives Products Thermodynamic Equation of State Approapriate for Material Acceleration and Overdriven Detonation: Theoretical Background and Fourmulation," Technical Report ARAED-TR-911013, U.S. Army Armament Research, Development and Engineering Center, Picatinney Arsenal, New Jersey, 1991.

Baker, E.L. and J. Orosz, J., "Advanced Warheads Concepts: An Advanced Equation of State for Overdriven Detonation," Technical Report ARAED-TR-911007, U.S. Army Armament Research, Development and Engineering Center, Picatinney Arsenal, New Jersey, 1991.

Baker, E.L. and L.I. Stiel, "Improved Quantitative Explosive Performance Prediction Using Jaguar," 1997 Insensitive Munitions and Energetic Materials Technology Symposium, Tampa, FL, (1997).

Bammann, D. J. and E.C. Aifantis, “A Model for Finite-Deformation Plasticity,” Acta Mechanica, 70, 1-13 (1987).

Bammann, D. J. and Johnson, G., “On the Kinematics of Finite-Deformation Plasticity,” Acta Mechanica, *69*, 97-117 (1987).

Bammann, D. J., “Modeling the Temperature and Strain Rate Dependent Large Deformation of Metals,” Proceedings of the 11th US National Congress of Applied Mechanics, Tuscon, Az., (1989).

Bammann, D.J., M.L. Chiesa, A. McDonald, W.A. Kawahara, J.J. Dike, and V.D. Revelli, “Predictions of Ductile Failure in Metal Structures,” in AMD-Vol. 107, Failure Criteria and Analysis in Dynamic Response, Edited by. H.E. Lindberg, 7-12, (1990).

Bandak, F.A., private communications, U.S. Dept. of Trans., Division of Biomechanics Research, 400 7th St., S.W. Washington, D.C. 20590 (1991).

Barlat, F., and J. Lian, "Plastic Behavior and Stretchability of Sheet Metals. Part I: A Yield Function for Orthotropic Sheets Under Plane Stress Conditions," Int. J. of Plasticity, Vol. 5, 51-66 (1989).

Barlat, F., D.J. Lege, and J.C. Brem, “A Six-Component Yield Function for Anisotropic Materials,” Int. J. of Plasticity, 7, 693-712, (1991).

Bazeley, G.P., W.K. Cheung, R.M. Irons, and O.C. Zienkiewicz, “Triangular Elements in Plate Bending-Confirming and Nonconforming Solutions in Matrix Methods and

Structural Mechanics," Proc. Conf. on Matrix Methods in Structural Analysis, Rept. AFFDL-R-66-80, Wright Patterson AFB, 547-576 (1965).

Belytschko, T.B. and A.H. Marchertas, "Nonlinear Finite Element Method for Plates and its Application to the Dynamic Response of Reactor Fuel Subassemblies," Trans, ASME J. Pressure Vessel Tech., 251-257 (1974).

Belytschko, T.B. and C.S. Tsay, "Explicit Algorithms for Nonlinear Dynamics of Shells," AMD-Vol.48, ASME, 209-231 (1981).

Belytschko, T.B. and C.S. Tsay, "Explicit Algorithms for Nonlinear Dynamics of Shells," Comp. Meth. Appl. Mech. Eng., 43, 251-276, (1984).

Belytschko, T.B., and C. S. Tsay, "A Stabilization Procedure for the Quadrilateral Plate Element with One-Point Quadrature," Int. J. Num. Method. Eng., 19, 405-419 (1983).

Belytschko, T.B., H. Stolarski, and N. Carpenter, "A C… Triangular Plate Element with One-Point Quadrature," Int. J. Num. Meth. Eng., 20, 787-802 (1984).

Belytschko, T.B., L. Schwer, and M.J. Klein, "Large Displacement Transient Analysis of Space Frames," Int. J. Num. Eng., 11, 65-84 (1977).

Benson, D. J. and J.O. Hallquist, "A Simple Rigid Body Algorithm for Structural Dynamics Program," Int. J. Numer. Meth. Eng., 22, (1986).

Benson, D.J. and J.O. Hallquist, "A Single Surface Contact Algorithm for the Postbuckling Analysis of Shell Structures," Comp. Meths. Appl. Mech. Eng., 78, 141-163 (1990).

Berstad, T., Langseth, M. and Hopperstad, O.S. "Elasto-viscoplastic Constitutive Models in the Explicit Finite Element Code LS-DYNA3D," Second International LS-DYNA3D conference, San Francisco, (1994).

Berstad, T., "Material Modelling of Aluminium for Crashworthiness Analysis", Dr.Ing. Dissertation, Department of Structural Engineering, Norwegian University of Science and Technology, Trondheim, Norway, (1996).

Bilkhu, S.S., M. Founas, and G.S. Nasholtz, "Material Modeling of Structural Foams in Finite Element Analysis Using Compressive Uniaxial and Triaxial Data," SAE (Nat. Conf.) Detroit 1993, pp. 4-34.

Brode, H.L. "Height of Burst Effects at High Overpressure," (DASA 2506).

Brown, B.E. and J.O. Hallquist, "TAURUS: An Interactive Post-Processor for the Analysis Codes NIKE3D, DYNA3D, TACO3D, and GEMINI," University of California, Lawrence Livermore National Laboratory, Rept. UCID-19392 (1982) Rev. 1 (1984).

Burton, D.E. et al. “Physics and Numerics of the TENSOR Code,” Lawrence Livermore National Laboratory, Internal Document UCID-19428, (July 1982).

Chang, F.K. and K.Y. Chang, “A Progressive Damage Model for Laminated Composites Containing Stress Concentration,” J. of Composite Materials, 21, 834-855 (1987a).

Chang, F.K. and K.Y. Chang, “Post-Failure Analysis of Bolted Composite Joints in Tension or Shear-Out Mode Failure,” J. of Composite Materials, 21, 809-833 (1987b).

Chang, F.S., “Constitutive Equation Development of Foam Materials,” Ph.D. Dissertation, submitted to the Graduate School, Wayne State University, Detroit, Michigan (1995).

Chang, F.S., J.O. Hallquist, D. X. Lu, B. K. Shahidi, C. M. Kudelko, and J.P. Tekelly, “Finite Element Analysis of Low Density High-Hystersis Foam Materials and the Application in the Automotive Industry,” SAE Technical Paper 950908, in Saftey Technology (SP-1041), International Congress and Exposition, Detroit, Michigan (1994).

Chen, W.F., and Baladi, G.Y.,*Soil Plasticity: Theory and Implementation*, Elesvier, New York, 1985.

Christensen, R.M., "A Nonlinear Theory of Viscoelasticity for Application to Elastomers," Transactions of the ASME, J. of Appl. Mech., Vol 47, 762-768, (1980).

Chung, K. and K. Shah, “Finite Element Simulation of Sheet Metal Forming for Planar Anisotropic Metals,” Int. J. of Plasticity, 8, 453-476, (1992).

Cochran, S.G. and J. Chan, “Shock Initiation and Detonation Models in One and Two Dimensions,” University of California, Lawrence Livermore National Laboratory, Rept. UCID-18024 (1979).

Couch, R., Albright, E. and Alexander, “The JOY Computer Code,” University of California, Lawrence Livermore National Laboratory, Rept. UCID-19688 (1983).

CRAY-1 Computer System CFT Reference Manual, Cray Research Incorporated, Bloomington, NM., Publication No. 2240009 (1978).

DeRuntz, J.A. Jr.,”Reference Material for USA, The Underwater Shock Analysis Code, USA-STAGS, and USA-STAGS-CFA,” Report LMSC-P032568, Computational Mechanics Laboratory, Lockheed Palo Alto Research Laboratory, Palo Alto, CA. (1993).

Desai, C.S., and H.J. Siriwardane, Constitutive Laws for Engineering Materials With Emphasis On Geologic Materials, Prentice-Hall, Chapter 10, (1984).

References

Dilger, W.H., R. Koch, and R. Kowalczyk, "Ductility of Plain and Confined Concrete Under Different Strain Rates," ACI Journal, January-February, (1984).

Dobratz, B.M., “LLNL Explosives Handbook, Properties of Chemical Explosives and Explosive Simulants,” University of California, Lawrence Livermore National Laboratory, Rept. UCRL-52997 (1981).

Englemann, B. E., R.G. Whirley, and G.L. Goudreau, “A Simple Shell Element Formulation for Large-Scale Elastoplastic Analysis,” CED-Vol. 3. Analytical and Computational Models of Shells, A.K. Noor, T. Belytschko, and J.C. Simo, Editors, 1989, pp. 399-416.

Flanagan, D.P. and T. Belytschko, “A Uniform Strain Hexahedron and Quadrilateral and Orthogonal Hourglass Control,” Int. J. Numer. Meths. Eng., 17, 679-706 (1981)

Ginsberg, M. and J. Johnson, “Benchmarking the Performance of Physical Impact Simulation Software on Vector and Parallel Computers,” Applications Track of Supercomputing, IEEE monograph, Computer Society Press, March, 1989.

Giroux, E.D. “HEMP User’s Manual,” University of California, Lawrence Livermore National Laboratory, Rept. UCRL-51079 (1973).

Goudreau, G.L. and J.O. Hallquist, “Recent Developments in Large Scale Finite Element Lagrangian Hydrocode Technology,” J. Comp. Meths. Appl. Mechs. Eng., 30 (1982).

Govindjee, S., Kay, J.G., and Simo, J.C. [1994], "Anisotropic Modeling and Numerical Simulation of Brittle Damage in Concrete," Report No. UCB/SEMM-94/18, Department of Civil Engineering, University of California, Berkeley, CA 94720.

Govindjee, S., Kay, J.G., and Simo, J.C. [1995], "Anisotropic Modeling and Numerical Simulation of Brittle Damage in Concrete," *Int. J. Numer. Meth. Engng,* **38**, 3611-3633.

Graefe, H., W. Krummheuer, and V. Siejak, “Computer Simulation of Static Deployment Tests for Airbags, Air Permeability of Uncoated Fabrics and Steady State Measurements of the Rate of Volume Flow Through Airbags,” SAE Technical Paper Series, 901750, Passenger Car Meeting and Expositition, Dearborn, Michigan, September 17-20, 1990.

Guccione, J., A. McCulloch, and L. Waldman, "Passive Material Properties of Intact Ventricular Myocardium Determined from a Cylindrical Model," published in the ASME Journal of Biomechanical Engineering, Vol. 113, pages 42-55, (1991).

Hallquist, J.O. “A Numerical Procedure for Three-Dimensional Impact Problems,” American Society of Civil Engineering, Preprint 2956 (1977).

Hallquist, J.O. “A Numerical Treatment of Sliding Interfaces and Impact,” in: K.C. Park and D.K. Gartling (eds.), Computational Techniques for Interface Problems, AMD Vol. 30, ASME, New York (1978).

Hallquist, J.O. “A Procedure for the Solution of Finite Deformation Contact-Impact Problems by the Finite Element Method,” University of California, Lawrence Livermore National Laboratory, Rept. UCRL-52066 (1976).

Hallquist, J.O. “NIKE2D: An Implicit, Finite-Element Code for Analyzing the Static and Dynamic Response of Two-Dimensional Solids,” University of California, Lawrence Livermore National Laboratory, Rept. UCRL-52678 (1979).

Hallquist, J.O. “Preliminary User’s Manuals for DYNA3D and DYNAP (Nonlinear Dynamic Analysis of Solids in Three Dimension),” University of California, Lawrence Livermore National Laboratory, Rept. UCID-17268, (1976) and Rev. 1 (1979).

Hallquist, J.O. “Theoretical Manual for DYNA3D,” University of California, Lawrence Livermore National Laboratory, Rept. UCID-19501 (1982).

Hallquist, J.O. “User’s Manual for DYNA2D—An Explicit Two-Dimensional Hydrodynamic Finite Element Code with Interactive Rezoning,” University of California, Lawrence Livermore National Laboratory, Rept. UCID-18756 (1980).

Hallquist, J.O. “User’s Manual for DYNA3D and DYNAP (Nonlinear Dynamic Analysis of Solids in Three Dimensions),” University of California, Lawrence Livermore National Laboratory, Rept. UCID-19156 (1981).

Hallquist, J.O. and D.J. Benson, “A Comparison of an Implicit and Explicit Implementation of the Hughes-Liu Shell,” Finite Element Methods for Plate and Shell Structures, T.J.R. Hughes and E. Hinton, Editors, 394-431, Pineridge Press Int., Swansea, U.K. (1986).

Hallquist, J.O., D.J. Benson, and G.L. Goudreau, “Implementation of a Modified Hughes-Liu Shell into a Fully Vectorized Explicit Finite Element Code,” Proceedings of the International Symposium on Finite Element Methods for Nonlinear Problems, University of Trondheim, Trondheim, Norway (1985).

Hallquist, J.O., G.L. Goudreau, and D.J. Benson, “Sliding Interfaces with Contact-Impact in Large Scale Lagrangian Computations,” Comp. Meths. Appl. Mech. Eng., 51, 107-137 (1985).

Herrmann, L.R. and F.E. Peterson, “A Numerical Procedure for Viscoelastic Stress Analysis,” Seventh Meeting of ICRPG Mechanical Behavior Working Group, Orlando, FL, CPIA Publication No. 177, 1968.

Hill, R., “A Theory of the Yielding and Plastic Flow of Anisotropic Metals,” Proceedings of the Royal Society of London, Series A., Vol. 193 1948 pp. 281-197.

Hill, R., “Constitutive Modelling of Orthotropic Plasticity in Sheet Metals,” J. Mech. Phys. Solids, Vol. 38, No. 3, 1989 pp. 405-417.

Hirth, A., P. Du Bois, and K. Weimar, "Improvement of LS-DYNA Material Law 83 (Fu Chang) for the Industrial Simulation of Reversible Energy-Absorbing Foams," CAD-FEM User's Meeting, Bad Neuenahr - Ahrweiler, Germany, October 7-9, Paper 2-40, (1998).

Hopperstad, O.S. and Remseth, S.," A return Mapping Algorithm for a Class of Cyclic Plasticity Models", International Journal for Numerical Methods in Engineering, Vol. 38, pp. 549-564, (1995).

Hughes, T.J.R. and W.K. Liu, “Nonlinear Finite Element Analysis of Shells: Part I. Three-Dimensional Shells.” Comp. Meths. Appl. Mechs., 27, 331-362 (1981a).

Hughes, T.J.R. and W.K. Liu, “Nonlinear Finite Element Analysis of Shells: Part II. Two-Dimensional Shells.” Comp. Meths. Appl. Mechs., 27, 167-181 (1981b)

Hughes, T.J.R. W.K. Liu and I. Levit, “Nonlinear Dynamics Finite Element Analysis of Shells.” Nonlinear Finite Element Analysis in Struct. Mech., Eds. W. Wunderlich, E. Stein, and K.J. Bathe, Springer-Verlag, Berlin, 151- 168 (1981c).

