

**LS-DYNA  
KEYWORD USER'S MANUAL**

**Nonlinear Dynamic Analysis  
of Structures**

**May 1999  
Version 950**

Copyright © 1992-1999

**LIVERMORE SOFTWARE  
TECHNOLOGY CORPORATION**

All Rights Reserved

**Mailing Address:**

**Livermore Software Technology Corporation  
2876 Waverley Way  
Livermore, California 94550-1740**

**Support Address:**

**Livermore Software Technology Corporation  
7374 Las Positas Road  
Livermore, California 94550**

**TEL: 925-449-2500**

**FAX: 925-449-2507**

**EMAIL: sales@lstc.com**

**Copyright © 1992-1999 by Livermore Software Technology Corporation  
All Rights Reserved**

---

# TABLE OF CONTENTS

## TABLE OF CONTENTS

<b>INTRODUCTION .....</b>	<b>I.1</b>
CHRONOLOGICAL HISTORY.....	I.1
DESCRIPTION OF KEYWORD INPUT.....	I.13
MATERIAL MODELS.....	I.25
SPATIAL DISCRETIZATION.....	I.27
CONTACT-IMPACT INTERFACES.....	I.31
INTERFACE DEFINITIONS FOR COMPONENT ANALYSIS.....	I.33
CAPACITY.....	I.34
CODE ORGANIZATION.....	I.35
SENSE SWITCH CONTROLS.....	I.36
PRECISION.....	I.37
EXECUTION SYNTAX.....	I.38
RESTART ANALYSIS.....	I.42
VDA/IGES DATABASES.....	I.44
MESH GENERATION.....	I.45
LS-POST POST-PROCESSING.....	I.46
EXECUTION SPEEDS .....	I.48
UNITS .....	I.50
GENERAL CARD FORMAT.....	I.51
<b>*AIRBAG.....</b>	<b>1.1</b>
* <i>AIRBAG_OPTION1_{OPTION2}</i> .....	1.1
* <i>AIRBAG_INTERACTION</i> .....	1.34
* <i>AIRBAG_REFERENCE_GEOMETRY_OPTION_OPTION</i> .....	1.36
<b>*ALE.....</b>	<b>2.1</b>
* <i>ALE_MULTI-MATERIAL_GROUP_OPTION</i> .....	2.2
* <i>ALE_SMOOTHING</i> .....	2.4
<b>*BOUNDARY.....</b>	<b>3.1</b>
* <i>BOUNDARY_CONVECTION_OPTION</i> .....	3.2
* <i>BOUNDARY_CYCLIC</i> .....	3.4
* <i>BOUNDARY_ELEMENT_METHOD_OPTION</i> .....	3.6
* <i>BOUNDARY_FLUX_OPTION</i> .....	3.16
* <i>BOUNDARY_NON_REFLECTING</i> .....	3.19
* <i>BOUNDARY_NON_REFLECTING_2D</i> .....	3.20

# TABLE OF CONTENTS

---

*BOUNDARY_PRESCRIBED_MOTION_OPTION .....	3.22
*BOUNDARY_PRESSURE_OUTFLOW_OPTION.....	3.26
*BOUNDARY_RADIATION_OPTION .....	3.28
*BOUNDARY_SLIDING_PLANE.....	3.30
*BOUNDARY_SPC_OPTION.....	3.31
*BOUNDARY_SYMMETRY_FAILURE .....	3.33
*BOUNDARY_TEMPERATURE_OPTION.....	3.34
*BOUNDARY_USA_SURFACE .....	3.35
<b>*COMPONENT.....</b>	<b>4.1</b>
*COMPONENT_GEBOD_OPTION .....	4.2
*COMPONENT_GEBOD_JOINT_OPTION .....	4.4
<b>*CONSTRAINED .....</b>	<b>5.1</b>
*CONSTRAINED_EXTRA_NODES_OPTION.....	5.2
*CONSTRAINED_GENERALIZED_WELD_OPTION.....	5.4
*CONSTRAINED_GLOBAL.....	5.17
*CONSTRAINED_JOINT_OPTION_{OPTION}.....	5.19
*CONSTRAINED_JOINT_STIFFNESS_OPTION .....	5.29
*CONSTRAINED_LAGRANGE_IN_SOLID.....	5.39
*CONSTRAINED_LINEAR.....	5.41
*CONSTRAINED_NODAL_RIGID_BODY_{OPTION}.....	5.45
*CONSTRAINED_NODE_SET .....	5.50
*CONSTRAINED_RIGID_BODIES.....	5.52
*CONSTRAINED_RIGID_BODY_STOPPERS.....	5.54
*CONSTRAINED_RIVET .....	5.57
*CONSTRAINED_SHELL_TO_SOLID .....	5.59
*CONSTRAINED_SPOTWELD_{OPTION} .....	5.61
*CONSTRAINED_TIE-BREAK.....	5.65
*CONSTRAINED_TIED_NODES_FAILURE .....	5.66
<b>*CONTACT.....</b>	<b>6.1</b>
*CONTACT_{OPTION1}_{OPTION2}_{OPTION3}_{OPTION4} .....	6.2
*CONTACT_ENTITY .....	6.33
*CONTACT_GEBOD_OPTION.....	6.42
*CONTACT_INTERIOR .....	6.45
*CONTACT_1D.....	6.47
*CONTACT_2D_OPTION1_{OPTION2} .....	6.48

# TABLE OF CONTENTS

---

<b>*CONTROL</b> .....	<b>7.1</b>
*CONTROL_ACCURACY.....	7.3
*CONTROL_ADAPSTEP .....	7.4
*CONTROL_ADAPTIVE.....	7.5
*CONTROL_ALE.....	7.9
*CONTROL_BULK_VISCOSITY.....	7.11
*CONTROL_CONTACT.....	7.12
*CONTROL_COUPLING.....	7.16
*CONTROL_CPU.....	7.18
*CONTROL_DYNAMIC_RELAXATION.....	7.19
*CONTROL_ENERGY .....	7.21
*CONTROL_EXPLOSIVE_SHADOW .....	7.22
*CONTROL_HOURLASS_{ <i>OPTION</i> }.....	7.23
*CONTROL_IMPLICIT_AUTO .....	7.25
*CONTROL_IMPLICIT_DYNAMICS.....	7.26
*CONTROL_IMPLICIT_GENERAL.....	7.28
*CONTROL_IMPLICIT_LINEAR .....	7.30
*CONTROL_IMPLICIT_NONLINEAR .....	7.32
*CONTROL_IMPLICIT_STABILIZATION.....	7.36
*CONTROL_OUTPUT .....	7.37
*CONTROL_PARALLEL .....	7.39
*CONTROL_RIGID .....	7.41
*CONTROL_SHELL.....	7.42
*CONTROL_SOLUTION.....	7.45
*CONTROL_SPH.....	7.46
*CONTROL_STRUCTURED_{ <i>OPTION</i> }.....	7.47
*CONTROL_SUBCYCLE.....	7.48
*CONTROL_TERMINATION.....	7.49
*CONTROL_THERMAL_NONLINEAR.....	7.50
*CONTROL_THERMAL_SOLVER.....	7.51
*CONTROL_THERMAL_TIMESTEP.....	7.53
*CONTROL_TIMESTEP .....	7.54
<b>*DAMPING</b> .....	<b>8.1</b>
*DAMPING_GLOBAL.....	8.1
*DAMPING_PART_MASS.....	8.3
*DAMPING_PART_STIFFNESS.....	8.5

# TABLE OF CONTENTS

---

<b>*DATABASE</b> .....	<b>9.1</b>
*DATABASE_OPTION.....	9.2
*DATABASE_BINARY_OPTION.....	9.5
*DATABASE_CROSS_SECTION_OPTION.....	9.7
*DATABASE_EXTENT_OPTION.....	9.11
*DATABASE_FORMAT.....	9.19
*DATABASE_HISTORY_OPTION.....	9.20
*DATABASE_NODAL_FORCE_GROUP.....	9.21
*DATABASE_SPRING_FORWARD.....	9.22
*DATABASE_SUPERPLASTIC_FORMING.....	9.23
*DATABASE_TRACER.....	9.24
<b>*DEFINE</b> .....	<b>10.1</b>
*DEFINE_BOX.....	10.2
*DEFINE_BOX_ADAPTIVE.....	10.3
*DEFINE_BOX_DRAWBEAD.....	10.5
*DEFINE_COORDINATE_NODES.....	10.6
*DEFINE_COORDINATE_SYSTEM.....	10.7
*DEFINE_COORDINATE_VECTOR.....	10.9
*DEFINE_CURVE.....	10.10
*DEFINE_CURVE_FEEDBACK.....	10.12
*DEFINE_CURVE_TRIM.....	10.15
*DEFINE_SD_ORIENTATION.....	10.18
*DEFINE_TABLE.....	10.20
*DEFINE_VECTOR.....	10.22
<b>*DEFORMABLE_TO_RIGID</b> .....	<b>11.1</b>
*DEFORMABLE_TO_RIGID.....	11.2
*DEFORMABLE_TO_RIGID_AUTOMATIC.....	11.3
*DEFORMABLE_TO_RIGID_INERTIA.....	11.7
<b>*ELEMENT</b> .....	<b>12.1</b>
*ELEMENT_BEAM_{OPTION}.....	12.2
*ELEMENT_DISCRETE.....	12.8
*ELEMENT_INERTIA.....	12.9
*ELEMENT_MASS.....	12.11
*ELEMENT_SEATBELT.....	12.12
*ELEMENT_SEATBELT_ACCELEROMETER.....	12.13

# TABLE OF CONTENTS

---

*ELEMENT_SEATBELT_PRETENSIONER.....	12.14
*ELEMENT_SEATBELT_RETRACTOR.....	12.16
*ELEMENT_SEATBELT_SENSOR.....	12.22
*ELEMENT_SEATBELT_SLIPRING.....	12.26
*ELEMENT_SHELL_{ <i>OPTION</i> }.....	12.28
*ELEMENT_SOLID_{ <i>OPTION</i> }.....	12.32
*ELEMENT_SPH.....	12.37
*ELEMENT_TRIM.....	12.38
*ELEMENT_TSHELL.....	12.39
<b>*EOS.....</b>	<b>13.1</b>
*EOS_LINEAR_POLYNOMIAL.....	13.2
*EOS_JWL.....	13.4
*EOS_SACK_TUESDAY.....	13.5
*EOS_GRUNEISEN.....	13.6
*EOS_RATIO_OF_POLYNOMIALS.....	13.8
*EOS_LINEAR_POLYNOMIAL_WITH_ENERGY_LEAK.....	13.12
*EOS_IGNITION_AND_GROWTH_OF_REACTION_IN_HE.....	13.14
*EOS_TABULATED_COMPACTIION.....	13.18
*EOS_TABULATED.....	13.20
*EOS_PROPELLANT_DEFLAGRATION.....	13.22
*EOS_TENSOR_PORE_COLLAPSE.....	13.27
*EOS_JWLB.....	13.30
<b>*HOURLASS.....</b>	<b>14.1</b>
*HOURLASS.....	14.1
<b>*INCLUDE.....</b>	<b>15.1</b>
*INCLUDE.....	15.1
<b>*INITIAL.....</b>	<b>16.1</b>
*INITIAL_DETONATION.....	16.2
*INITIAL_FOAM_REFERENCE_GEOMETRY.....	16.5
*INITIAL_MOMENTUM.....	16.6
*INITIAL_STRAIN_SHELL.....	16.7
*INITIAL_STRESS_BEAM.....	16.8
*INITIAL_STRESS_SHELL.....	16.10
*INITIAL_STRESS_SOLID.....	16.12
*INITIAL_TEMPERATURE_ <i>OPTION</i> .....	16.14

# TABLE OF CONTENTS

---

*INITIAL_VEHICLE_KINEMATICS.....	16.15
*INITIAL_VELOCITY .....	16.18
*INITIAL_VELOCITY_NODE.....	16.20
*INITIAL_VELOCITY_GENERATION.....	16.21
*INITIAL_VOID_OPTION.....	16.23
<b>*INTEGRATION .....</b>	<b>17.1</b>
*INTEGRATION_BEAM.....	17.1
*INTEGRATION_SHELL.....	17.6
<b>*INTERFACE.....</b>	<b>18.1</b>
*INTERFACE_COMPONENT_OPTION.....	18.1
*INTERFACE_LINKING_DISCRETE_NODE_OPTION.....	18.2
*INTERFACE_LINKING_SEGMENT .....	18.3
*INTERFACE_LINKING_EDGE .....	18.4
*INTERFACE_JOY.....	18.5
*INTERFACE_SPRINGBACK_OPTION1_OPTION2 .....	18.6
<b>*LOAD.....</b>	<b>19.1</b>
*LOAD_BEAM_OPTION .....	19.2
*LOAD_BODY_OPTION.....	19.4
*LOAD_BODY_GENERALIZED.....	19.8
*LOAD_BRODE .....	19.10
*LOAD_DENSITY_DEPTH.....	19.12
*LOAD_HEAT_GENERATION_OPTION .....	19.14
*LOAD_MASK.....	19.15
*LOAD_NODE_OPTION.....	19.17
*LOAD_RIGID_BODY.....	19.20
*LOAD_SEGMENT.....	19.22
*LOAD_SEGMENT_SET .....	19.24
*LOAD_SHELL_OPTION .....	19.26
*LOAD_SSA.....	19.28
*LOAD_SUPERPLASTIC_FORMING.....	19.31
*LOAD_THERMAL_OPTION.....	19.33
*LOAD_THERMAL_CONSTANT.....	19.34
*LOAD_THERMAL_CONSTANT_NODE .....	19.35
*LOAD_THERMAL_LOAD_CURVE.....	19.36
*LOAD_THERMAL_TOPAZ.....	19.37