Isenberg, J., D.K. Vaughn, and I. Sandler, "Nonlinear Soil-Structure Interaction," EPRI Report MP-945, Weidlinger Associates, December, (1978).

Johnson, G. C. and Bammann, D. J., “A discussion of stress rates in finite deformation problems, “ Int. J. Solids Struct, 20, 725-737 (1984).

Johnson, G.R. and W. H. Cook, “A Constitutive Model and Data for Metals Subjected to Large Strains, High Strain Rates and High Temperatures.” Presented at the Seventh International Symposium on Ballistics, the Hague, The Netherlands, April 1983.

Kenchington, G. J., “A Non-Linear Elastic Material Model for DYNA3D,” Proceedings of the DYNA3D Users Group Conference, September 1988, published by Boeing Computer Services (Europe) Limited.

Key, S.W. “HONDO - A Finite Element Computer Program for the Large Deformation Dynamic Response of Axisymmetric Solids,” Sandia National Laboratories, Albuquerque, N.M., Rept. 74-0039 (1974).

Krieg, R.D. and S.W. Key, Implementation of a Time Dependent Plasticity Theory into Structural Computer Programs, Vol. 20 of Constitutive Equations in Viscoplasticity: Computational and Engineering Aspects (American Society of Mechanical Engineers, New York, N.Y., 1976), pp. 125-137.

Lee, E.L. and C.M. Tarver, "Phenomenological Model of Shock Initiation in Heterogenous Explosives," PHYS. Fluids, Vol. 23, p. 2362 (1980).

Maker, B. N. Private communication Lawrence Livermore National Laboratory, Dr. Maker programmed and implemented the compressible Mooney Rivlin rubber model (1987).

Mauldin, P.J., R.F. Davidson, and R.J. Henninger, "Implementation and Assessment of the Mechanical=Threshold=Stress Model Using the EPIC2 and PINON Computer Codes," Report LA-11895-MS, Los Alamos National Laboratory (1990).

Neilsen, M.K., Morgan, H.S., and Krieg, R.D., "A Phenomenological Constitutive Model for Low Density Polyurethane Foams," Rept. SAND86-2927, Sandia National Laboratories, Albuquerque, N.M., (1987)

Nusholtz, G.S., Fong, W., Wu, J., Suo, L., and Wylie, B.E., "Airbag Inflator Gas Jet Evaluation", Preprint, (1991).

Nusholtz, G.S., Fong, W. and Wu, J., "Air Bag Wind-Blast Phenomena Evaluation", Experimental Techniques, 38-39, (November/December 1991).

Ogden, R.W., Non-Linear Elastic Deformations, Ellis Horwood Ltd., Market Cross House, Cooper St., Chichester, W. Sussex, England, (1984).

Oliver, J. [1989], "A Consistent Characteristic Length of Smeared Cracking Models," *Int. J. Numer. Meth. Engng*, **28**, 461-474.

Papadrakakis, M.,"A Method for the Automatic Evaluation of the Dynamic Relaxation Parameters," Comp. Meth. Appl. Mech. Eng., Vol. 25, pp. 35-48, (1981).

Pelessone, D., Private communication, GA Technologies, P. O. Box 85608, San Diego, CA., Telephone No. 619-455-2501 (1986).

Sackett, S.J., "Geological/Concrete Model Development," Private Communication (1987).

Sandler, I.S. and D. Rubin, "An Algorithm and a modular subroutine for the cap model," Int. J. Numer. Analy. Meth. Geomech., 3 pp. 173-186 (1979).

Shkolnikov, M.B. Private Communication, (1990-1991).

Schwer, L.E., W. Cheva, and J.O. Hallquist, "A Simple Viscoelastic Model for Energy Absorbers Used in Vehicle-Barrier Impacts," in preparation.

Simo, J. C., J.W. Ju, K.S. Pister and R.L. Taylor, "An Assessment of the Cap Model: Consistent Return Algorithms and Rate-Dependent Extension," J. Eng. Mech., Vol. 114, No. 2, 191-218 (1988a).

Simo, J. C., J.W. Ju, K.S. Pister and R.L. Taylor, "Softening Response, Completeness Condition, and Numerical Algorithms for the Cap Model," Int. J. Numer. Analy. Meth. Eng., (1990).

Steinberg, D.J. and M.W. Guinan, "A High-Strain-Rate Constitutive Model for Metals," University of California, Lawrence Livermore National Laboratory, Rept. UCRL-80465 (1978).

Steinberg, D.J. and C.M. Lund, "A Constitutive Model for Strain Rates form 10^{-4} to 10^{6} S^{-1}," J. Appl. Phys., 65, p. 1528 (1989).

Stillman, D. W. and J. O. Hallquist, "INGRID: A Three-Dimensional Mesh Generator for Modeling Nonlinear Systems," University of California, Lawrence Livermore National Laboratory, Rept. UCID-20506. (1985).

Stojko, S., privated communication, NNC Limited, Engineering Development Center, [1990].

Storakers, B. "On Material Representation and Constitutive Branching in Finite Compressible Elasticity", Royal Institute of Technology, Stockholm Sweden, (1985).

Stout, M. G., D. E. Helling, T. L. Martin and G. R. Canova, Int. J. Plasticity, Vol. 1, pp. 163-174, 1985.

Sussman, T. and K.J. Bathe, "A Finite Element Formulation for Nonlinear Incompressible Elastic and Inelastic Analysis", Computers and Structures, Vol. 26, No. 1/2, pp. 357-409 (1987).

Taylor, L.M. and Flanagan, D.P., "PRONTO3D A Three-Dimensional Transient Solid Dynamics Program," Sandia Report: SAND87-1912, UC-32, (1989).

TNO, MADYMO USER'S MANUAL 3D, Version 4.3, TNO Road-Vehicles Research Institute, Department of Injury Prevention, The Hague, The Netherlands, (1990)

Tuler, F.R. and B.M. Butcher, "A Criterion for the Time Dependence of Dynamic Fracture," *The International Journal of Fracture Mechanics*, Vol. 4, No. 4, (1968).

Vawter, D., "A Finite Element Model for Macroscopic Deformation of the Lung," published in the Journal of Biomechanical Engineering, Vol.102, pp. 1-7, (1980).

Wang, J.T., "An Analytical Model for an Airbag with a Hybrid Inflator", Publication R&D 8332, General Motors Development Center, Warren, Mi. (1995).

Wang, J.T., "An Analytical Model for an Airbag with a Hybrid Inflator", AMD-Vol. 210, BED-Vol. 30, ASME, pp 467-497, (1995).

Wang, J. T. and O. J. Nefske, "A New CAL3D Airbag Inflation Model," SAE paper 880654, 1988.

Whirley, R. G., and J. O. Hallquist, "DYNA3D, A Nonlinear, Explicit, Three-Dimensional Finite Element Code for Solid and Structural Mechanics-Users

Manual,” Report No.UCRL-MA-107254 , Lawrence Livermore National Laboratory, (1991).

Whirley, R. G., J. O. Hallquist and G. L. Goudreau, “An Assessment of Numerical Algorithms for Plane Stress and Shell Elastoplasticity on Supercomputers,” Engineering Computations, Vol. 6, pp. 116-126, (1989).

Wilkins, M.L., R.E. Blum, E. Cronshagen, and P. Grantham, “A Method for Computer Simulation of Problems in Solid Mechanics and Gas Dynamics in Three Dimensions and Time,” University of California, Lawrence Livermore National Laboratory, Rept. UCRL-51574 (1974).

Woodruff, J.P. “KOVEC User’s Manual,” University of California, Lawrence Livermore National Laboratory, Rept. UCRL-51079 (1973).

APPENDIX A: User Defined Materials

The addition of user material subroutine into LS-DYNA is relatively simple. A control card, starting with card 14 in the control section, is required for each user subroutine. The number of history variables is arbitrary and can be any number greater than or equal to 0. When the material requires the deformation gradient, the number of history variables must be increased by 9 for its storage. The coordinate system definition is optional but is probably necessary if the model involves material that have directional properties such as composites and anisotropic plasticity models. When the coordinate system option is used then all data passed to the constitutive model is in the local system. A bulk modulus and shear modulus are required for transmitting boundaries, contact interfaces, rigid body constraints, and time step size calculations. The number of constants read in columns 6-10 include the eight values for the coordinate system option if it is nonzero and two values for the bulk and shear modulus.Up to ten user subroutines can currently be implemented simultaneously to update the stresses is solids, shells, thick shells, and beam elements. A sample subroutine is given in this Appendix for treating an elastic material.

The deformation gradient matrix is stored in 9 of the history variables requested on the control cards. To compute the deformation gradient matrix for <u>solid elements only</u> add the call:

CALL COMPUTE_FS(F11,F21,F31,F12,F22,F32,F13,F23,F33)

if the user subroutine is scalar or

CALL COMPUTE_F (F11,F21,F31,F12,F22,F32,F13,F23,F33,LFT,LLT)

for a vectorized implementation. These calls must be placed at the beginning of the user subroutine, where F11 through F33 are the history variable arrays containing the individual components of the deformation gradient matrix, and LFT and LLT indicate the range over the arrays. For the non-vectorized subroutine F11 through F33 are scalars.

When implementing plane stress constitutive models for shells and beams, the strain increments in the directions of the zero normal stress must be determined. In shell elements this is the strain increment EPS(3) which is normal to the midsurface and in beam elements this includes the strain increments EPS(2) and EPS(3) which are normal to the axis. These strain increments are used in the shell elements to account for thickness changes.

A sample subroutine is provided below for treating an elastic material.

```
      SUBROUTINE UMAT41 (CM,EPS,SIG,HISV,DT1,CAPA,ETYPE,TIME)
C*******************************************************************
C|     LIVERMORE SOFTWARE TECHNOLOGY CORPORATION (LSTC)            |
C|     ------------------------------------------------------------|
C|     COPYRIGHT 1987, 1988, 1989 JOHN O. HALLQUIST, LSTC          |
C|     ALL RIGHTS RESERVED                                         |
C*******************************************************************
C
C     ISOTROPIC ELASTIC MATERIAL (SAMPLE USER SUBROUTINE)
C
C     VARIABLES
C
C     CM(1)=YOUNG'S MODULUS
C     CM(2)=POISSON'S RATIO
C
C     EPS(1)=LOCAL X STRAIN
C     EPS(2)=LOCAL Y STRAIN
C     EPS(3)=LOCAL Z STRAIN
C     EPS(4)=LOCAL XY STRAIN
C     EPS(5)=LOCAL YZ STRAIN
C     EPS(6)=LOCAL ZX STRAIN
C     EPS(1)=LOCAL X STRAIN
C
C     SIG(1)=LOCAL X STRESS
C     SIG(2)=LOCAL Y STRESS
C     SIG(3)=LOCAL Z STRESS
C     SIG(4)=LOCAL XY STRESS
C     SIG(5)=LOCAL YZ STRESS
C     SIG(6)=LOCAL ZX STRESS
C
C     HISV(1)=1ST HISTORY VARIABLE
C     HISV(2)=2ND HISTORY VARIABLE
C        .
C        .
C        .
C        .
C     HISV(N)=NTH HISTORY VARIABLE
C
C     DT1=CURRENT TIME STEP SIZE
C     CAPA=REDUCTION FACTOR FOR TRANSVERSE SHEAR
C     ETYPE:
C        EQ."BRICK" FOR SOLID ELEMENTS
C        EQ."SHELL" FOR ALL SHELL ELEMENTS
C        EQ."BEAM"  FOR ALL BEAM ELEMENTS
C
C     TIME=CURRENT PROBLEM TIME.
C
C
C     ALL TRANSFORMATIONS INTO THE ELEMENT LOCAL SYSTEM ARE PERFORMED
C     PRIOR TO ENTERING THIS SUBROUTINE.  TRANSFORMATIONS BACK TO
C     THE GLOBAL SYSTEM ARE PERFORMED AFTER EXISTING THIS SUBROUTINE.
C
C     ALL HISTORY VARIABLES ARE INITIALIZED TO ZERO IN THE INPUT PHASE.
C     INITIALIZATION OF HISTORY VARIABLES TO NONZERO VALUES MAY BE DONE
C     DURING THE FIRST CALL TO THIS SUBROUTINE FOR EACH ELEMENT.
C
C     ENERGY CALCULATIONS FOR THE DYNA3D ENERGY BALANCE ARE DONE
C     OUTSIDE THIS SUBROUTINE.
C
```

```
      CHARACTER*(*) ETYPE
      DIMENSION CM(*),EPS(*),SIG(*),HISV(*)
C
C     COMPUTE SHEAR MODULUS, G
C
      G2=CM(1)/(1.+CM(2))
      G =.5*G
C
      IF (ETYPE.EQ.'BRICK') THEM
      DAVG=(-EPS(1)-EPS(2)-EPS(3))/3.
      P=-DAVG*CM(1)/((1.-2.*CM(2)))
      SIG(1)=SIG(1)+P+G2*(EPS(1)+DAVG)
      SIG(2)=SIG(2)+P+G2*(EPS(2)+DAVG)
      SIG(3)=SIG(3)+P+G2*(EPS(3)+DAVG)
      SIG(4)=SIG(4)+G*EPS(4)
      SIG(5)=SIG(5)+G*EPS(5)
      SIG(6)=SIG(6)+G*EPS(6)
C
      ELSEIF (ETYPE.EQ.'SHELL') THEN
C
      GC =CAPA*G
      Q1    =CM(1)*CM(2)/((1.0+CM(2))*(1.0-2.0*CM(2)))
      Q3    =1./(Q1+G2)
      EPS(3)=-Q1*(EPS(1)+EPS(2))*Q3
      DAVG  =(-EPS(1)-EPS(2)-EPS(3))/3.
      P     =-DAVG*CM(1)/((1.-2.*CM(2)))
      SIG(1)=SIG(1)+P+G2*(EPS(1)+DAVG)
      SIG(2)=SIG(2)+P+G2*(EPS(2)+DAVG)
      SIG(3)=0.0
      SIG(4)=SIG(4)+G *EPS(4)
      SIG(5)=SIG(5)+GC*EPS(5)
      SIG(6)=SIG(6)+GC*EPS(6)
C
      ELSEIF (ETYPE.EQ.'BEAM') THEN
      Q1    =CM(1)*CM(2)/((1.0+CM(2))*(1.0-2.0*CM(2)))
      Q3    =Q1+2.0*G
      GC    =CAPA*G
      DETI  =1./(Q3*Q3-Q1*Q1)
      C22I  = Q3*DETI
      C23I  =-Q1*DETI
      FAC   =(C22I+C23I)*Q1
      EPS(2)=-EPS(1)*FAC-SIG(2)*C22I-SIG(3)*C23I
      EPS(3)=-EPS(1)*FAC-SIG(2)*C23I-SIG(3)*C22I
      DAVG  =(-EPS(1)-EPS(2)-EPS(3))/3.
      P     =-DAVG*CM(1)/(1.-2.*CM(2))
      SIG(1)=SIG(1)+P+G2*(EPS(1)+DAVG)
      SIG(2)=0.0
      SIG(3)=0.0
      SIG(4)=SIG(4)+GC*EPS(4)
      SIG(5)=0.0
      SIG(6)=SIG(6)+GC*EPS(6)
      ENDIF
C
      RETURN
      END
```