# TABLE OF CONTENTS

---

*LOAD_THERMAL_VARIABLE.....	19.38
*LOAD_THERMAL_VARIABLE_NODE.....	19.40
<b>*MAT.....</b>	<b>20.1</b>
*MAT_ADD_EROSION.....	20.10
*MAT_ELASTIC_{ <i>OPTION</i> }.....	20.12
*MAT_ <i>OPTION</i> TROPIC_ELASTIC.....	20.15
*MAT_PLASTIC_KINEMATIC.....	20.21
*MAT_ELASTIC_PLASTIC_THERMAL.....	20.24
*MAT_SOIL_AND_FOAM.....	20.27
*MAT_VISCOELASTIC.....	20.31
*MAT_BLATZ-KO_RUBBER.....	20.32
*MAT_HIGH_EXPLOSIVE_BURN.....	20.33
*MAT_NULL.....	20.36
*MAT_ELASTIC_PLASTIC_HYDRO_{ <i>OPTION</i> }.....	20.38
*MAT_STEINBERG.....	20.42
*MAT_STEINBERG_LUND.....	20.46
*MAT_ISOTROPIC_ELASTIC_PLASTIC.....	20.49
*MAT_ISOTROPIC_ELASTIC_FAILURE.....	20.50
*MAT_SOIL_AND_FOAM_FAILURE.....	20.52
*MAT_JOHNSON_COOK.....	20.53
*MAT_PSEUDO_TENSOR.....	20.57
*MAT_ORIENTED_CRACK.....	20.65
*MAT_POWER_LAW_PLASTICITY.....	20.67
*MAT_STRAIN_RATE_DEPENDENT_PLASTICITY.....	20.69
*MAT_RIGID.....	20.72
*MAT_ORTHOTROPIC_THERMAL.....	20.76
*MAT_COMPOSITE_DAMAGE.....	20.80
*MAT_TEMPERATURE_DEPENDENT_ORTHOTROPIC.....	20.84
*MAT_PIECEWISE_LINEAR_PLASTICITY.....	20.89
*MAT_GEOLOGIC_CAP_MODEL.....	20.93
*MAT_HONEYCOMB.....	20.99
*MAT_MOONEY-RIVLIN_RUBBER.....	20.106
*MAT_RESULTANT_PLASTICITY.....	20.109
*MAT_FORCE_LIMITED.....	20.110
*MAT_CLOSED_FORM_SHELL_PLASTICITY.....	20.116
*MAT_FRAZER_NASH_RUBBER_MODEL.....	20.117

# TABLE OF CONTENTS

---

*MAT_LAMINATED_GLASS .....	20.120
*MAT_BARLAT_ANISOTROPIC_PLASTICITY.....	20.122
*MAT_BARLAT_YLD96 .....	20.125
*MAT_FABRIC.....	20.129
*MAT_PLASTIC_GREEN-NAGHDI_RATE.....	20.133
*MAT_3-PARAMETER_BARLAT .....	20.134
*MAT_TRANSVERSELY_ANISOTROPIC_ELASTIC_PLASTIC.....	20.138
*MAT_BLATZ-KO_FOAM.....	20.141
*MAT_FLD_TRANSVERSELY_ANISOTROPIC .....	20.143
*MAT_NONLINEAR_ORTHOTROPIC.....	20.145
*MAT_USER_DEFINED_MATERIAL_MODELS.....	20.149
*MAT_BAMMAN.....	20.152
*MAT_BAMMAN_DAMAGE.....	20.157
*MAT_CLOSED_CELL_FOAM.....	20.160
*MAT_ENHANCED_COMPOSITE_DAMAGE .....	20.163
*MAT_LOW_DENSITY_FOAM.....	20.170
*MAT_LAMINATED_COMPOSITE_FABRIC .....	20.174
*MAT_COMPOSITE_FAILURE_OPTION_MODEL .....	20.180
*MAT_ELASTIC_WITH_VISCOSITY .....	20.184
*MAT_KELVIN-MAXWELL_VISCOELASTIC.....	20.187
*MAT_VISCOUS_FOAM .....	20.189
*MAT_CRUSHABLE_FOAM.....	20.191
*MAT_RATE_SENSITIVE_POWERLAW_PLASTICITY.....	20.193
*MAT_MODIFIED_ZERILLI_ARMSTRONG .....	20.195
*MAT_LINEAR_ELASTIC_DISCRETE_BEAM.....	20.198
*MAT_NONLINEAR_ELASTIC_DISCRETE_BEAM.....	20.200
*MAT_NONLINEAR_PLASTIC_DISCRETE_BEAM.....	20.203
*MAT_SID_DAMPER_DISCRETE_BEAM .....	20.208
*MAT_HYDRAULIC_GAS_DAMPER_DISCRETE_BEAM.....	20.212
*MAT_CABLE_DISCRETE_BEAM.....	20.214
*MAT_CONCRETE_DAMAGE .....	20.216
*MAT_LOW_DENSITY_VISCOUS_FOAM.....	20.220
*MAT_BILKHU/DUBOIS_FOAM.....	20.224
*MAT_GENERAL_VISCOELASTIC .....	20.226
*MAT_HYPERELASTIC_RUBBER.....	20.230
*MAT_OGDEN_RUBBER.....	20.234

# TABLE OF CONTENTS

---

*MAT_SOIL_CONCRETE.....	20.238
*MAT_HYSTERETIC_SOIL .....	20.242
*MAT_RAMBERG_OSGOOD.....	20.245
*MAT_PLASTICITY_WITH_DAMAGE.....	20.247
*MAT_FU_CHANG_FOAM.....	20.252
*MAT_ORTHOTROPIC_VISCOELASTIC.....	20.259
*MAT_CELLULAR_RUBBER.....	20.262
*MAT_MTS.....	20.267
*MAT_PLASTIC_POLYMER .....	20.272
*MAT_ACOUSTIC.....	20.275
*MAT_BRITTLE_DAMAGE .....	20.277
*MAT_SPOTWELD.....	20.280
*MAT_ANISOTROPIC_VISCOPLASTIC .....	20.283
*MAT_DAMAGE_1 .....	20.288
*MAT_DAMAGE_2.....	20.293
*MAT_MODIFIED_PIECEWISE_LINEAR_PLASTICITY .....	20.297
*MAT_PLASTICITY_COMPRESSION_TENSION.....	20.300
*MAT_MODIFIED_HONEYCOMB.....	20.302
*MAT_ARRUDA_BOYCE_RUBBER .....	20.308
*MAT_HEART_TISSUE.....	20.311
*MAT_LUNG_TISSUE.....	20.314
*MAT_SPECIAL_ORTHOTROPIC.....	20.317
*MAT_SPRING_ELASTIC .....	20.321
*MAT_DAMPER_VISCOUS.....	20.322
*MAT_SPRING_ELASTOPLASTIC.....	20.323
*MAT_SPRING_NONLINEAR_ELASTIC.....	20.324
*MAT_DAMPER_NONLINEAR_VISCOUS.....	20.325
*MAT_SPRING_GENERAL_NONLINEAR.....	20.326
*MAT_SPRING_MAXWELL .....	20.328
*MAT_SPRING_INELASTIC.....	20.330
*MAT_SPRING_MUSCLE .....	20.331
*MAT_SEATBELT .....	20.336
*MAT_THERMAL_OPTION.....	20.338
*MAT_THERMAL_ISOTROPIC.....	20.339
*MAT_THERMAL_ORTHOTROPIC.....	20.340
*MAT_THERMAL_ISOTROPIC_TD.....	20.342

# TABLE OF CONTENTS

---

*MAT_THERMAL_ORTHOTROPIC_TD.....	20.344
*MAT_THERMAL_ISOTROPIC_PHASE_CHANGE.....	20.347
*MAT_THERMAL_ISOTROPIC_TD_LC.....	20.350
<b>NODE.....</b>	<b>21.1</b>
*NODE.....	21.1
<b>*PART.....</b>	<b>22.1</b>
*PART_{ <i>OPTION1</i> }_{ <i>OPTION2</i> }_{ <i>OPTION3</i> }.....	22.2
*PART_MOVE.....	22.9
<b>*RIGIDWALL.....</b>	<b>23.1</b>
*RIGIDWALL_GEOMETRIC_ <i>OPTION</i> _{ <i>OPTION</i> }.....	23.2
*RIGIDWALL_PLANAR_{ <i>OPTION</i> }_{ <i>OPTION</i> }_{ <i>OPTION</i> }.....	23.12
<b>*SECTION.....</b>	<b>24.1</b>
*SECTION_BEAM.....	24.2
*SECTION_DISCRETE.....	24.7
*SECTION_SEATBELT.....	24.10
*SECTION_SHELL_{ <i>OPTION</i> }.....	24.11
*SECTION_SOLID_{ <i>OPTION</i> }.....	24.16
*SECTION_SPH.....	24.19
*SECTION_TSHELL.....	24.20
<b>*SET.....</b>	<b>25.1</b>
*SET_BEAM_ <i>OPTION</i> .....	25.2
*SET_DISCRETE_{ <i>OPTION</i> }.....	25.4
*SET_NODE_ <i>OPTION</i> .....	25.6
*SET_PART_ <i>OPTION</i> .....	25.9
*SET_SEGMENT.....	25.12
*SET_SHELL_ <i>OPTION</i> .....	25.14
*SET_SOLID_{ <i>OPTION</i> }.....	25.17
*SET_TSHELL_{ <i>OPTION</i> }.....	25.19
<b>*TERMINATION.....</b>	<b>26.1</b>
*TERMINATION_NODE.....	26.2
*TERMINATION_BODY.....	26.3
*TERMINATION_CONTACT.....	26.4

# TABLE OF CONTENTS

---

<b>*TITLE .....</b>	<b>27.1</b>
*TITLE.....	27.1
<b>*TRANSLATE.....</b>	<b>28.1</b>
*TRANSLATE_ANSYS_OPTION.....	28.1
*TRANSLATE_IDEAS_{OPTION}.....	28.3
*TRANSLATE_NASTRAN.....	28.5
<b>*USER.....</b>	<b>29.1</b>
*USER_INTERFACE_OPTION .....	29.1
*USER_LOADING.....	29.3
<b>RESTART INPUT DATA.....</b>	<b>30.1</b>
*CHANGE_OPTION .....	30.3
*CONTROL_DYNAMIC_RELAXATION.....	30.17
*CONTROL_TERMINATION.....	30.19
*CONTROL_TIMESTEP .....	30.20
*DAMPING_GLOBAL.....	30.21
*DATABASE_OPTION.....	30.22
*DATABASE_BINARY_OPTION.....	30.24
*DELETE_OPTION.....	30.25
*INTERFACE_SPRINGBACK.....	30.27
*RIGID_DEFORMABLE_OPTION.....	30.29
*STRESS_INITIALIZATION_{OPTION}.....	30.32
*TERMINATION_OPTION .....	30.35
*TITLE.....	30.37
<b>REFERENCES .....</b>	<b>31.1</b>
<b>APPENDIX A</b>	
USER DEFINED MATERIALS.....	A.1
<b>APPENDIX B</b>	
USER DEFINED AIRBAG SENSOR .....	B.1
<b>APPENDIX C</b>	
USER DEFINED SOLUTION CONTROL.....	C.1
<b>APPENDIX D</b>	
USER DEFINED INTERFACE CONTROL.....	D.1

# TABLE OF CONTENTS

---

## APPENDIX E

USER DEFINED INTERFACE FRICTION .....	E.1
---------------------------------------	-----

## APPENDIX F

OCCUPANT SIMULATION INCLUDING THE COUPLING TO PROGRAMS CAL3D AND MADYMO .....	F.1
INTRODUCTION .....	F.1
THE LS-DYNA/OCCUPANT SIMULATION PROGRAM LINK .....	F.1
DUMMY MODELING .....	F.4
AIRBAG MODELING .....	F.4
KNEE BOLSTER .....	F.6
COMMON ERRORS .....	F.6

## APPENDIX G

INTERACTIVE GRAPHICS COMMANDS .....	G.1
-------------------------------------	-----

## APPENDIX H

INTERACTIVE MATERIAL MODEL DRIVER .....	H.1
INTRODUCTION .....	H.1
INPUT DEFINITION .....	H.1
INTERACTIVE DRIVER COMMANDS .....	H.3

## APPENDIX I

VDA DATABASE .....	I.1
--------------------	-----

## APPENDIX J

COMMANDS FOR TWO-DIMENSIONAL REZONING .....	J.1
REZONING COMMANDS BY FUNCTION .....	J.2

## APPENDIX K

RIGID BODY DUMMIES .....	K.1
--------------------------	-----

## APPENDIX L

LS-DYNA MPP USER GUIDE .....	L.1
------------------------------	-----

## APPENDIX M

IMPLICIT SOLVER .....	M.1
-----------------------	-----

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

### INTRODUCTION

#### CHRONOLOGICAL HISTORY

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

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

The 1981 version [Hallquist 1981a] evolved from the 1979 version. Nine additional material models were added to allow a much broader range of problems to be modeled including explosive-structure and soil-structure interactions. Body force loads were implemented for angular velocities and base accelerations. A link was also established from the 3D Eulerian code JOY [Couch, et. al.,

# INTRODUCTION

---

1983] for studying the structural response to impacts by penetrating projectiles. An option was provided for storing element data on disk thereby doubling the capacity of DYNA3D.

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

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

In the 1987 version of DYNA3D [Hallquist and Benson 1987] metal forming simulations and composite analysis became a reality. This version included shell thickness changes, the Belytschko-Tsay shell element [Belytschko and Tsay, 1981], and dynamic relaxation. Also included were non-reflecting boundaries, user specified integration rules for shell and beam elements, a layered composite damage model, and single point constraints.

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

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



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

Capabilities added in 1989-1990:

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

# INTRODUCTION

---

- user defined subroutines,
- warpage checks on shell elements,
- thickness consideration in all contact types,
- automatic orientation of contact segments,
- sliding interface energy dissipation calculations,
- nodal force and energy database for applied boundary conditions,
- defined stonewall velocity with input energy calculations,

Capabilities added in 1991-1992:

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

- seat belt capability.