APPENDIX B: User Defined Airbag Sensor

The addition of a user sensor subroutine into DYNA3D is relatively simple. The sensor is mounted on a rigid body which is attached to the structure. The motion of the sensor is provided in the local coordinate system defined for the rigid body in the definition of material model 20-the rigid material. When the user defined criterion is met for the deployment of the airbag, a flag is set and the deployment begins. All load curves relating to the mass flow rate versus time are then shifted by the initiation time. The user subroutine is given below with all the necessary information contained in the comment cards.

```
      SUBROUTINE AIRUSR (RBU,RBV,RBA,TIME,DT1,DT2,PARAM,HIST,ITRNON,
      . RBUG,RBVG,RBAG)
C*******************************************************************
C|     LIVERMORE SOFTWARE TECHNOLOGY CORPORATION (LSTC)            |
C|     ------------------------------------------------------------|
C|     COPYRIGHT 1987, 1988, 1989 JOHN O. HALLQUIST, LSTC          |
C|     ALL RIGHTS RESERVED                                         |
C*******************************************************************
C
C     USER SUBROUTINE TO INITIATE THE INFLATION OF THE AIRBAG
C
C     VARIABLES
C
C     DISPLACEMENTS ARE DEFINED AT TIME N+1 IN LOCAL SYSTEM
C     VELOCITIES ARE DEFINED AT TIME N+1/2 IN LOCAL SYSTEM
C     ACCELERATIONS ARE DEFINED AT TIME N IN LOCAL SYSTEM
C
C           RBU(1-3) TOTAL DISPLACEMENTS IN THE LOCAL XYZ DIRECTIONS
C           RBU(3-6) TOTAL ROTATIONS ABOUT THE LOCAL XYZ AXES
C           RBV(1-3) VELOCITIES IN THE LOCAL XYZ DIRECTIONS
C           RBV(3-6) ROTATIONAL VELOCITIES ABOUT THE LOCAL XYZ AXES
C           RBA(1-3) ACCELERATIONS IN THE LOCAL XYZ DIRECTIONS
C           RBA(3-6) ROTATIONAL ACCELERATIONS ABOUT THE LOCAL XYZ AXES
C           TIME IS THE CURRENT TIME
C           DT1 IS TIME STEP SIZE AT N-1/2
C           DT2 IS TIME STEP SIZE AT N+1/2
C           PARAM IS USER DEFINED INPUT PARAMETERS
C           HIST IS USER DEFINED HISTORY VARIABLES
C           ITRNON IS FLAG TO TURN ON THE AIRBAG INFLATION
C           RBUG,RBVG,RBAG, ARE SIMILAR TO RBU,RBV,RBA BUT ARE DEFINED
C           GLOBALLY.
C
C     THE USER SUBROUTINE SETS THE VARIABLE ITRNON TO:
C
C              ITRNON=0 BAG IS NOT INFLATED
C              ITRNON=1 BAG INFLATION BEGINS AND THIS SUBROUTINE IN NOT
C                       CALLED AGAIN
C
      DIMENSION RBU(6),RBV(6),PARAM(25),HIST(25),
      . RBUG(6),RBVG(6),RBAG(6)
      RETURN
      END
```

APPENDIX C: User Defined Solution Control

This subroutine may be provided by the user to control the I/O, monitor the energies and other solution norms of interest, and to shut down the problem whenever he pleases. The arguments are defined in the listing provided below.

```
      SUBROUTINE UCTRL1 (NUMNP,NDOF,TIME,DT1,DT2,PRTC,PLTC,FRCI,PRTO,
     . PLTO,FRCO,VT,VR,AT,AR,UT,UR,XMST,XMSR,IRBODY,RBDYN,USRHV,
     . MESSAG,TOTALM,CYCL,IDRINT)
C*******************************************************************
C|     LIVERMORE SOFTWARE TECHNOLOGY CORPORATION (LSTC)            |
C|     ------------------------------------------------------------ |
C|     COPYRIGHT 1987, 1988, 1989 JOHN O. HALLQUIST, LSTC          |
C|     ALL RIGHTS RESERVED                                         |
C*******************************************************************
C
      CHARACTER*(*) MESSAG
      INTEGER CYCLE
C
C
C     USER SUBROUTINE FOR SOLUTION CONTROL
C
C     NOTE:  LS-DYNA USED AN INTERNAL NUMBERING SYSTEM TO
C            ACCOMODATE ARBITRARY NODE NUMBERING.  TO ACCESS
C            INFORMATION FOR USER NODE N, ADDRESS ARRAY LOCATION M,
C            M=LQF(N,1).  TO OBTAIN USER NODE NUMBER, N,
C            CORRESPONDING TO ARRAY ADDRESS M, SET N=LQFINV(M,1)
C
C     ARGUMENTS:
C        NUMNP=NUMBER OF NODAL POINTS
C        NDOF=NUMBER OF DEGREES IF FREEDOM PER NODE
C        TIME=CURRENT SOLUTION TIME
C        PRTC=OUTPUT INTERVAL FOR TAURUS TIME HISTORY DATA
C        PLTC=OUTPUT INTERVAL FOR TAURUS STATE DATA
C        FRCI=OUTPUT INTERVAL FOR TAURUS INTERFACE FORCE DATA
C        PRTO=OUTPUT TIME FOR TIME HISTORY FILE
C        PLTO=OUTPUT TIME FOR STATE DATA
C        FRCO=OUTPUT TIME FOR FORCE DATA
C        VT(3,NUMNP)  =NODAL TRANSLATIONAL VELOCITY VECTOR
C        VR(3,NUMNP)  =NODAL ROTATIONAL VELOCITY VECTOR.  THIS ARRAY
C                      IS DEFINED IF AND ONLY IF NDOF=6
C        AT(3,NUMNP)  =NODAL TRANSLATIONAL ACCELERATION VECTOR
C        AR(3,NUMNP)  =NODAL ROTATIONAL ACCELERATION VECTOR.  THIS
C                      ARRAY IS DEFINED IF AND ONLY IF NDOF=6
C        UT(3,NUMNP)  =NODAL TRANSLATIONAL DISPLACEMENT VECTOR
C        UR(3,NUMNP)  =NODAL ROTATIONAL DISPLACEMENT VECTOR.  THIS ARRAY
C                      IS DEFINED IF AND ONLY IF NDOF=6
C        XMST(NUMNP)  =RECIPROCAL OF NODAL TRANSLATIONAL MASSES
C        XMSR(NUMNP)  =RECIPROCAL OF NODAL ROTATIONAL MASSES.  THIS
C                      ARRAY IS DEFINED IF AND ONLY IF NDOF=6
C        IRBODY       =FLAG FOR RIGID BODY NODAL POINTS
C                      IF DEFORMABLE NODE THEN SET TO 1.0
C                      IF RIGID BODY NODE THEN SET TO 0.0
C                      DEFINED IF AN ONLY IF RIGID BODY ARE PRESENT
C                      I.E.,IRBODY.NE.0 IF NO RIGID BODY ARE PRESENT
C        USRHV(LENHV) =USER DEFINED HISTORY VARIABLES THAT ARE STORED
C                       IN THE RESTART FILE.  LENHV=100+U*NUMMAT WHERE
```

```
C                       NUMMAT IS THE # OF MATERIALS IN THE PROBLEM.
C                       ARRAY USRHV IS UPDATED ONLY IN THIS SUBROUTINE.
C        MESSAG        =FLAG FOR DYNA3D WHICH MAY BE SET TO:
C                       ‘SW1.’ LS-DYNA TERMINATES WITH RESTART FILE
C                       ‘SW3.’ LS-DYNA WRITES A RESTART FILE
C                       ‘SW4.’ LS-DYNA WRITES A PLOT STATE
C        TOTALM        =TOTAL MASS IN PROBLEM
C           CYCLE =CYCLE NUMBER
C           IDRINT      =FLAG FOR DYNAMIC RELAXATION PHASE
C                       .NE.0:  DYNAMIC RELAXATION IN PROGRESS
C                       .EQ.0:  SOLUTION PHASE
C
      COMMON/PTIMES/  PRTIMS(32),PRTLST(32),IGMPRT
C
C     PRTIMS(32)=OUTPUT INTERVALS FOR ASCII FILES
C
C     ASCII FILES
C           ( 1)=CROSS SECTION FORCES
C           ( 2)=RIGID WALL FORCES
C           ( 3)=NODAL DATA
C           ( 4)=ELEMENT DATA
C           ( 5)=GLOBAL DATA
C           ( 6)=DISCRETE ELEMENTS
C           ( 7)=MATERIAL ENERGIES
C           ( 8)=NODAL INTERFACE FORCES
C           ( 9)=RESULTANT INTERFACE FORCES
C           (10)=SMUG ANIMATOR
C           (11)=SPC REACTION FORCES
C           (12)=NODAL CONSTRAIN RESULTANT FORCES
C           (13)=AIRBAG STATISTICS
C           (14)=AVS DATABASE
C           (15)=NODAL FORCE GROUPS
C           (16)=OUTPUT INTERVALS FOR NODAL BOUNDARY CONDITIONS
C           (17)-(32)=UNUSED AT THIS TIME
C
C     PRTLST(32)=OUTPUT TIMES FOR ASCII FILES ABOVE.  WHEN SOLUTION TIME
C            EXCEEDS THE OUTPUT TIME A PRINT STATE IS DUMPED.
C
      COMMON/RBKENG/ENRBDY,RBDYX,RBDYY,RBDYZ
C
C     TOTAL RIGID BODY ENERGIES AND MOMENTUMS:
C            ENRBDY=RIGID BODY KINETIC ENERGY
C            RBDYX =RIGID BODY X-MOMENTUM
C            RBDYY =RIGID BODY Y-MOMENTUM
C            RBDYZ =RIGID BODY Z-MOMENTUM
C
      COMMON/RBKENG/ENRBDY,RBDYX,RBDYY,RBDYZ
C
C     TOTAL RIGID BODY ENERGIES AND MOMENTUMS:
C            SWXMOM=STONEWALL X-MOMENTUM
C            SWYMOM=STONEWALL Y-MOMENTUM
C            SWZMOM=STONEWALL Z-MOMENTUM
C            ENRBDY=STONEWALL KINETIC ENERGY
C
      COMMON/DEENGS/DEENG
C
C     DEENG=TOTAL DISCRETE ELEMENT ENERGY
C
      COMMON/ENERGY/XPE
C
```

```
C     XPE   =TOTAL INTERNAL ENERGY IN THE FINITE ELEMENTS
C
      DIMENSION VT(3,*),VR(3,*),AT(3,*),AR(3,*),UT(3,*),UR(3,*)
      XMST(*),XMSR(*),RBDYN(*),USRHV(*)
C
C     SAMPLE MOMENTUM AND KINETIC ENERGY CALCULATIONS
C
C     REMOVE ALL COMMENTS IN COLUMN 1 BELOW TO ACTIVATE
CC
CC
CC    INITIALIZE KINETIC ENERGY, XKE, AND X,Y,Z MOMENTUMS.
CC
C     XKE=2.*SWKENG+2.*ENRBDY
C     XM-SWXMOM+RBDYX
C     YM=SWYMOM+RBDYY
C     ZM=SWZMOM+RBDYZ
CC
C     NUMNP2=NUMNP
C     IF (NDOF.EQ.6) THEN
C     NUMNP2=NUMNP+NUMNP
C     ENDIF
C     PRINT *,NDOF
C     IF(IRBODY.EQ.0) THEN
CC
CC
CC    NO RIGID BODIES PRESENT
CC
CC    NOTE IN BLANK COMMENT VR FOLLOWS VT.  THIS FACT IS USED BELOW.
C     DO 10 N=1,NUMNP2
C     XMSN=1./XMST(N)
C     VN1=VT(1,N)
C     VN2=VT(2,N)
C     VN3=VT(3,N)
C     XM=XM+XMSN*VN1
C     YM=YM+XMSN*VN2
C     ZM=ZM+XMSN*VN3
C     XKE=XKE+XMSN*(VN1*VN1+VN2*VN2+VN3*VN3)
C  10 CONTINUE
CC
C     ELSE
CC
CC    RIGID BODIES PRESENT
CC
C     DO 20 N=1,NUMNP
C     XMSN=1./XMST(N)
C     VN1=RBDYN(N)*VT(1,N)
C     VN2=RBDYN(N)*VT(2,N)
C     VN3=RBDYN(N)*VT(3,N)
C     XM=XM+XMSN*VN1
C     YM=YM+XMSN*VN2
C     ZM=ZM+XMSN*VN3
C     XKE=XKE+XMSN*(VN1*VN1+VN2*VN2+VN3*VN3)
C  20 CONTINUE
C     IF (NDOF.EQ.6) THEN
C     DO 30 N=1,NUMNP
C     XMSN=1./XMSR(N)
C     VN1=RBDYN(N)*VR(1,N)
C     VN2=RBDYN(N)*VR(2,N)
C     VN3=RBDYN(N)*VR(3,N)
C     XM=XM+XMSN*VN1
```

```
C     YM=YM+XMSN*VN2
C     ZM=ZM+XMSN*VN3
C     XKE=XKE+XMSN*(VN1*VN1+VN2*VN2+VN3*VN3)
C  30 CONTINUE
C     ENDIF
CC
C     ENDIF
      RETURN
      END
CC
CC.....TOTAL KINETIC ENERGY
C      XKE=.5*XKE
CC.....TOTAL INTERNAL ENERGY
C      XIE=.XPE+DEENG
CC.....TOTAL ENERGY
C      XTE=XKE+XPE+DEENG
CC.....TOTAL X-RIGID BODY VELOCITY
C      XRBV=XM/TOTALM
CC.....TOTAL Y-RIGID BODY VELOCITY
C      YRBV=YM/TOTALM
CC.....TOTAL Z-RIGID BODY VELOCITY
C      ZRBV=ZM/TOTALM
C
      RETURN
      END
```