Capabilities added in 1993-1994:

- Arbitrary Lagrangian Eulerian brick elements,
- Belytschko-Wong-Chiang quadrilateral shell element,
- Warping stiffness in the Belytschko-Tsay shell element,
- Fast Hughes-Liu shell element,
- Fully integrated thick shell element,
- Discrete 3D beam element,
- Generalized dampers,
- Cable modeling,
- Airbag reference geometry,
- Multiple jet model,
- Generalized joint stiffnesses,
- Enhanced rigid body to rigid body contact,
- Orthotropic rigid walls,
- Time zero mass scaling,
- Coupling with USA (Underwater Shock Analysis),
- Layered spot welds with failure based on resultants or plastic strain,
- Fillet welds with failure,
- Butt welds with failure,
- Automatic eroding contact,
- Edge-to-edge contact,
- Automatic mesh generation with contact entities,
- Drawbead modeling,
- Shells constrained inside brick elements,
- NIKE3D coupling for springback,
- Barlat's anisotropic plasticity,
- Superplastic forming option,
- Rigid body stoppers,
- Keyword input,
- Adaptivity,
- First MPP (Massively Parallel) version with limited capabilities.
- Built in least squares fit for rubber model constitutive constants,
- Large hysteresis in hyperelastic foam,
- Bilhku/Dubois foam model,

# INTRODUCTION

---

- Generalized rubber model,

Capabilities added in 1995:

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

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

Capabilities added in 1996-1997 in Version 940:

- Part/Material ID's may be specified with 8 digits.
- Rigid body motion can be prescribed in a local system fixed to the rigid body.
- Nonlinear least squares fit available for the Ogden rubber model.
- Least squares fit to the relaxation curves for the viscoelasticity in rubber.
- Fu-Chang rate sensitive foam.
- 6 term Prony series expansion for rate effects in model 57-now 73
- Viscoelastic material model 76 implemented for shell elements.
- Mechanical threshold stress (MTS) plasticity model for rate effects.
- Thermoelastic-plastic material model for Hughes-Liu beam element.
- Ramberg-Osgood soil model
- Invariant local coordinate systems for shell elements are optional.
- Second order accurate stress updates.
- Four noded, linear, tetrahedron element.
- Co-rotational solid element for foam that can invert without stability problems.
- Improved speed in rigid body to rigid body contacts.
- Improved searching for the a\_3, a\_5 and a10 contact types.
- Invariant results on shared memory parallel machines with the a\_n contact types.
- Thickness offsets in type 8 and 9 tie break contact algorithms.
- Bucket sort frequency can be controlled by a load curve for airbag applications.

# INTRODUCTION

---

- In automatic contact each part ID in the definition may have unique:
  - Static coefficient of friction
  - Dynamic coefficient of friction
  - Exponential decay coefficient
  - Viscous friction coefficient
  - Optional contact thickness
  - Optional thickness scale factor
  - Local penalty scale factor
- Automatic beam-to-beam, shell edge-to-beam, shell edge-to-shell edge and single surface contact algorithm.
- Release criteria may be a multiple of the shell thickness in types a\_3, a\_5, a10, 13, and 26 contact.
- Force transducers to obtain reaction forces in automatic contact definitions. Defined manually via segments, or automatically via part ID's.
- Searching depth can be defined as a function of time.
- Bucket sort frequency can be defined as a function of time.
- Interior contact for solid (foam) elements to prevent "negative volumes."
- Locking joint
- Temperature dependent heat capacity added to Wang-Nefske inflator models.
- Wang Hybrid inflator model [Wang, 1996] with jetting options and bag-to-bag venting.
- Aspiration included in Wang's hybrid model [Nusholtz, Wang, Wylie, 1996].
- Extended Wang's hybrid inflator with a quadratic temperature variation for heat capacities [Nusholtz, 1996].
- Fabric porosity added as part of the airbag constitutive model .
- Blockage of vent holes and fabric in contact with structure or itself considered in venting with leakage of gas.
- Option to delay airbag liner with using the reference geometry until the reference area is reached.
- Birth time for the reference geometry.
- Multi-material Euler/ALE fluids,
  - 2nd order accurate formulations.
  - Automatic coupling to shell, brick, or beam elements
  - Coupling using LS-DYNA contact options.
  - Element with fluid + void and void material
  - Element with multi-materials and pressure equilibrium
- Nodal inertia tensors.

- 2D plane stress, plane strain, rigid, and axisymmetric elements
- 2D plane strain shell element
- 2D axisymmetric shell element.
- Full contact support in 2D, tied, sliding only, penalty and constraint techniques.
- Most material types supported for 2D elements.
- Interactive remeshing and graphics options available for 2D.
- Subsystem definitions for energy and momentum output.
- Boundary element method for incompressible fluid dynamics and fluid-structure interaction problems.

Capabilities added during 1997-1998 in Version 950:

- Adaptive refinement can be based on tooling curvature with FORMING contact.
- The display of drawbeads is now possible since the drawbead data is output into the D3PLOT database.
- An adaptive box option, \*DEFINE\_BOX\_ADAPTIVE, allows control over the refinement level and location of elements to be adapted.
- A root identification file, ADAPT.RID, gives the parent element ID for adapted elements.
- Draw bead box option, \*DEFINE\_BOX\_DRAWBEAD, simplifies drawbead input.
- The new control option, CONTROL\_IMPLICIT, activates an implicit solution scheme.
- 2D Arbitrary-Lagrangian-Eulerian elements are available.
- 2D automatic contact is defined by listing part ID's.
- 2D r-adaptivity for plane strain and axisymmetric forging simulations is available.
- 2D automatic non-interactive rezoning as in LS-DYNA2D.
- 2D plane strain and axisymmetric element with 2x2 selective-reduced integration are implemented.
- Implicit 2D solid and plane strain elements are available.
- Implicit 2D contact is available.
- The new keyword, \*DELETE\_CONTACT\_2DAUTO, allows the deletion of 2D automatic contact definitions.
- The keyword, \*LOAD\_BEAM is added for pressure boundary conditions on 2D elements.
- A viscoplastic strain rate option is available for materials:

# INTRODUCTION

---

\*MAT\_PLASTIC\_KINEMATIC  
\*MAT\_JOHNSON\_COOK  
\*MAT\_POWER\_LAW\_PLASTICITY  
\*MAT\_STRAIN\_RATE\_DEPENDENT\_PLASTICITY  
\*MAT\_PIECEWISE\_LINEAR\_PLASTICITY  
\*MAT\_RATE\_SENSITIVE\_POWERLAW\_PLASTICITY  
\*MAT\_ZERILLI-ARMSTRONG  
\*MAT\_PLASTICITY\_WITH\_DAMAGE  
\*MAT\_PLASTICITY\_COMPRESSION\_TENSION

- Material model, \*MAT\_PLASTICITY\_WITH\_DAMAGE, has a piecewise linear damage curve given by a load curve ID.
- The Arruda-Boyce hyper-viscoelastic rubber model is available, see \*MAT\_ARRUDA\_BOYCE.
- Transverse-anisotropic-viscoelastic material for heart tissue, see \*MAT\_HEART\_ISSUE.
- Lung hyper-viscoelastic material, see \*MAT\_LUNG\_TISSUE.
- Compression/tension plasticity model, see \*MAT\_PLASTICITY\_COMPRESSION\_TENSION.
- The Lund strain rate model, \*MAT\_STEINBERG\_LUND, is added to Steinberg-Guinan plasticity model.
- Rate sensitive foam model, \*MAT\_FU\_CHANG\_FOAM, has been extended to include engineering strain rates, etc.
- Model, \*MAT\_MODIFIED\_PIECEWISE\_LINEAR\_PLASTICITY, is added for modeling the failure of aluminum.
- Material model, \*MAT\_SPECIAL\_ORTHOTROPIC, added for television shadow mask problems.
- Erosion strain is implemented for material type, \*MAT\_BAMMAN\_DAMAGE.
- The equation of state, \*EOS\_JWL, is available for modeling the expansion of explosive gases.
- The reference geometry option is extended for foam and rubber materials and can be used for stress initialization, see \*INITIAL\_FOAM\_REFERENCE\_GEOMETRY.
- A vehicle positioning option is available for setting the initial orientation and velocities, see \*INITIAL\_VEHICLE\_KINEMATICS.
- A boundary element method is available for incompressible fluid dynamics problems.
- The thermal materials work with instantaneous coefficients of thermal expansion:
  - \*MAT\_ELASTIC\_PLASTIC\_THERMAL
  - \*MAT\_ORTHOTROPIC\_THERMAL
  - \*MAT\_TEMPERATURE\_DEPENDENT\_ORTHOTROPIC
  - \*MAT\_ELASTIC\_WITH\_VISCOSITY.



- Airbag interaction flow rate versus pressure differences.
- Contact segment search option, [bricks first optional]
- A through thickness Gauss integration rule with 1-10 points is available for shell elements. Previously, 5 were available.
- Shell element formulations can be changed in a full deck restart.
- The tied interface which is based on constraint equations, TIED\_SURFACE\_TO\_SURFACE, can now fail if \_FAILURE, is appended.
- A general failure criteria for solid elements is independent of the material type, see \*MAT\_ADD\_EROSION
- Load curve control can be based on thinning and a flow limit diagram, see \*DEFINE\_CURVE\_FEEDBACK.
- An option to filter the spotweld resultant forces prior to checking for failure has been added the the option, \*CONSTRAINED\_SPOTWELD, by appending, \_FILTERED\_FORCE, to the keyword.
- Bulk viscosity is available for shell types 1, 2, 10, and 16.
- When defining the local coordinate system for the rigid body inertia tensor a local coordinate system ID can be used. This simplifies dummy positioning.
- Prescribing displacements, velocities, and accelerations is now possible for rigid body nodes.
- One way flow is optional for segmented airbag interactions.
- Pressure time history input for airbag type, LINEAR\_FLUID, can be used.
- An option is available to independently scale system damping by part ID in each of the global directions.
- An option is available to independently scale global system damping in each of the global directions.
- Added option to constrain global DOF along lines parallel with the global axes. The keyword is \*CONSTRAINED\_GLOBAL. This option is useful for adaptive remeshing.
- Beam end code releases are available, see \*ELEMENT\_BEAM.
- An initial force can be directly defined for the cable material, \*MAT\_CABLE\_DISCRETE\_BEAM. The specification of slack is not required if this option is used.
- Airbag pop pressure can be activated by accelerometers.
- Termination may now be controlled by contact, via \*TERMINATION\_CONTACT.

# INTRODUCTION

---

- Modified shell elements types 8, 10 and the warping stiffness option in the Belytschko-Tsay shell to ensure orthogonality with rigid body motions in the event that the shell is badly warped. This is optional in the Belytschko-Tsay shell and the type 10 shell.
- A one point quadrature brick element with an exact hourglass stiffness matrix has been implemented for implicit and explicit calculations.
- Automatic file length determination for D3PLOT binary database is now implemented. This insures that at least a single state is contained in each D3PLOT file and eliminates the problem with the states being split between files.
- The dump files, which can be very large, can be placed in another directory by specifying *d=/home/user /test/d3dump* on the execution line.
- A print flag controls the output of data into the MATSUM and RBDOUT files by part ID's. The option, PRINT, has been added as an option to the \*PART keyword.
- Flag has been added to delete material data from the D3THDT file. See \*DATABASE\_EXTENT\_BINARY and column 25 of the 19th control card in the structured input.
- After dynamic relaxation completes, a file is written giving the displaced state which can be used for stress initialization in later runs.

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

## DESCRIPTION OF KEYWORD INPUT

The keyword input provides a flexible and logically organized database that is simple to understand. Similar functions are grouped together under the same keyword. For example, under the keyword \*ELEMENT are included solid, beam, shell elements, spring elements, discrete dampers, seat belts, and lumped masses.

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

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

Alternatively, acceptable input could also be of the form:

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

# INTRODUCTION

---

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

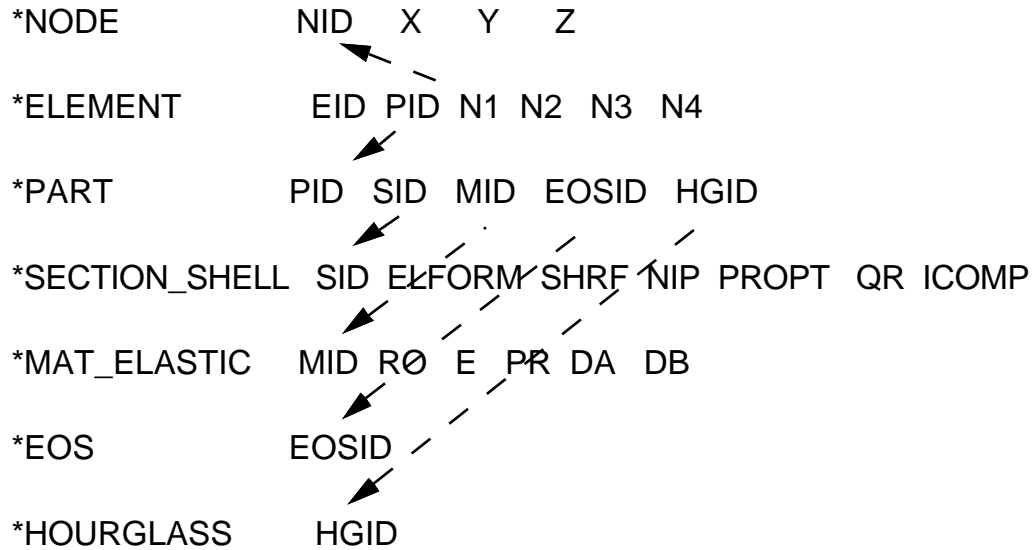
A data block begins with a keyword followed by the data pertaining to the keyword. The next keyword encountered during the reading of the block data defines the end of the block and the beginning of a new block. A keyword must be left justified with the “\*” contained in column one. A dollar sign “\$”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

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

During the keyword input phase where data is read, only limited checking is performed on the data since the data must first be counted for the array allocations and then reordered.

# INTRODUCTION

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



**Figure I.1** Organization of the keyword input.

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

```
*NODE
10101,x ,y ,z
10201,x ,y ,z
*ELEMENT_SHELL
10201,pid,n1,n2,n3,n4
10301,pid,n1,n2,n3,n4
```

# INTRODUCTION

---

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

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

## **\*AIRBAG**

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

## **\*BOUNDARY**

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

## **\*COMPONENT**

This section contains an analytical rigid body dummy that can be placed within vehicle and integrated implicitly.

## **\*CONSTRAINED**

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

## \*CONTACT

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

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

# INTRODUCTION

---

The \*CONTACT\_ENTITY section treats contact between a rigid surface, usually defined as an analytical surface, and a deformable structure. Applications of this type of contact exist in the metal forming area where the punch and die surface geometries can be input as VDA surfaces which are treated as rigid. Another application is treating contact between rigid body occupant dummy hyper-ellipsoids and deformable structures such as airbags and instrument panels. This option is particularly valuable in coupling with the rigid body occupant modeling codes MADYMO and CAL3D. The \*CONTACT\_1D is for modeling rebars in concrete structure.