APPENDIX D: User Defined Interface Control

This subroutine may be provided by the user to turn the interfaces on and off. The arguments are defined in the listing provided below.

```
      SUBROUTINE UCTRL2 (NSI,NTY,TIME,CYCLE,MSR,NMN,NSV,NSN,
     1     THMR,THSV,VT,XI,UT,ISKIP,IDRINT,NUMNP,DT2,NINPUT,UA)
C*******************************************************************
C|    LIVERMORE SOFTWARE TECHNOLOGY CORPORATION (LSTC)              |
C|    ------------------------------------------------------------  |
C|    COPYRIGHT 1987, 1988, 1989 JOHN O. HALLQUIST, LSTC            |
C|    ALL RIGHTS RESERVED                                           |
C*******************************************************************
C
      INTEGER CYCLE
C
C
C     USER SUBROUTINE FOR INTERFACE CONTROL
C
C     NOTE:  LS-DYNA USED AN INTERNAL NUMBERING SYSTEM TO
C            ACCOMODATE ARBITRARY NODE NUMBERING.  TO ACCESS
C            INFORMATION FOR USER NODE N, ADDRESS ARRAY LOCATION M,
C            M=LQF(N,1).  TO OBTAIN USER NODE NUMBER, N,
C            CORRESPONDING TO ARRAY ADDRESS M, SET N=LQFINV(M,1)
C
C     ARGUMENTS:
C        NSI           =NUMBER OF SLIDING INTERFACE
C        NTY           =INTERFACE TYPE.
C                      .EQ.4:SINGLE SURFACE
C                      .NE.4:SURFACE TO SURFACE
C        TIME          =CURRENT SOLUTION TIME
C        CYCLE         =CYCLE NUMBER
C        MSR(NMN)      =LIST OF MASTER NODES NUMBERS IN INTERNAL
C                       NUMBERING SCHEME
C        NMN           =NUMBER OF MASTER NODES
C        NSV(NSN)      =LIST OF SLAVE NODES NUMBERS IN INTERNAL
C                       NUMBERING SCHEME
C        NSN           =NUMBER OF SLAVE NODES
C        THMR(NMN)     =MASTER NODE THICKNESS
C        THSV(NSN)     =SLAVE NODE THICKNESS
C        VT(3,NUMNP)   =NODAL TRANSLATIONAL VELOCITY VECTOR
C        XI(3,NUMNP)   =INITIAL COORDINATES AT TIME=0
C        UT(3,NUMNP)   =NODAL TRANSLATIONAL DISPLACEMENT VECTOR
C        IDRINT        =FLAG FOR DYNAMIC RELAXATION PHASE
C                      .NE.0:DYNAMIC RELAXATION IN PROGRESS
C                      .EQ.0:SOLUTION PHASE
C        NUMNP         =NUMBER OF NODAL POINTS
C        DT2           =TIME STEP SIZE AT N+1/2
C        NINPUT        =NUMBER OF VARIABLES INPUT INTO UA
C        UA(*)         =USER’S ARRAY, FIRST NINPUT LOCATIONS
C                       DEFINED BY USER.  THE LENGTH OF THIS
C                       ARRAY IS DEFINED ON CONTROL CARD 10.
C                       THIS ARRAY IS UNIQUE TO INTERFACE NSI.
C
C     SET FLAG FOR ACTIVE CONTACT
C       ISKIP=0 ACTIVE
C       ISKIP=1 INACTIVE
```

```
C
C*******************************************************************
      DIMENSION MSR(*),NSV(*),THMR(*),THSV(*),VT(3,*),XI(3,*),
                UT(3,*)UA(*)
C
C     THE FOLLOWING SAMPLE OF CODEING IS PROVIDED TO ILLUSTRATE HOW
C     THIS SUBROUTINE MIGHT BE USED.  HERE WE CHECK TO SEE IF THE
C     SURFACES IN THE SURFACE TO SURFACE CONTACT ARE SEPARATED.  IF
C     SO THE ISKIP=1 AND THE CONTACT TREATMENT IS SKIPPED.
C
      IF (NTY.EQ.4) RETURN
      DT2HLF=DT2/2.
      XMINS= 1.E20
      XMAXS=-XMINS
      YMINS= 1.E20
      YMAXS=-YMINS
      ZMINS= 1.E20
      ZMAXS=-ZMINS
      XMINM= 1.E20
      XMAXM=-XMINM
      YMINM= 1.E20
      YMAXM=-YMINM
      ZMINM= 1.E20
      ZMAXM=-ZMINM
      THKS=0.0
      THKM=0.0
      DO 10 I=1,NSN
      DSP1=UT(1,NSV(I))+DT2HLF*VT(1,NSV(I))
      DSP2=UT(2,NSV(I))+DT2HLF*VT(2,NSV(I))
      DSP3=UT(3,NSV(I))+DT2HLF*VT(3,NSV(I))
      X1=XI(1,NSV(I))+DSP1
      X2=XI(2,NSV(I))+DSP2
      X3=XI(3,NSV(I))+DSP3
      THKS =MAX(THSV(I),THKS)
      XMINS=MIN(XMINS,X1)
      XMAXS=MAX(XMAXS,X1)
      YMINS=MIN(YMINS,X2)
      YMAXS=MAX(YMAXS,X2)
      ZMINS=MIN(ZMINS,X3)
      ZMAXS=MAX(ZMAXS,X3)
   10 CONTINUE
      DO 20 I=1,NMN
      DSP1=UT(1,MSR(I))+DT2HLF*VT(1,MSR(I))
      DSP2=UT(2,MSR(I))+DT2HLF*VT(2,MSR(I))
      DSP3=UT(3,MSR(I))+DT2HLF*VT(3,MSR(I))
      X1=XI(1,MSR(I))+DSP1
      X2=XI(2,MSR(I))+DSP2
      X3=XI(3,MSR(I))+DSP3
      THKM =MAX(THMR(I),THKS)
      XMINS=MIN(XMINM,X1)
      XMAXS=MAX(XMAXM,X1)
      YMINS=MIN(YMINM,X2)
      YMAXS=MAX(YMAXM,X2)
      ZMINS=MIN(ZMINM,X3)
      ZMAXS=MAX(ZMAXM,X3)
   20 CONTINUE
      IF (XMAXS+THKS.LT.XMINM-THKM) GO TO 40
      IF (YMAXS+THKS.LT.YMINM-THKM) GO TO 40
      IF (ZMAXS+THKS.LT.ZMINM-THKM) GO TO 40
      IF (XMAXS+THKM.LT.XMINS-THKS) GO TO 40
```

```
      IF (YMAXS+THKM.LT.YMINS-THKS) GO TO 40
      IF (ZMAXS+THKM.LT.ZMINS-THKS) GO TO 40
      ISKIP=0
      RETURN
   40 ISKIP=1
      RETURN
      END
```

APPENDIX E: User Defined Interface Friction

This subroutine may be provided by the user to set the Coulomb friction coefficients. The arguments are defined in the listing provided below.

```
      SUBROUTINE USRFRC (NSI,TIME,CYCLE,DT2,NSLAVE,AREAS,XS,YS,ZS,
     . MSN,MASTRS,AREAM,XCM,YCM,ZCM,STFSN,STFMS,FORCEN,RVX,RVY,RVZ,
     . FRIC1,FRIC2,FRIC3,FRIC4,NINPUT,UA,SIDE)
C*******************************************************************
C|     LIVERMORE SOFTWARE TECHNOLOGY CORPORATION (LSTC)             |
C|     ------------------------------------------------------------ |
C|     COPYRIGHT 1987, 1988, 1989 JOHN O. HALLQUIST, LSTC           |
C|     ALL RIGHTS RESERVED                                          |
C*******************************************************************
C
      INTEGER CYCLE
      CHARACTER*(*) SIDE
      DIMENSION UA(*),MASTRS(4),XCM(4),YCM(4),ZCM(4)
C
C
C     USER SUBROUTINE FOR INTERFACE FRICTION CONTROL
C
C     NOTE:  LS-DYNA USES AN INTERNAL NUMBERING SYSTEM TO
C            ACCOMODATE ARBITRARY NODE NUMBERING.  TO ACCESS
C            INFORMATION FOR USER NODE N, ADDRESS ARRAY LOCATION M,
C            M=LQF(N,1).  TO OBTAIN USER NODE NUMBER, N,
C            CORRESPONDING TO ARRAY ADDRESS M, SET N=LQFINV(M,1)
C
C     ARGUMENTS:
C        NSI          =NUMBER OF SLIDING INTERFACE
C        TIME         =CURRENT SOLUTION TIME
C        CYCLE        =CYCLE NUMBER
C        DT2          =TIME STEPS SIZE AT N+1/2
C        NSLAVE       =SLAVE NODE NUMBER IN LS-DYNA INTERNAL
C                      NUMBERING
C        AREAS        =SLAVE NODE AREA (INTERFACE TYPES 5&10 ONLY)
C        XS           =X-COORDINATE SLAVE NODE (PROJECTED)
C        YS           =Y-COORDINATE SLAVE NODE (PROJECTED)
C        ZS           =Z-COORDINATE SLAVE NODE (PROJECTED)
C        MSN          =MASTER SEGMENT NUMBER
C        MASTRS(4)    =MASTER SEGMENT NODE IN LS-DYNA INTERNAL
C                      NUMBERING
C        AREAM        =MASTER SEGMENT NUMBER
C        XCM(4)       =X-COORDINATES MASTER SURFACE (PROJECTED)
C        YCM(4)       =Y-COORDINATES MASTER SURFACE (PROJECTED)
C        ZCM(4)       =Z-COORDINATES MASTER SURFACE (PROJECTED)
C        STFSN        =SLAVE NODE PENALTY STIFFNESS
C        STFMS        =MASTER SEGMENT PENALTY STIFFNESS
C        FORCEN       =NORMAL FORCE
C        RVX,RVY,RVZ,=RELATIVE X,Y,Z-VELOCITY BETWEEN SLAVE NODE AND
                       MASTER SEGMENT
```

```
C*********************************************************************
C     THE FOLLOWING VALUES ARE TO BE SET BY USER
C
C        FRIC1        =STATIC FRICTION COEFFICIENT
C        FRIC2        =DYNAMIC FRICTION COEFFICIENT
C        FRIC3        =DECAY CONSTANT
C        FRIC4        =VISCOUS FRICTION COEFFICIENT (SETTING FRIC4=0
                       TURNS THIS OPTION OFF)
C
C*********************************************************************
C
C        NINPUT       =NUMBER OF VARIABLES INPUT INTO UA
C        UA(*)        =USERS' ARRAY, FIRST NINPUT LOCATIONS
C                      DEFINED BY USER.  THE LENGTH OF THIS
C                      ARRAY IS DEFINED ON CONTROL CARD 15.
C                      THIS ARRAY IS UNIQUE TO INTERFACE NSI.
C
C        SIDE         ='MASTER' FOR FIRST PASS.  THE MASTER
C                      SURFACE IS THE SURFACE DESIGNATED IN THE
C                      INPUT.
C                     ='SLAVE' FOR SECOND PASS AFTER SLAVE AND
C                      MASTER SURFACES HAVE BE SWITCHED FOR
C                      THE TYPE 3 SYMMETRIC INTERFACE TREATMENT
C
C*********************************************************************
C
      RETURN
      END
```

APPENDIX F: Version 912 Control Cards

Control Cards

Card 1

(I5,6I10,E10.0,E5.0)

Columns	Quantity	Format
1-5	Number of property sets (NUMMAT)	I5
6-15	Number of nodal points (NUMNP)	I10
16-25	Number of solid hexahedron elements (NUMELH)	I10
26-35	Number of beam elements (NUMELB)	I10
36-45	Number of 4-node shell elements (NUMELS)	I10
46-55	Number of 8-node solid shell elements (NUMELT)	I10
56-65	Number of interface definitions for component analyses (NUMIFS)	I10
66-75	Output interval for interface file (Δt)	E10.0
76-80	Shell element minimum time step assignment	E5.0

When a shell controls the time step, element material properties will be modified such that the time step does not fall below step size assigned. Applicable only to shell elements using material models 3, 18, 19, and 24.

Card 2
(8I5,E10.0,I5,E10.0,3I5)

Columns	Quantity	Format
1-5	Number of nodal time history blocks (NUMDS)	I5
6-10	Number of hexahedron element time history blocks (NSTH)	I5
11-15	Number of beam element time history blocks (NSTB)	I5
16-20	Number of shell element time history blocks (NSTS)	I5
21-25	Number of thick shell element time history blocks (NSTT)	I5
26-30	Problem status report interval EQ.0: default set to 1000	I5
31-35	Number of nodal rigid body constraint sets (NUMRBS)	I5
36-40	Number of traction boundary cards for beam elements (NUMBPC)	I5
41-50	Scale factor for rigid wall penalties for treating rigid bodies interacting with <u>fixed</u> rigid walls. The penalties are set so that a scale factor of unity should be optimal; however, this may be very problem dependent. If rigid/deformable material 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 will interact with <u>fixed</u> rigid walls. A value of 1.0 is recommended.	E10.0
51-55	Number of nodes in each interface for cyclic symmetry, NNCSYM	I5
56-65	Contact surface maximum penetration check multiplier, XPENE. If the small penetration checking (Section 31, Card 1, Column 45-50) option on the contact surface control card is active, then nodes whose penetration the exceeds the product of XPENE and the element thickness are set free. EQ.0: default is set to 4.0	E10.0
66-70	Implicit coupling for springback calculations EQ.0: off EQ.1: on	I5
71-75	Superplastic analysis input option, ISUPER EQ.0: no input EQ.1: read superplastic input section	I5
76-80	Number of generalized joint stiffnesses, NJTS.	I5

The number of blocks specified in columns 1-25 are limited to 2000 if the TAURUS post processor is used.

Card 3
(6I5,A5,3I5)

Columns	Quantity	Format
1-5	Number of nodes in DYNA3D-JOY interface (NUMSNC)	I5
6-10	Number of sliding boundary planes (NUMRC)	I5
11-15	Number of symmetry planes with failure (NUMRCF)	I5
16-20	Number of points in density vs depth curve (NUMDP)	I5
21-25	Brode function flag (IBRODE) EQ.0: Brode parameters are not defined EQ.1: Brode parameters are defined in input	I5
26-30	Number of rigid body merge cards (NRBC)	I5
31-35	Nodal coordinate format (NIF): either E10.0 or E20.0	A5
36-40	Number of cross section definitions (for force output) (NUMCSD)	I5
41-45	Blank	I5
46-50	Number of generalized body force loads (NUMGBL)	I5

Card 4 (16I5)

Columns	Quantity	Format
1-5	Number of load curves (NLCUR)	I5
6-10	Number of concentrated nodal loads (NUMCL)	I5
11-15	Number of segments having pressure loads applied (NUMPC)	I5
16-20	Number of velocity/acceleration boundary condition cards (NUMVC)	I5
21-25	Number of rigid walls (stonewalls) (NUMRV)	I5
26-30	Number of nodal constraint cards (NUMCC)	I5
31-35	Initial condition parameter (INITV) EQ.0: initialize velocities to zero EQ.1: initial velocities are read in EQ.2: all nodes have same input value EQ.3: same as 2 but exempted nodes are defined EQ.4: box option (refer to Section 30)	I5
36-40	Number of sliding interfaces (NUMSI)	I5
41-45	Base acceleration in x-direction (NTHPX) EQ.0: no EQ.1: yes	I5
46-50	Base acceleration in y-direction (NTHPY) EQ.0: no EQ.1: yes	I5
51-55	Base acceleration in z-direction (NTHPZ) EQ.0: no EQ.1: yes	I5
56-60	Angular velocity about x-axis (NTHSX) EQ.0: no EQ.1: yes	I5
61-65	Angular velocity about y-axis (NTHSY) EQ.0: no EQ.1: yes	I5
66-70	Angular velocity about z-axis (NTHSZ) EQ.0: no EQ.1: yes	I5
71-75	Number of solid hexahedron elements for momentum deposition (NELMD)	I5
76-80	Number of detonation points (NDTPTS)	I5

Card 5 (3E10.0,2I5,2E10.0,2I5,E10.0)

Columns	Quantity	Format
1-10	Termination time (ENDTIM)	E10.0
11-20	Time interval between dumps of time history data* (PRTC)	E10.0
21-30	Time interval between complete state dumps and interface force databases (PLTC). (This interval must be defined, otherwise the time interval will be equal to the time step size.)	E10.0
31-35	Number of time steps between restart dump files (IRDECK) (default=99999)	I5
36-40	Number of time steps between running restart dumps (NCBRFF). (The same file is overwritten.)	I5
41-50	Initial time step size (DT2OLD) EQ.0.0: LS-DYNA determines initial step size	E10.0
51-60	Scale factor for sliding interface penalties (SLSFAC) EQ.0.0: default = .10	E10.0
61-65	Thermal effects option (ITEMP) EQ.0: no thermal effects EQ.n: temperature-time history is defined by load curve n LT.0: nodal temperatures are defined in TOPAZ3D generated disk files	I5
66-70	Reset default viscosities (IRQ) EQ.1: new defaults are read on Card 13	I5
71-80	Scale factor for computed time step (SCFT) (Default = .90; if high explosives are used, the default is lowered to .67.)	E10.0

*The time interval between dumps of time history data refers to the output frequency for the file specified by the "F=" parameter of the LS-DYNA command line. Only a subset of the nodes and elements are output as specified by the node and element print blocks which are read in if the flags are set on control card 2.

Card 6 (9I5,5X,5I5)

Columns	Quantity	Format
1-5	Number of joint definitions (NJT)	I5
6-10	Number of rigid bodies for which extra nodes are defined (NXTRA)	I5
11-15	Number of shell-solid element interface definitions (NBLK)	I5
16-20	Number of tie-breaking shell slidelines (NTBSL)	I5
21-25	Number of tied node sets with failure definitions (NTNWF)	I5
26-30	Load curve number that limits maximum time step size (optional) (ICTM)	I5
31-35	FLAG = 1 for springs, dampers, and lumped mass input (INPSD)	I5
36-40	Number of rigid bodies for which inertial properties are defined (optional) (NUMRBI)	I5
41-45	FLAG = 1 to dump shell strain tensors at inner and outer surface for plotting by TAURUS and ASCII file, ELOUT (ISTRN)	I5
51-55	Warping stiffness for Belytschko-Tsay shells EQ.1: Belytschko-Wong-Chiang warping stiffness added EQ.2: Belytschko-Tsay (default)	I5
56-60	Hughes-Liu shell normal update option (IRNXX) EQ.-2: unique nodal fibers EQ.-1: compute normals each cycle EQ.0: default set to -1 EQ.1: compute on restarts EQ.n: compute every n cycles	I5
61-65	Shell thickness change option (ISTUPD) EQ.0: no change EQ.1: thickness change caused by membrane straining	I5
66-70	Shell theory (IBELYT) EQ.1: Hughes-Liu EQ.2: Belytschko-Tsay (default)	I5
71-75	Number of nonreflecting boundary segments (NNRBS)	I5

Card 7 (2I5,E10.0,5I5,3E10.0,I5)

Columns	Quantity	Format
1-5	Number of single point constraint nodes (NODSPC)	I5
6-10	Number of coordinate systems (NSPCOR)	I5
11-20	Reduction factor for initial time step size to determine minimum step (TSMIN). DTMIN=DTSTART*TSMIN where DTSTART is the initial step size determined by LS-DYNA. When DTMIN is reached LS-DYNA terminates with a restart dump.	E10.0
21-25	Number of user specified beam integration rules (NUSBIR)	I5
26-30	Maximum number of integration points required in the user specified rules for beam elements (MPUBR)	I5
31-35	Number of user specified shell integration rules (NUSSIR)	I5
36-40	Maximum number of integration, points required in the user specified rules for shell elements (MPUSR)	I5
41-45	Number of iterations between convergence checks, for dynamic relaxation option (NRCYCK) (default=250)*	I5
46-55	Convergence tolerance for dynamic relaxation option (DRTOL) (default=0.001)*	E10.0
56-65	Dynamic relaxation factor (DRFCTR) (default=.995)*	E10.0
66-75	Scale factor for computed time step during dynamic relaxation; if zero, the value is set to SCFT defined on Control Card 5 (TSSFDR). After converging, the scale factor is reset to SCFT.	E10.0
76-80	Basis of time step 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)	I5

*Dynamic relaxation parameters are highly problem dependent and should be set carefully. The defaults were chosen after extensive numerical experimentation. This option is not invoked unless flagged in the control cards.

Card 8 (8I5,E10.0,6I5)

Columns	Quantity	Format
1-5	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	I5
6-10	Printout flag for element time step sizes on the first cycle EQ.0: no printout EQ.1: the governing time step sizes for each element is printed	I5
11-15	Number of 1D slideline definitions	I5
16-20	TAURUS database during dynamic relaxation option EQ.0: database is not written EQ.1: TAURUS database state for each convergence check	I5
21-25	Arbitrary node, element, and material labels, INEM EQ.0: consecutive EQ.1: arbitrary	I5
26-30	General printout option flag EQ.0: No printout except for standard high speed printer file EQ.1: Read three cards defining output intervals for various databases	I5
31-35	Print suppression during input phase flag EQ.0: no suppression EQ.1: nodal coordinates, element connectivities, rigid wall definitions and initial velocities are not printed	I5
36-40	Load curve number which specifies system damping constant, LCDAMP EQ.-1: system damping is defined for each material by load curves. The data is defined in a separate input section called "System Damping." EQ.0: no 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*	I5
41-50	System damping constant, *d* (This option is bypassed if the load curve number defined above is nonzero.)	E10.0
51-55	Number of additional integration points history variables written to the Taurus database for solid elements	I5
56-60	Number of additional integration points history variables written to the Taurus database for shell elements for each integration point	I5

Columns	Quantity	Format
61-65	Number of shell integration points written to the Taurus database (default=3)	I5
66-70	Nonzero flag for input of lumped parameter control volumes (LPCV)	I5
71-75	Flag, IARB, for defining material models by specifying constitutive, equation-of-state, and cross-section property definitions EQ.0: standard LS-DYNA EQ.1: read in extra card giving the number of constitutive models, equations-of-state, and cross section to be defined.	I5
76-80	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	I5

Below we summarize the Material Property Input for New Input Scheme.

(IARB=1 on Control Card 8, Columns 71-75):

1. Input additional control card for materials (Page 36 - C. C. 15 + NUMUMT). This gives the number of constitutive models, equation-of-state, and section definition.
2. For each constitutive model, input the following:

   Card 1 Constitutive Model Control card (pages 147-148).

   Card 2 Constitutive Model Description (pages 42)

   Cards 3-8 Constitutive Model Constants (pages 43-119)
3. For each equation-of-state, input the following:

   Card 1 Equation-of-state control card

   Cards 2,... Equation of state data (pages 120-137)

   (This corresponds to cards 9,.. in the original scheme)
4. For each section property definition, input the following

   Card 1 Section property control card (pages 127-128).

   Card 2 Cross section description (page 130)

   Cards 3-4 Section Properties

Beams:	Pages 151-152
Shells:	Pages 153-155
Brick Shells	Pages 155-155
5. For each material property set (control card 1, columns 1-5), input a property set definition. Each material property set must select a constitutive model and (optionally) an equation of state or section property (page 156).

For each material property set (Control Card 1, Columns 1-5) input the cards which are required in the following table. The card numbers refer to the old input scheme.

		Bricks	Beams	Shells	Brick Shells
Card 1:	Material Control Card pages 147-148	Required	Required	Required	Required
Card 2:	Material description page 42	Required	Required	Required	Required
Card 3:	Constitutive Model Constants pages 43-119	Required	Required	Required	Required
Equations-of-state		Req'd for #'s 8,9,10, 11,15,16	Do not Input	Do not Input	Do not Input
Card 9:	Section Property description	Do not Input	Required	Required	Required
Card 10,11:	Section Property	Do not Input	Pages 151-152	Pages 153-154	Pages 155-155
Card 12,...:	Composite Material Angles (Materials 2, 21, 22, 23)	Do not Input	Do not Input	Page 155	Page 155

Card 9 (8I5,3E10.0,I2,I3,I5)

Columns	Quantity	Format
1-5	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	I5
6-10	Stonewall energy dissipation option EQ.1: energy dissipation is not computed EQ.2: energy dissipation is computed and included in the energy balance (default)	I5
11-15	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	I5
16-20	Maximum number of 4-node segments (optional) specified in definition of any stonewall. This option is necessary to get the stonewall force distribution.	I5
21-25	Initial penetration check in contact surfaces with indication of initial penetration in output file. EQ.1: no checking (default) EQ.2: full checks of initial penetration is performed.	I5
26-30	Dynamic relaxation flag for stress initialization EQ.0: not active EQ.1: dynamic relaxation is activated EQ.2: initialization to a prescribed geometry (see comment below)	I5
31-35	Number of nodal force groups	I5
36-40	Automatic sorting of triangular shell elements to treat degenerate quadrilateral shell elements as C0 triangular shells EQ.1: full sorting EQ.2: no sorting required (default)	I5
41-50	Optional termination time for dynamic relaxation. Termination occurs at this time or when convergence is attained. (Default=infinity)	E10.0
51-60	Time interval between dumps of interface force database. If zero, the default is the same as for complete state dumps.	E10.0

Columns	Quantity	Format
61-70	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 flag for limited mass scaling below.	E10.0
71-72	Limit mass scaling to the first step and fix the mass vector afterwards. The time step will not be fixed but may drop during the calculation from the specified minimum. EQ.0: no EQ.1: yes	I2
73-75	Node and element suppression 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 are suppressed	I3
76-80	Composite material stress output EQ.0: global EQ.1: local	I5

Stress initialization in LS-DYNA for small strains may be accomplished by linking to an implicit code (option 2 in Columns 26-30). A displacement state is required that gives for each nodal point its label, xyz displacements, and xyz rotations. This data is read from unit 7 (*m*=) with the format (i8,6e15.0).

Card 10 (5I5,I10,I5,E10.0,4I5)

Columns	Quantity	Format
1-5	Shell thickness considered in type 3, 5, and 10 contact options where options 1 and 2 below activate the new contact algorithms EQ.0: thickness is not considered EQ.1: thickness is considered but rigid bodies are excluded EQ.2: thickness is considered, including rigid bodies	I5
6-10	Penalty stiffness value EQ.1: minimum of master segment and slave node (default) EQ.2: use master segment stiffness (old way) EQ.3: use slave node value EQ.4: use slave node value - area or mass (type=5) weighted EQ.5: same as 4 but inversely proportional to the shell thickness. This may require special scaling and is not generally recommended.	I5
11-15	Shell thickness changes considered in type four single surface contact EQ.0: no consideration EQ.1: shell thickness changes are included	I5
16-20	Sliding interface energy dissipation option EQ.1: energy dissipation is not computed (default) EQ.2: energy dissipation is computed and included in the energy balance	I5
21-25	Optional automatic reorientation of contact interface segments during initialization EQ.1: inactive (default) EQ.2: active	I5
26-35	Termination cycle	I10
36-40	Debug option EQ.0: no printout EQ.1: progress of input phase is tracked in message file	I5
41-50	Shell element warpage angle in degrees. If a warpage greater than this angle is found, a warning message is printed. EQ.0.0: default set to 20 degrees	E10.0
51-55	Number of user defined material subroutines (NUMUMT)	I5
56-60	Number of nodes (NTNPFL) tied to nodes in an interface database generated in another run via Section 11 input. This file is specified on the execute line by specifying L=inf2, where inf2 is the file name.	I5
61-65	Percent change in energy ratio for termination of calculation	I5

Columns	Quantity	Format
66-70	Flag for rigid/deformable material switching (IRDMS). If this flag is set to 2 then any deformable material in the model may be switched between rigid and deformable during the calculation. Materials that are defined as type 20 in the input are permanently rigid. Additional input is required if this flag is set. EQ.1: off (default) EQ.2: on	I5
71-75	Storage per contact interface for user supplied interface control subroutine. 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.	I5
76-80	Storage per contact interface for user supplied interface friction subroutine. 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.	I5

The shell thickness change option must be active on Control Card 6 (col. 65) before the shell thickness changes can be included in contact types 3, 4, 5, 10, 11, and 12. If the shell thickness change is active and if a nonzero flag is set in col. 5 the thickness changes are automatically included in contact types 3, 5, and 10. The new contact algorithms that include the shell thickness are relatively recent and have yet to be fully optimized. The searching in the new algorithms is considerably more extensive and therefore somewhat more expensive.

The warpage angle is found by computing the normal vectors at each element node based on the edges. If diagonally opposite vectors have an included angle that exceeds the specified warpage angle a warning message is printed.