## **\*CONTROL**

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

## **\*DAMPING**

Defines damping either globally or by part identifier.

## **\*DATABASE**

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

## **\*DEFINE**

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



## **\*DEFORMABLE\_TO\_RIGID**

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

## **\*ELEMENT**

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

## **\*EOS**

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

## **\*HOURLASS**

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

## **\*INCLUDE**

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

## **\*INITIAL**

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

# INTRODUCTION

---

## **\*INTEGRATION**

In this section the user defined integration rules for beam and shell elements are specified. IRID refers to integration rule number IRID on \*SECTION\_BEAM and \*SECTION\_SHELL cards respectively. Quadrature rules in the \*SECTION\_SHELL and \*SECTION\_BEAM cards need to be specified as a negative number. The absolute value of the negative number refers to user defined integration rule number. Positive rule numbers refer to the built in quadrature rules within LS-DYNA3D.

## **\*INTERFACE**

Interface definitions are used to define surfaces, nodal lines, and nodal points for which the displacement and velocity time histories are saved at some user specified frequency. This data may then be used in subsequent analyses as an interface ID in the \*INTERFACE\_LINKING\_DISCRETE\_NODE as master nodes, in \*INTERFACE\_LINKING\_SEGMENT as master segments and in \*INTERFACE\_LINKING\_EDGE as the master edge for a series of nodes. This capability is especially useful for studying the detailed response of a small member in a large structure. For the first analysis, the member of interest need only be discretized sufficiently that the displacements and velocities on its boundaries are reasonably accurate. After the first analysis is completed, the member can be finely discretized in the region bounded by the interfaces. Finally, the second analysis is performed to obtain highly detailed information in the local region of interest. When beginning the first analysis, specify a name for the interface segment file using the Z=parameter on the LS-DYNA3D execution line. When starting the second analysis, the name of the interface segment file created in the first run should be specified using the L=parameter on the LS-DYNA command line. Following the above procedure, multiple levels of sub-modeling are easily accommodated. The interface file may contain a multitude of interface definitions so that a single run of a full model can provide enough interface data for many component analyses. The interface feature represents a powerful extension of LS-DYNA's analysis capabilities.

## **\*KEYWORD**

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

## **\*LOAD**

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

## **\*MAT**

This section allows the definition of constitutive constants for all material models available in LS-DYNA3D including springs, dampers, and seat belts. The material identifier, MID, points to the MID on the \*PART card.

## **\*NODE**

Define nodal point identifiers and their coordinates.

## **\*PART**

This keyword serves two purposes.

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

## **\*RIGIDWALL**

Rigid wall definitions have been divided into two separate sections, \_PLANAR and \_GEOMETRIC. Planar walls can be either stationary or moving in translational motion with mass and initial velocity. The planar wall can be either finite or infinite. Geometric walls can be planar as well as have the geometric shapes such as rectangular prism, cylindrical prism and sphere. By default, these walls are stationary unless the option MOTION is invoked for either prescribed translational velocity or displacement. Unlike the planar walls, the motion of the geometric wall is governed by a load curve. Multiple geometric walls can be defined to model combinations of geometric shapes available. For example, a wall defined with the \_CYLINDER option can be combined with two walls defined with the \_SPHERICAL option to model hemispherical surface caps on the two ends of a cylinder. Contact entities are also analytical surfaces but have the significant advantage that the motion can be influenced by the contact to other bodies, or prescribed with six full degrees-of-freedom.

# INTRODUCTION

---

## **\*SET**

A concept of grouping nodes, elements, materials, etc., in sets is employed throughout the LS-DYNA3D input deck. Sets of data entities can be used for output. So-called slave nodes used in contact definitions, slaves segment sets, master segment sets, pressure segment sets and so on can also be defined. The keyword, \*SET, can be defined in two ways:

1. Option `_LIST` requires a list of entities, eight entities per card, and define as many cards as needed to define all the entities.
2. Option `_COLUMN`, where applicable, requires an input of one entity per line along with up to four attribute values which are needed to specify, for example, failure criterion input that is needed for `*CONTACT_CONSTRAINT_NODES_TO_SURFACE`.

## **\*TITLE**

In this section a title for the analysis is defined.

## **\*USER\_INTERFACE**

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

## **RESTART**

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

## **\*RIGID\_TO\_DEFORMABLE**

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

## **\*STRESS\_INITIALIZATION**

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

## SUMMARY OF COMMONLY USED OPTIONS

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

**Table I.1.** Keywords for the most commonly used options.

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

# INTRODUCTION

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

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

## MATERIAL MODELS

Some of the material models presently implemented are:

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

# INTRODUCTION

---

- laminated glass model,
- fabric,
- unified creep plasticity,
- temperature and rate dependent plasticity,
- elastic with viscosity,
- anisotropic plasticity,
- user defined,
- crushable cellular foams (Neilsen, Morgan, and Krieg 1987),
- urethane foam model with hysteresis,

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

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

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

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



## SPATIAL DISCRETIZATION

The elements shown in Figure I.2 are presently available. Currently springs, dampers, beams, membranes, shells, bricks, thick shells and seatbelt elements are included.

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

- It is incrementally objective (rigid body rotations do not generate strains), allowing for the treatment of finite strains that occur in many practical applications;
- It is compatible with brick elements, because the element is based on a degenerated brick element formulation. This compatibility allows many of the efficient and effective techniques developed for the DYNA3D brick elements to be used with this shell element;
- It includes finite transverse shear strains;
- A through-the-thickness thinning option (see [Hughes and Carnoy 1981]) is also available.

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

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

Since the Belytschko-Tsay element is based on a perfectly flat geometry, warpage is not considered. Although this generally poses no major difficulties and provides for an efficient element, incorrect results in the twisted beam problem and similar situations are obtained where the nodal points of the elements used in the discretization are not coplanar. The Hughes-Liu shell

# INTRODUCTION

---

element considers non-planar geometries and gives good results on the twisted beam. The effect of neglecting warpage in a typical application cannot be predicted beforehand and may lead to less than accurate results, but the latter is only speculation and is difficult to verify in practice. Obviously, it would be better to use shells that consider warpage if the added costs are reasonable and if this unknown effect is eliminated. Another shell published by Belytschko, Wong, and Chiang [Belytschko, Wong, and Chiang 1989, 1992] proposes inexpensive modifications to include the warping stiffness in the Belytschko-Tsay shell. An improved transverse shear treatment also allows the element to pass the Kirchhoff patch test. This element is now available in LS-DYNA. Also, two fully integrated shell elements, based on the Hughes and Liu formulation, are available in LS-DYNA, but are rather expensive. A much faster fully integrated element which is essentially a fully integrated version of the Belytschko, Wong, and Chiang element, type 16, is a more recent addition and is recommended if fully integrated elements are needed due to its cost effectiveness.