Card 11 (3I5,E10.0,9I5,1x,4i1,3x,2I1)

Columns	Quantity	Format
1-5	Flag for geometric entities for geometry based rigid body contact EQ.0: do not define geometric entities EQ.1: define geometric entities	I5
6-10	Number of linear constraint equations (NOCEQS).	I5
11-15	Automatic control for dynamic relaxation option (IRELAL) based on algorithm of [Papadrakakis] EQ.0: no EQ.1: yes	I5
16-25	Convergence tolerance on automatic control of dynamic relaxation (EDTTL)	E10.0
26-30	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	I5
31-35	Lagrange multiplier contact option, reading Section 65 input data EQ.0: off EQ.1: on	I5
36-40	Number of CPU’s for parallel execution (NCPU) (default =1)	I5
41-45	Sorting for parallel assembly of the right hand EQ.0: on EQ.1: off (results in the use of less memory)	I5
46-50	Flag for automatic subcycling EQ.0: off (default) EQ.1: on	I5
51-55	Flag for Rayleigh damping input EQ.0: off (default) EQ.1: on	I5
56-60	Flag for Rayleigh damping energy calculations EQ.0: off (default) EQ.1: on	I5
61-65	Number of time steps between contact searches EQ.0: default set to 11 EQ.n: n time steps between contact searches	I5
66-70	Intermittent searching in type 3 contact EQ.0: off EQ.1: on	I5

Card 11 (continued)

Columns	Quantity	Format
72	Flag for including stress tensor in the shell TAURUS database	I1
73	Flag for including effective plastic strains in the shell database	I1
74	Flag for including stress resultants in the shell database	I1
75	Flag for including internal energy and thickness in the database	I1
79	Flag for seatbelt and accelerometer input on next card EQ.0: the next card is not read EQ.1: Read card 11.1 below	I1

Optional Card 11.1 for Seat Belts and Accelerometers (7I5)

Columns	Quantity	Format
1-5	Number of belt materials	I5
6-10	Number of seat belt elements	I5
11-15	Number of sliprings	I5
16-20	Number of retractors	I5
21-25	Number of sensors	I5
26-30	Number of pretensioners	I5
31-35	Number of accelerometers	I5

Card 11.2 required for MADYMO3D/CAL3D Coupling

Columns	Quantity	Format
1-10	UNLENG. Unit conversion factor for length. MADYMO3D/GM-CAL3D lengths are multiplied by UNLENG to obtain LS-DYNA lengths.	E10.0
11-20	UNTIME. Unit conversion factor for time. MADYMO3D/GM-CAL3D time is multiplied by UTIME to obtain LS-DYNA time.	E10.0
21-30	UNFORC. Unit conversion factor for force. MADYMO3D/GM-CAL3D force is multiplied by UNFORC to obtain LS-DYNA force	E10.0
31-35	Material repositioning flag for MADYMO/GM-CAL3D coupling EQ.0: off EQ.1: on. Section 66 input data is read.	I5
38	Flag for flipping X-coordinate of CAL3D/MADYMO relative to the LS-DYNA model. EQ.0: off EQ.1: on	
39	Flag for flipping Y-coordinate of CAL3D/MADYMO relative to the LS-DYNA model. EQ.0: off EQ.1: on	
40	Flag for flipping Z-coordinate of CAL3D/MADYMO relative to the LS-DYNA model. EQ.0: off EQ.1: on	

Cards 12, 13, and 14 (8E10.0)

Define these cards only if printout option on Control Card 8 (col. 30) is flagged. If the output intervals are defined as 0.0 no output is provided for the corresponding file.

Columns		Quantity	Format
1-10	Card 12	Output interval for cross-section forces	E10.0
11-20		Output interval for rigid wall forces	E10.0
21-30		Output interval for nodal point data	E10.0
31-40		Output interval for element data	E10.0
41-50		Output interval for global data	E10.0
51-60		Output interval for discrete elements	E10.0
61-70		Output interval for material energies	E10.0
71-80		Output interval for nodal interface forces	E10.0
1-10	Card 13	Output interval for resultant interface forces	E10.0
11-20		Output interval for smug animator instant	E10.0
21-30		Output interval for spc reaction forces	E10.0
31-40		Output for nodal constraint resultants (spotwelds and rivets)	E10.0
41-50		Output interval for airbag statistics	E10.0
51-60		Output interval for AVS database	E10.0
61-70		Output interval for nodal force groups	E10.0
71-80		Output interval for boundary condition forces and energy on nodal points with discrete forces, pressures, or designated velocities	E10.0
1-10	Card 14	Output interval for rigid body data	E10.0
11-20		Output interval for geometric contact entities	E10.0
21-30		Output interval for MPGS database	E10.0
31-40		Output interval for MOVIE database	E10.0
41-50		Output interval for sliding interface database	E10.0

Columns	Quantity	Format
51-60	Output interval for seat belt database	E10.0
61-70	Output interval for joint forces	E10.0

By defining the output interval a file is created for the output. Each output type is placed into a separate file. Normally these names are assigned by LS-DYNA. Using the "W=" option on startup, a root name can be specified. Extensions are then added to this root to form the output file names **(This option is available only on designated installations and is to some degree machine dependent).** The file names and corresponding unit numbers are:

	I/O UNIT #	FILE NAME
Cross-section forces	i/o unit#31	SECFORC
Rigidwall forces	i/o unit#32	RWFORC
Nodal point data	i/o unit#33	NODOUT
Element data	i/o unit#34	ELOUT
Global data	i/o unit#35	GLSTAT
Discrete elements	i/o unit#36	DEFORC
Material energies	i/o unit#37	MATSUM
Nodal interface forces	i/o unit#38	NCFORC
Resultant interface forces	i/o unit#39	RCFORC
Smug animator database	i/o unit#40	DEFGEO
Nastran BDF file	i/o unit#49	NASBDF
SPC reaction forces	i/o unit#41	SPCFORC
Nodal constraint resultants (spotwelds/rivets)	i/o unit #42	SWFORC
Airbag statistics	i/o unit #43	ABSTAT
ASCII database	i/o unit #44	AVSFLT
Nodal force group	i/o unit #45	NODFOR
Boundary conditions nodal forces and energies	i/o unit #46	BNDOUT
Rigid body data	i/o unit #47	RBDOUT
Contact entities	i/o unit #48	GCEOUT
Interface energies	i/o unit #51	SLEOUT
Seat belts	i/o unit #52	SBTOUT

THE REST OF THE INPUT DATA FOLLOWS THE 920 USERS MANUAL.

APPENDIX G: Occupant Simulation Including the Coupling to Programs CAL3D and MADYMO

INTRODUCTION

LS-DYNA is coupled to occupant simulation codes to generate solutions in automotive crashworthiness that include occupants interacting with the automotive structure. In such applications LS-DYNA provides the simulation of the structural and deformable aspects of the model and the OSP (Occupant Simulation Program) simulates the motion of the occupant. There is some overlap between the two programs which provides flexibility in the modeling approach. For example, both the OSP and LS-DYNA have the capability of modeling seat belts and other deformable restraints. The advantage of using the OSP is related to the considerable databases and expertise that have been developed in the past for simulating dummy behavior using these programs.

The development of the interface provided LSTC a number of possible approaches. The approach selected is consistent with the LSTC philosophy of providing the most flexible and useful interface possible. This is important because the field of non-linear mechanics is evolving rapidly and techniques which are used today are frequently rendered obsolete by improved methodologies and lower cost computing which allows more rigorous techniques to be used. This does make the learning somewhat more difficult as there is not any single procedure for performing a coupling.

One characteristics of LS-DYNA is the large number of capabilities, particularly those associate with rigid bodies. This creates both an opportunity and a difficulty: LS-DYNA3D has many ways approximating different aspects of problems, but they are frequently not obvious to users without considerable experience. Therefore, in this Appendix we emphasize modeling methods rather than simply listing capabilities.

THE LS-DYNA/OCCUPANT SIMULATION PROGRAM LINK

Coupling between the OSP and LS-DYNA is performed by combining the programs into a single executable. In the case of CAL3D, LS-DYNA calls CAL3D as a subroutine, but in the case of MADYMO, LS-DYNA is called as a subroutine. The two programs are then integrated in parallel with the results being passed between the two until a user defined termination time is reached.

The OSP and LS-DYNA have different approaches to the time integration schemes. The OSP time integrators are based on accurate implicit integrators which are valid for large time steps which are on the order of a millisecond for the particular applications of interest here. An iterative solution is used to insure that the problem remains in equilibrium. The implicit integrators are extremely good for smoothly varying loads, however, sharp nonlinear pulses can introduce considerable error. An automatic time step size control which decreases the time step size quickly restores the accuracy for such events. The LS-DYNA time integrator is based on an explicit central difference scheme. Stability requires that the time step size be less than the highest frequency in the system. For a coarse airbag mesh, this number is on the order of 100 microseconds while an actual car crash simulation is on the order of 1 microsecond. The smallest LS-DYNA models have at least 1,000 elements. Experience indicates that the cost of a single LS-DYNA time step for a small model is at least as great as the cost of a time step in the OSP. Therefore, in the coupling, the LS-DYNA time step is used to control the entire simulation including the OSP part. This approach has negligible cost penalties and avoids questions of stability and accuracy that would result by using a subcycling scheme between the two programs.

LS-DYNA has a highly developed rigid body capability which is used in different parts of automobile crash simulation. In particular, components such as the engine are routinely modeled with rigid bodies. These rigid bodies have been modified so that they form the basis of the coupling procedure in LS-DYNA to the OSP.

In LS-DYNA the geometry of a model is broken down into nodal points which identify positions in space. These nodes are then connected by elements so that the volume of a structure is identified. Each element has a “material” associated with it. If the element is deformable, then the material will specify it’s characteristics such as density and Young’s Modulus. A crash model can consist of 100 or more separate materials which are each assigned a “material number” and each material number has an associated “material type” which determines if it is elastic, plastic, viscoelastic, orthotropic, etc.

The material type may also specify that it is an rigid body. In this case, all elements of the same material number are treated as a single rigid body. These elements are integrated to determine the mass, centroid and moments of inertia for the group. This group is then treated as a rigid body with six degrees-of-freedom including three translations and three rotation. The positions of the rigid bodies are updated in LS-DYNA by a time integrator which works together with the central difference time integration.

There is an additional flag which specifies that the LS-DYNA rigid body is coupled to an OSP rigid body. This flag can be found in the description of the rigid body material, type 20, in Section 3 of this manual. In coupled update, the OSP rigid body time integrator takes over control of the LS-DYNA rigid body and the normal LS-DYNA updates are bypassed. The time integration procedure is then as follows:

1. At the beginning of a step, LS-DYNA determines the locations and updates the positions of all of the rigid bodies which are coupled to the OSP. This information is obtained from common block information in the OSP.

2. Using the information on rigid body locations, LS-DYNA proceeds to update the stresses and history variables of all of the deformable structures and computes the resultant forces acting on all rigid bodies.

3. The resultant forces are stored into an OSP common block along with the current time step. Control is then returned to the OSP so that the step can be completed by the OSP determining the new positions of the rigid bodies based on the applied forces.

At the end of the calculation LS-DYNA terminates normally, closing its files and then control is returned to OSP which will also terminate normally. The termination time for the coupled run is taken as the minimum of the termination time provided to LS-DYNA and the termination time provided to the OSP.

The executable for the coupling with MADYMO currently needs to be specially created at each site. TNO provides all of the appropriate load modules with their libraries, and the appropriate load modules for LS-DYNA may be obtained by the corporate contact point at the LS-DYNA distributor. A complete executable must then be made by linking the two libraries. A revised password file must be obtained from TNO prior to running the coupled code. Coupling with CAL3D requires special on site modification of the client's CAL3D version to eliminate conflicting I/O unit numbers and to ensure that the common block lengths between the codes are consistent. LSTC does not distribute or support CAL3D.

To make the coupled program run, an input deck must be provided to both the OSP and LS-DYNA. The two input decks must be provided in the same set of consistent units. This can potentially required a major conversion to either the OSP input or the LS-DYNA input. With two legitimate and consistent input decks, the coupled program should run to completion with no problems. Additional inputs are required to make the models interact between the OSP and LS-DYNA portions of the run.

The simplest form of a coupled simulation is simply to include a single body in a OSP run. No special modification are needed to the OSP input deck for use in the coupled simulation. Ellipsoids and planes in the OSP are usually attached to “segments” which correspond to LS-DYNA “rigid bodies”. Because the coupling procedure works on the basis of shared information on LS-DYNA rigid bodies with the OSP segments, the ellipsoids/planes listed in the OSP section must correspond to the segments which are to be coupled. These ellipsoids and planes may be actual geometry which is used for contact or they may be simply artificial shapes to permit the data transfer between the OSP and LS-DYNA.

DUMMY MODELING

The dummy is typically modeled entirely within the OSP. The coupling of the dummy into LS-DYNA requires the creation of a separate LS-DYNA rigid body material for each segment of the OSP. The easiest way to create a mesh for the model is to set the LS-DYNA rigid body coupling option to 2.0. This caused LS-DYNA to search all of the ellipsoids connected to the appropriate segment and generate meshes which are then slaved the OSP dummy. Thus, with minimal input, a complete dummy may be generated and the kinematics may be traced in LS-DYNA and displayed in the LS-DYNA post-processor, LS-TAURUS

Once the basic dummy coupling has been accomplished, the deformable finite element structure can be added. Assuming that an ellipsoid is available for the steering wheel, a flat airbag can be added in the proper location. One or more nodes must be attached to the steering wheel. This is done by identifying the attached nodes as “Extra Nodes for Rigid Body” which is input in the LS-DYNA in Section 37. The nodes are slaved to the LS-DYNA material which has been coupled to the MADYMO steering wheel model. Contact must now be identified between the airbag and the steering wheel, the windshield, and the various body parts which may be affected. This requires the use of one geometric contact entity (Section 57) for each plane or ellipsoid which may interact with the airbag. A control volume specifying inflation properties for the airbag must be specified (Section 56) to complete the model.

AIRBAG MODELING

Modeling of airbags is accomplished by use of shell or membrane elements in conjunction with a control volume (Section 55) and possibly a single surface contact algorithm to eliminate interpenetrations during the inflation phase (Section 31). Current recommended materials types for the airbags are:

Type 1. Elastic

Type 22. Layered orthotropic elastic for composites.

Type 34. Fabric model for folded airbags.

Model 34 is a “fabric” model which can be used for flat bags. As a user option this model may or may not support compression.

The elements which can be used are as follows:

Belytschko-Tsay quadrilateral with 1 point quadrature. This element behaves rather well for folded and unfolded cases with only a small tendency to hourglass. The element ends to be a little stiff. Stiffness form hourglass control is recommended.

Belytschko-Tsay membrane. This model is softer tan the normal Belytschko-Tsay element and can hourglass quite badly. Stiffness form hourglass is recommended.

C0 Triangular element. The C0 triangle is very good for flat bag inflation and has no tendency to hourglass.

Fully integrated membrane based on Belytschko-Tsay. We believe that this element when used with model 34 provides the best behavior. This is the most costly option.

As an airbag inflates, a considerable amount of energy is transferred to the surrounding air. This energy transfer decreases the kinetic energy of the bag as it inflates. In the control volume logic this is simulated either by using either a mass weighted damping option or a back pressure on the bag based on a stagnation pressure. In both cases, the energy that is absorbed is a function of the fabric velocity relative to a rigid body velocity for the bag. For the mass weighted case, the damping force on a node is proportional to the mass times the damping factor times the velocity vector. This is quite effective in maintaining a stable system, but has little physical justification. The latter approach using the stagnation pressure method estimates the pressure needed to accelerate the surrounding air to the speed of the fabric. The formula for this is:

$$P = Area \times \alpha \times \left(\left(\vec{V}_i - \vec{V}_{cg} \right) \cdot \hat{n} \right)^2$$

This formula accomplishes a similar function and has a physical justification. Values of the damping factor, α, are limited to the range of 0 to 1 but a value of 0.1 or less is more likely to be a good value.

KNEE BOLSTER

The knee to knee bolster interactions are characterized by the stiffness of the knee being comparable to that of the knee bolster. Therefore the modeling the knee as a rigid body may produce large errors in the interaction forces. Calibrated force-deflection curves could be determined, but they would have no predictive value for slight changes to knee bolster designs. For this reason, a more accurate modeling of the compliance of the knee bolster and the knee is required.

The knee can be modeled as a combine rigid/deformable body. The rigid body is coupled to the OSP. Overlaying the rigid body are brick elements which model the “skin” that exists over the knees of the dummy. These brick elements use material type 6 which is a viscoelastic model that does a reasonable job of approximating the hysteretic behavior of rubbers. The inner layer of the brick elements is attached to the rigid body through the “Extra Nodes for Rigid Bodies” (Section 37). Between the knee bolster is a type 3 contact definition (Section 31).

COMMON ERRORS

1. Improper airbag inflation or no inflation.

The most common problem is inconsistency in the units used for the input constants. An inflation load curve must also be specified. The normals for the airbag segments must all be consistent and facing outwards. If a negative volume results, this can sometimes be quickly cured by using the “flip” flag on the control volume definition to force inward facing normals to face outwards.

2. Excessive airbag distortions.

Check the material constants. Triangular elements should have less distortion problems than quadrilaterals. Overlapped elements at time zero can cause locking to occur in the contact leading to excessive distortions. The considerable energy input to the bag will create numerical noise and some damping is recommended to avoid problems.

3. The dummy passes through the airbag.

A most likely problem is that the contacts are improperly defined. Another possibility is that the models were developed in an incompatible unit system. The extra check for penetration flag if set to 1 on the contact control cards (Column 50 in Section 31) may sometimes cause nodes to be prematurely released due to the softness of the penalties. In this case the flag should be turned off.

4. The OSP fails to converge.

This may occur when excessively large forces are passed to the OSP. First, check that unit systems are consistent and then look for improperly defined contacts in the LS-DYNA input.

5. Time step approaches zero.

This is almost always in the airbag. If material 1 or 22 is being used, then switch to material 34 which is less time step size sensitive and use the fully integrated membrane element. Increasing the damping in the control volume usually helps considerably. Also, check for “cuts” in the airbag where nodes are not merged. These can allow elements to deform freely and cut the time step to zero.

APPENDIX H: Interactive Graphics Commands

Only the first four or less characterers of command are significant. These commands are available in the interactive phase of LS-DYNA. The interactive graphics is available by using the “SW5.” command after invoking the Ctrl-C interupt. The MENU command brings up a push button menu.

ANIMATE	Animate saved sequence, stop with switch 1.
BACK	Return to previous display size after zoom then list display attributes.
BGC	Change display background color RGB proportions BGC <red> <green> <blue>.
BIP	Select beam integration point for contour; BIP <#>.
CENTER	Center model, center on node, or center with mouse, i.e. center cent <value> or cent gin.
CL	Classification labels on display; class commercial_in_confidence
CMA	Color materials on limited color displays
COLOR	Set or unset shaded coloring of materials.
CONTOUR	View with colored contour lines; contour <component #> <list mat #>; see TAURUS manual.
COOR	Get node information with mouse.
COP	Hardcopy of display on the PC copy <laserj paintj tekcol coljet or epson>.
CR	Restores cutting plane to default position
CUT	Cut away model outside of zoom window; use mouse to set zoom window size.
CX	Rotate slice plane at zmin about x axis.
CY	Rotate slice plane at zmin about y axis.
CZ	Rotate slice plane at zmin about z axis.
DIF	Change diffused light level for material; DIF <mat #, -1 for all> <value>.

DISTANCE	Set distance of model from viewer; DIST <value in normalized model dimensions>.
DMATERIALS	Delete display of material in subsequent views; DMAT <ALL or list of numbers>.
DRAW	Display outside edges of model.
DSCALE	Scale current displacement from initial shape.
DYN	After using TAURUS command will reset display to read current DYNA3D state data.
ELPLT	Set or unset element numbering in subsequent views.
END	Delete display and return to execution.
ESCAPE	Escapes from menu pad mode.
EXECUTE	Return to execution and keep display active.
FCL	Fix or unfix current contour levels.
FOV	Set display field of view angle; FOV <value in degrees>.
FRINGE	View with colored contour fringes; fringe <component #> <list mat #>; see TAURUS manual.
GETFRAME	Display a saved frame; GETF <frame #>.
HARDWARE	Hardware mode; workstation hardware calls are used to draw, move and color model; repeat, command to reset to normal mode.
HELP	
HZB	Switch on or off hardware zbuffer for a subsequent view, draw or contour command; rotations and translations will be in hardware.
LIMIT	Set range of node numbers subsequent views; limit <first node #> <last node #>.
MAT	Re-enable display of deleted materials mat <all or list of numbers>.
MENU	Button menu pad mode.

MOTION	Motion of model through mouse movement or use of a dial box. The left button down enables translation in the plane, middle button rotation about axes in the plane; and with right button down in the out of plane axis; left and middle button down quit this mode.
MOV	Drag picked part to new position set with mouse.
NDPLT	Set or unset node numbering in subsequent views.
NOFRAME	Set and unset drawing of a frame around the picture.
PAUSE	Animation display pause in seconds
PHS2 or THISTORY	Time history plotting phase. Similar to LS-TAURUS.
PICK	Get element information with mouse.
POST	Enable or disenable postscript mode on the PC and eps file is written as picture is drawn; remove eofs and initgraphics for eps use.
QUIT	Same as execute.
RANGE	Set fix range for contour levels; range <minvalue> <maxvalue>.
RAX	Reflect model about xy plane; restore command will switch-off reflections.
RAY	Reflect model about yz plane; restore command will switch-off reflections.
RAZ	Reflect model about zx plane, restore command will switch-off reflections.
RESTORE	Restores model to original position also switches off element and node numbers, slice capper, reflections and cut model.
RETURN	Exit.
RGB	Change color red green blue element <mat #> <red> <green> <blue>.
RX	Rotate model about x axis.
RY	Rotate model about y axis.
RZ	Rotate model about z axis.

SAVE	Set or unset saving of display for animation.
SEQUENCE	Periodic plot during execution; SEQ <# of cycles> <commands> EXE.
SHR	Shrink element facets towards centoids in subsequent views, shrink <value>.
SIP	Select shell integration point for contour; SIP <#>.
SLICE	Slice model a z-minimum plane; slice <value in normalized model dimension> this feature is removed after using restore. Slice enables internal details for brick elements to be used to generate new polygons on the slice plane.
SNORMAL	Set or unset display of shell direction normals will to indicate topology order.
SPOT	Draw node numbers on model spot <first #> <last # for range>.
TAURUS	TAURUS database, TAU <state #>, or state <state #>, reads TAURUS file to extract previous state data.
TRIAD	Set or unset display of axis triad.
TSHELL	Set or unset shell element thickness simulation in subsequent views.
TV	Change display type.
TX	Translates model along x axis
TY	Translates model along y axis
TZ	Translates model along z axis
V	Display model using painters algorithm.
VECTOR v or d	View with vector arrows of velocity or displacement; <v> or <d>.
ZB	Switch on or off zbuffer algorithm for subsequent view; or draw commands.
ZIN	Zoom in using mouse to set display size and position.
ZMA	Set position of zmax plane; ZMAX <value in normalized model dimesions>.

ZMI	Set position of zmin plane; ZMIN <value in normalized model dimesions>.
ZOUT	Zoom out using mouse to set displays size expansion and position.

APPENDIX I: Interactive Material Model Driver

INTRODUCTION

The interactive material model driver in LS-DYNA allows calculation of the material constitutive response to a specified strain path. Since the constitutive model subroutines in LS-DYNA are directly called by this driver, the behavior of the constitutive model is precisely that which can be expected in actual applications. In the current implementation the constitutive subroutines for both shell elements and solid elements can be examined.

INPUT DEFINITION

The material model driver is invoked by setting the total number of beam, shell, and solid elements to zero in a standard DYNA3D input file. The number of material model definitions should be set to one, the number of load curves should to nine, and the termination time to the desired length of the driver run. The complete state dump interval is interpreted as the time step to be used in the material model driver run. Plotting information is saved for every step of a driver run and sufficient memory is allocated to save this information in core for the interactive plotting phase.

The input deck consists only of the TITLE card, the CONTROL cards, one MATERIAL DEFINITION, and NINE LOAD CURVES describing the strain path should be defined. These nine curves define the time history of the displacement gradient components shown in Table 1.

The velocity gradient matrix, L_{ij} , is approximated by taking the time derivative of the components in Table 1. If these components are considered to form a tensor S_{ij} , then

$$\mathrm{L}_{ij}(t) = \frac{S_{ij}(t) - S_{ij}(t_{k-1})}{(t - t_k)}$$

and the strain rate tensor is defined as

$$d_{ij} = \frac{L_{ij} + L_{ij}^t}{2}$$

and the spin tensor as

$$\omega_{ij} = \frac{L_{ij} - L_{ij}^t}{2}$$

Load Curve Number	Component Definition
1	$\frac{\partial u}{\partial x}$
2	$\frac{\partial v}{\partial y}$
3	$\frac{\partial w}{\partial z}$
4	$\frac{\partial u}{\partial y}$
5	$\frac{\partial v}{\partial x}$
6	$\frac{\partial u}{\partial z}$
7	$\frac{\partial w}{\partial x}$
8	$\frac{\partial v}{\partial z}$
9	$\frac{\partial w}{\partial y}$

Table 1. Load Curve Definitions

INTERACTIVE DRIVER COMMANDS

After reading the input file and completing the calculations, LS-DYNA gives a command prompt to the terminal. A summary of the available interactive commands is given below. An on-line help package is available by typing HELP.

ACCL	Scale all abscissa data by f. default is f=1.
ASET amin omax	Set min and max values on abscissa to amin and amax, respectively. If amin=amax=0, scaling is automatic.
CHGL n	Change label for component n. DYNA3D prompts for new label.
CONTINUE	Re-analyze material model.
CROSS c_1 c_2	Plot component c_1 versus c_2.
ECOMP	Display component numbers on the graphics display. 1 x-stress 2 y-stress 3 z-stress 4 xy-stress 5 yz-stress 6 zx-stress 7 effective plastic strain 8 pressure 9 von Mises (effective) stress 10 1st principal deviatoric stress 11 2nd principal deviatoric stress 12 3rd principal deviatoric stress 13 maximum shear stress 14 1st principal stress 15 2nd principal stress 16 3rd principal stress 17 ln (v/v0) 18 relative volume 19 v0/v - 1.0 20 1st history variable 21 2nd history variable Adding 100 or 400 to component numbers 1-16 yields strains and strain rates, respectively.
FILE name	Change pampers filename to name.

GRID	Graphics displays will be overlaid by a grid of orthogonal lines.
NOGRID	Graphics displays will not be overlaid by a grid of orthogonal liens
OSCL	Scale all ordinate data by f. default is f=1.
OSET omin omax	Set min and max values on ordinate to omin and omax, respectively. If omin=omax=0, scaling is automatic.
PRINT	Print plotted time history data in file "pampers." Only data plotted after this command is printed. File name can be changed with the "file" command.
QUIT, END, T	Exit the material model driver program.
RDLC m n r_1 z_1 ... r_n z_n	Redefine load curve m using n coordinate pairs (r_1,z_1) (r_2,z_2),...(r_n,z_n).
TIME c	Plot component c versus time.
TV n	Use terminal output device type n. LS-DYNA provides a list of available devices.

Presently, the material model drive is implemented for solid and shell element material models. The driver does not yet support material models for beam elements.

APPENDIX J: Commands for Two-Dimensional Rezoning

The rezoner in LS-DYNA contains many commands that can be broken down into the following categories:

- general,
- termination of interactive rezoning,
- redefinition of output intervals for data,
- graphics window controls,
- graphics window controls for x versus y plots,
- mesh display options,
- mesh modifications,
- boundary modifications,
- MAZE line definitions,
- calculation graphics display control parameters,
- calculation graphics display,
- cursor commands.

The use of the rezoner is quite simple. Commands for rezoning material number n can be invoked after the material is specified by the “M n” command. To view material n, the command “V” is available. The interior mesh can be smoothed with the “S” command and the boundary nodes can be adjusted after the “B” command is used to display the part side and boundary node numbers. Commands that are available for adjusting boundary nodes following the “B” command include:

ER, EZ, ES, VS, BD, ERS, EZS, ESS, VSS, BDS, SLN, SLNS

Rezoning is performed material by material. An example is shown.

Do not include the graphics display type number (see the “TV” command below) when setting up a command file for periodic noninteractive rezoning. No plotting is done when the rezoner is used in this mode.

REZONING COMMANDS BY FUNCTION

Interactive Real Time Graphics

SEQ n commands EXE	Every n time steps execute the graphics commands which follow. For example the line seq 100 g exe would cause the grid to be updated on the graphics display device every 100 cycles. The real time graphics can be terminated by using ctrl-c and typing "sw7.".

General

C	Comment - proceed to next line.
FRAME	Frame plots with a reference grid (default).
HELP	Enter HELP package and display all available commands. Description of each command is available in the HELP package.
HELP/commandname	Do not enter HELP package but print out the description on the terminal of the command following the slash.
LOGO	Put LLNL logo on all plots (default). Retyping this command removes the logo.
NOFRAME	Do not plot a reference grid.
PHP ans	Print help package - If answer equals 'y' the package is printed in the high speed printer file.
RESO n_x n_y	Set the x and y resolutions of plots to n_x and n_y, respectively. We default both n_x and n_y to 1024.
TV n	Use graphics output device type n. The types are installation dependent and a list will be provided after this command is invoked.
TR t	At time t, LS-DYNA will stop and enter interactive rezoning phase.

Termination of Interactive Rezoning

F	Terminate interactive phase, remap, continue in execution phase.
FR	Terminate interactive phase, remap, write restart dump, and call exit.
T or END	Terminate.

Redefinition of Output Intervals for Data

PLTI Δt	Reset the node and element data dump interval Δt.
PRTI Δt	Reset the node and element printout interval Δt.
TERM t	Reset the termination to t.

Graphics Window Controls

ESET n	Center picture at element n with a Δr by Δz window. This window is set until it is released by the unfix command or reset with another window.
FF	Encircle picture with reference grid with tickmarks. Default grid is plotted along bottom and left side of picture.
FIX	Set the display to its current window. This window is set until it is reset by the “GSET, “FSET”, or “SETF” commands or released by the “UNFIX” command.
FSET n Δr Δz	Center display at node n with a rectangular $\Delta r \times \Delta z$ window. This window is set until it is reset with or the “UNFIX” command is typed.
GSET r z Δl	Center display picture at point (r,z) with square window of width Δl. This window is set until it is reset or the “UNFIX” command is typed.
GRID	Overlay graphics displays with a grid of orthogonal lines.
NOGRID	Do not overlay graphics displays with a grid of orthogonal lines (default).
SETF r z Δr Δz	Center display at point (r,z) with a rectangular $\Delta r \times \Delta z$ window. This window is set until it is reset or the “UNFIX” command is typed.
UNFIX	Release current display window set by the “FIX”, “GSET”, “FSET” or “SETF” commands.
UZ a b Δl	Zoom in at point (a,b) with window Δl where a, b, and Δl are numbers between 0 and 1. The picture is assumed to lie in a unit square.
UZG	Cover currently displayed picture with a 10 by 10 square grid to aid in zooming with the unity zoom, “UZ”, command.

UZOU a b Δl	Zoom out at point (a,b) with window Δl where a, b, and Δl are numbers between 0 and 1. The current window is scaled by the factor $1/\Delta l$.The picture is assumed to lie in a unit square.
Z r z Δl	Zoom in at point (r,z) with window Δl.
ZOUT r z Δl	Zoom out at point (r,z) with window Δl. The window is enlarged by the ratio of the current window and Δl. The cursor may be used to zoom out via the cursor command DZOU and entering two points with the cursor to define the window. The ratio of the current window with the specified window determines the picture size reduction. An alternative cursor command, DZZO, may be used and only needs one point to be entered at the location where the reduction ($2\times$) is expected.

Graphics Window Controls for x versus y plots

The following commands apply to line plots, interface plots, etc.

ASCL f_a	Scale all abscissa data by f_a. The default is $f_a = 1$.
ASET amin amax	Set minimum and maximum values on abscissa to amin and amax, respectively. If amin=amax=0.0 (default) LS-DYNA determines the minimum and maximum values.
OSCL f_o	Scale all ordinate data by f_o. The default is $f_o = 1$.
OSET omin omax	Set minimum and maximum values on ordinate to omin and omax, respectively. If omin=omax=0.0 (default) LS-DYNA determines the minimum and maximum values.
SMOOTH n	Smooth a data curve by replacing each data point by the average of the 2n adjacent points. The default is n=0.

Mesh Display Options

ELPLT	Plot element numbers on mesh of material n.
FSOFF	Turn off the “FSON” command.
FSON	Plot only free surfaces and slideline interfaces with “O” command. (Must be used before “O” command.)
G	View mesh.

GO	View mesh right of centerline and outline left of centerline.
GS	View mesh and solid fill elements to identify materials by color.
M n	Material n is to be rezoned.
MNOFF	Do not plot material numbers with the “O”, “G”, and “GO” commands (default).
MNON	Plot material numbers with “O”, “G”, and “GO” commands.
NDPLT	Plot node numbers on mesh of material n.
O	Plot outlines of all material.
RPHA	Reflect mesh, contour, fringe, etc., plots about horizontal axis. Retyping “RPHA” turns this option off.
RPVA	Reflect mesh, contour, fringe, etc., plots about vertical axis. Retyping “RPVA” turns this option off.
TN r z Δl	Type node numbers and coordinates of all nodes within window $(r \pm \Delta l/2,\ z \pm \Delta l/2)$.
UG	Display undeformed mesh.
V	Display material n on graphics display. See command M.
VSF	Display material n on graphics display and solid fill elements.

Mesh Modifications

BACKUP	Restore mesh to its previous state. This command undoes the result of the last command.
BLEN s	Smooth option where s=0 and s=1 correspond to equipotential and isoparametric smoothing, respectively. By letting $0 \le s \le 1$ a combined blending is obtained.
CN m r z	Node m has new coordinate (r,z).
DEB n f_1 l_1 ... f_n l_n	Delete n element blocks consisting of element numbers f_1 to l_1, f_2 to l_2 ... , and f_o l_n inclusive. These elements will be inactive when the calculation resume.
DE e_1 e_2	Delete elements e_1 to e_2.

DMB n m_1 m_2 ... m_n	Delete n material blocks consisting of all elements with material numbers m_1, m_2,..., and m_n. These materials will be inactive when the calculations resume.
DM n m_1 m_2 ... m_n	Delete n materials including m_1, m_2,..., and m_n.
R	Restore original mesh.
S	Smooth mesh of material n. To smooth a subset of elements, a window can be set via the “GSET”, “FSET”, OR “SETF” commands. Only the elements lying within the window are smoothed.

Boundary Modifications

A	Display all slidelines. Slave sides are plotted as dashed lines.
B	Determine boundary nodes and sides of material n and display boundary with nodes and side numbers.
BD m n	Dekink boundary from boundary node m to boundary node n (counterclockwise).
BDS s	Dekink side s.
DSL n l_1 l_2...l_n	Delete n slidelines including slideline numbers l_1 l_2..., and l_n.
ER m n	Equal space in r-direction boundary nodes m to n (counterclockwise).
ERS s	Equal space in the r-direction boundary nodes on side s.
ES m n	Equal space along boundary, boundary nodes m to n (counterclockwise).
ESS s	Equal space along boundary, boundary nodes on side s.
EZ m n	Equal space in z-direction boundary nodes m to n (counterclockwise).
EZS s	Equal space in the z-direction boundary nodes on side s.
MC n	Check master nodes of slideline n and put any nodes that have penetrated through the slave surface back on the slave surface.
MD n	Dekink master side of slideline n. After using this command, the SC or MC command is sometimes advisable.

MN n	Display slideline n with master node numbers.
SC n	Check slave nodes of slideline n and put any nodes that have penetrated through the master surface back on the master surface.
SD n	Dekink slave side of slideline n; after using this command, the SC or MC command is sometimes advisable.
SLN m n	Equal space boundary nodes between nodes m to n on a straight line connecting node m to n.
SLNS n	Equal space boundary nodes along side n on a straight line connecting the corner nodes.
SN n	Display slideline n with slave node numbers.
VS m n r	Vary the spacing of boundary nodes m to n such that r is the ratio of the first segment length to the last segment length.
VSS s r	Vary the spacing of boundary nodes on side s such that r is the ratio of the first segment length to the last segment length.

MAZE Line Definitions

B	Determine boundary nodes and sides of material n and display boundary with nodes and side numbers. See command “M”.
LD n k l	Line defintion n for MAZE includes boundary nodes k to l
LDS n l	Line definition n for MAZE consists of side number l.
M n	Material n is active for the boundary command B.

Calculation Graphics Display Control Parameters

MOLP	Overlay the mesh on the contour, fringe, principal stress, and principal strain plots. Retyping “MOLP” turns this option off.
NLOC	Do not plot letters on contour lines.
NUMCON n	Plot n contour levels. The default is 9.

PLOC	Plot letters on contour lines to identify their levels (default).
RANGE r_1 r_2	Set the range of levels to be between r_1 and r_2 instead of in the range chosen automatically by LS-DYNA. To deactivate this command, type RANGE 0 0.

Calculation Graphics Display

CONTOUR c n m_1 m_2...m_n	Contour component number c on n materials including materials m_1, m_2, ..., m_n. If n is zero, only the outline of material m_1 with contours is plotted. Component numbers are given in Table 1.
FRINGE c n m_1 m_2...m_n	Fringe component number c on n materials including m_1, m_2,...,m_n. If n is zero, only the outline of material m_1 with contours is plotted. Component numbers are given in Table 1.
IFD n	Begin definition of interface n. If interface n has been previously defined, this command has the effect of destroying the old definition.
IFN l m	Include boundary nodes l to m (counterclockwise) in the interface definition. This command must follow the “B” command.
IFP c m	Plot component c of interface m. Component numbers are given in Table 2.
IFS m	Include side m in the interface definition. Side m is defined for material n by the “B” command.
IFVA r_c z_c	Plot the angular location of the interface based on the center point (r_c,z_c) along the abscissa. Positive angles are measured counterclockwise from the y axis.
IFVS	Plot the distance along the interface from the first interface node along the abscissa (default).
LINE c n m_1 m_2...m_n	Plot variation of component c along line defined with the “NLDF”, “PLDF”, “NSDF”, or the “NSSDF” commands given below. In determining variation, consider n materials including material number m_1, m_2,...m_n.
NCOL n	Number of colors in fringe plots is n. The default value for n is 6 which includes colors magenta, blue, cyan, green, yellow, and red. An alternative value for n is 5 which eliminates the minimum value magenta.

NLDF n n_1 n_2...n_3	Define line for "LINE" command using n nodes including node numbers n_1, n_2,...n_n. This line moves with the nodes.
NSDF m	Define line for "LINE" command as side m. Side m is defined for material n by the "B" command.
NSSDF l m	Define line for "LINE" command and that includes boundary nodes l to m (counterclockwise) in the interface definitions. This command must follow the "B" command.
PLDF n r_1 z_1...r_n z_n	Define line for "LINE" command using n coordinate pairs (r_1,z_1), (r_2,z_2), ...(r_n,z_n). This line is fixed in space.
PRIN c n m_1 m_2...m_n	Plot lines of principal stress and strain in the yz plane on n materials including materials m_1, m_2,...,m_n. If n is zero, only the outline of material m_1 is plotted. The lines are plotted in the principal stress and strain directions. Permissible component numbers in Table 1 include 0, 5, 6, 100, 105, 106,...,etc. Orthogonal lines of both maximum and minimum stress are plotted if components 0, 100, 200, etc. are specified.
PROFILE c n m_1 m_2...m_n	Plot component c versus element number for n materials including materials m_1, m_2,...,m_n. If n is Ø then component c is plotted for all elements. Component numbers are given in Table 1.
VECTOR c n m_1 m_2...m_n	Make a vector plot of component c on n materials including materials m_1, m_2,...,m_n. If n is zero, only the outline of material m_1 with vectors is plotted. Component c may be set to "D" and "V" for vector plots of displacement and velocity, respectively.

No.	Component	No.	Component
1	y	21*	ln (V⁄Vo) (volumetric strain)
2	z	22*	y-displacement
3	hoop	23*	z-displacement
4	yz	24*	maximum displacement
5	maximum principal	25*	y-velocity, y-heat flux
6	minimum principal	26*	z-velocity, y-heat flux
7	von Mises (Appendix A)	27*	maximum velocity, maximum heat flux
8	pressure or average strain	28	ij normal
9	maximum principal-minimum principal	29	jk normal
10	y minus hoop	30	kl normal
11	maximum shear	31	li normal
12	ij and kl normal (Appendix B)	32	ij shear
13	jk and li normal	33	jk shear
14	ij and kl shear	34	kl shear
15	jk and li shear	35	li shear
16	y-deviatoric	36*	relative volume V⁄Vo
17	z-deviatoric	37*	Vo⁄V-1
18	hoop-deviatoric	38*	bulk viscosity, Q
19*	effective plastic strain	39*	P + Q
20*	temperature/internal energy density	40*	density
41*-70*	element history variables		
71*	r-peak acceleration	76*	peak value of min in plane prin. stress
72*	z-peak acceleration	77*	peak value of maximum hoop stress
73*	r-peak velocity	78*	peak value of minimum hoopstress
74*	z-peak velocity	79*	peak value of pressure
75*	peak value of max. in plane prin. stress		

Table 1. Component numbers for element variables. By adding 100, 200 300, 400, 500 and 600 to the component numbers not followed by an asterick, component numbers for infinitesimal strains, lagrange strains, almansi strains, strain rates, extensions, and residual strain are obtained. Maximum and minimum principal stresses and strains are in the rz plane. The corresponding hoop quantities must be examined to determine the overall extremum. ij, jk, etc. normal components are normal to the ij, jk, etc side. The peak value database must be flagged on Control Card 4 in columns 6-10 or components 71-79 will not be available for plotting.

No.	Component
1	pressure
2	shear stress
3	normal force
4	tangential force
5	y-force
6	z-force

Table 2. Component numbers for interface variables. In axisymmetric geometries the force is per radian.

Cursor Commands

DBD a b	Use cursor to define points a and b on boundary. Dekink boundary starting at a, moving counterclockwise, and ending at b.
DCN a b	Use cursor to define points a and b. The node closest to point a will be moved to point b.
DCSN n a	Move nodal point n to point a defined by the cursor.
DCNM a b	Use cursor to define points a and b. The node at point a is given the coordinate at point b.
DER a b	Use cursor to define points a and b on boundary. Equal space nodes in r-direction along boundary starting at a, moving counterclockwise, and ending at b.
DES a b	Use cursor to define points a and b on boundary. Equal space nodes along boundary starting at a, moving counterclockwise, and ending at b.
DEZ a b	Use cursor to define points a and b on boundary. Equal space nodes in z-direction along boundary starting at a, moving counterclockwise, and ending at b.
DTE a b	Use cursor to define points a and b on the diagonal of a window. The element numbers and coordinates of elements lying within the window are typed on the terminal.
DTN a b	Use cursor to define points a and b on the diagonal of a window. The node numbers and coordinates of nodal points lying within the window are typed on the terminal.
DTNC a	Use cursor to define point a. The nodal point number and nodal coordinates of the node lying closest to point a will be printed.

DVS a b r	Use cursor to define points a and b on boundary. Variable space nodes along boundary starting at a, moving counter-wise, and ending at b. The ratio of the first segment length to the last segment length is give by r (via terminal).
DZ a b	Use cursor to define points a and b on the diagonal of a window for zooming.
DZOUT a b	Enter two points with the cursor to define the window. The ratio of the current window with the specified window determines the picture size reduction.
DZZ a	Use cursor to define point a and zoom in at this point. The new window is .15 as large as the previous window. The zoom factor can be reset by the crzf command for the .15 default.
DZZO a	Zoom out at point a by enlarging the picture two times.