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

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

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

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

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

Solid elements in LS-DYNA may be defined using from 4 to 8 nodes. The standard elements are based on linear shape functions and use one point integration and hourglass control. A selective-reduced integrated (called fully integrated) 8 node solid element is available for situations when the hourglass control fails. Also, two additional solid elements, a 4 noded tetrahedron and an 8 noded

hexahedron, with nodal rotational degrees of freedom, are implemented based on the idea of Allman [1984] to replace the nodal midside translational degrees of freedom of the elements with quadratic shape functions by corresponding nodal rotations at the corner nodes. The latter elements, which do not need hourglass control, require many numerical operations compared to the hourglass controlled elements and should be used at places where the hourglass elements fail. However, it is well known that the elements using more than one point integration are more sensitive to large distortions than one point integrated elements.

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

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

# INTRODUCTION

---

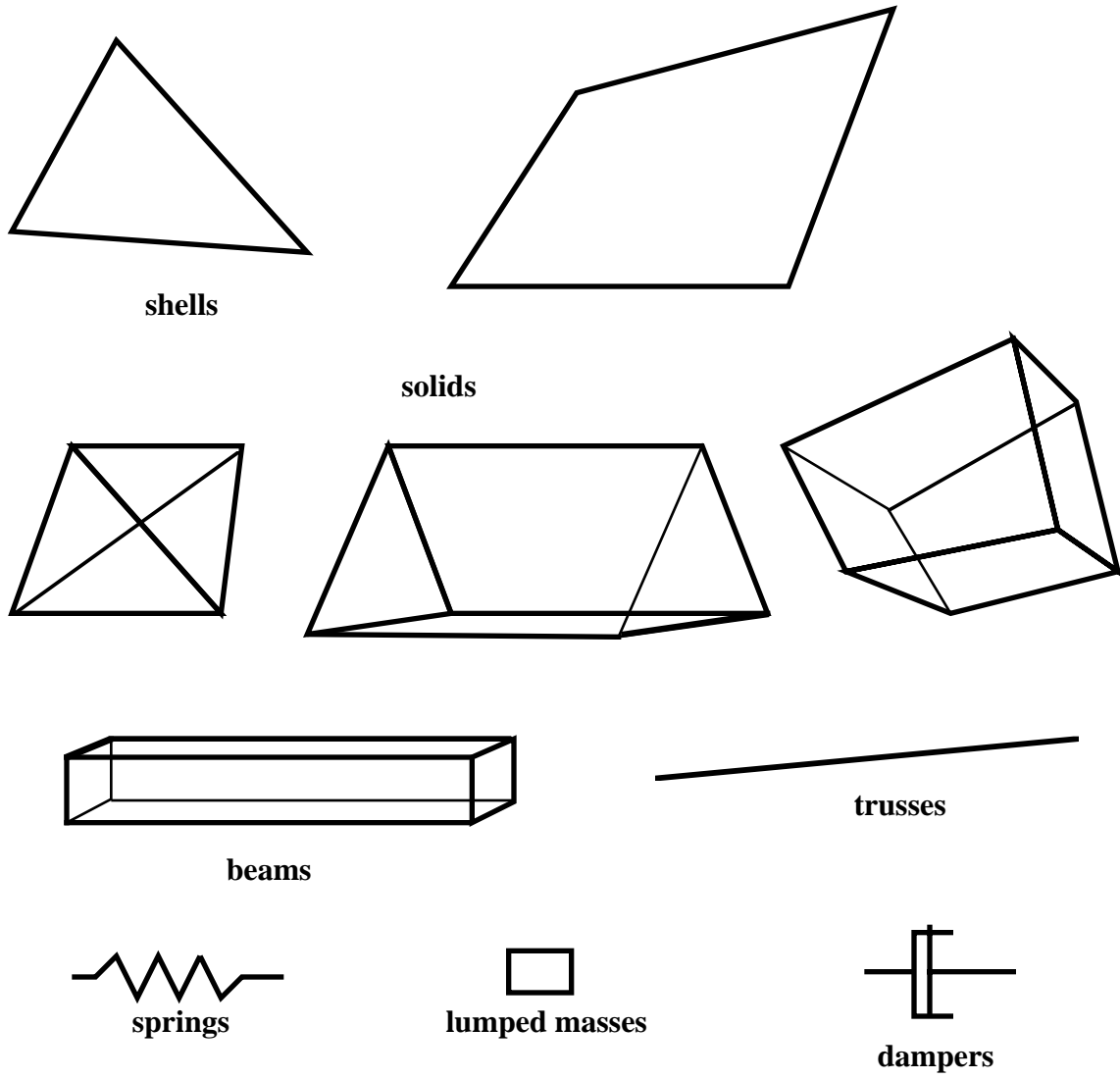


Figure I.2. Elements in LS-DYNA.

## CONTACT-IMPACT INTERFACES

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

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

# INTRODUCTION

---

edge-to-edge,  
draw beads.

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

For geometric contact entities, contact has to be separately defined. It must be noted that for the contact of a rigid body with a flexible body, either the sliding interface definitions as explained above or the geometric contact entity contact can be used. Currently, the geometric contact entity definition is recommended for metalforming problems due to high accuracy and computational efficiency.

## INTERFACE DEFINITIONS FOR COMPONENT ANALYSIS

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

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

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

# INTRODUCTION

---

## CAPACITY

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



## 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 with some parts programmed in C. LS-DYNA has eight segments in the main code. They are:

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

Parallel versions of LS-DYNA for shared memory are supported for the SGI and CRAY computers, and a distributed memory version of LS-DYNA has been ported to a subset of the commercially available MPP machines including the CRAY-T3D, IBM SP1/SP2, and the INTEL PARAGON.

# INTRODUCTION

---

## 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:

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

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

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

## PRECISION

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

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

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

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

# INTRODUCTION

---

## EXECUTION SYNTAX

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

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

where

- inf** = input file (user specified)
- otf** = high speed printer file (default=D3HSP)
- ptf** = binary plot file for graphics (default=D3PLOT)
- dpf** = dump file for restarting (default=D3DUMP)
- thf** = binary plot file for time histories of selected data (default=D3THDT)
- xtf** = binary plot file for time extra data (default=XTFILE)
- tpf** = optional temperature file (TOPAZ3D plotfile)
- rrd** = running restart dump file (default=RUNRSF)
- sif** = stress initialization file (user specified)
- jif** = optional JOY interface file
- iff** = interface force file (user specified)
- isf1** = interface segment save file to be created (user specified)
- isf2** = existing interface segment save file to be used (user specified)
- rlf** = binary plot file for dynamic relaxation (default=D3DRFL)
- efl** = echo file containing optional input echo with or without node/element data
- root** = root file name for general print option
- scl** = scale factor for binary file sizes (default=7)
- cpu** = cpu limit in seconds, applies to total calculation not just cpu from a restart
- kill** = if LS-DYNA encounters this file name it will terminate with a restart file (default=D3KIL)
- vda** = VDA/IGES database for geometrical surfaces
- c3d** = CAL3D input file
- nwds** = Number of words to be allocated. On engineering workstations a word is usually 32bits. **This number is ignored if memory is specified on the \*KEYWORD card at the beginning of the input deck.**
- ncpu** = Overrides **NCPU** and **CONST** defined in \*CONTROL\_PARALLEL. A positive value sets CONST=2 and a negative values sets CONST=1. See \*CONTROL\_PARALLEL for an explanation of these parameters.
- npara** = Overrides **PARA** defined in \*CONTROL\_PARALLEL.
- time** = Overrides **ENDTIM** defined in \*CONTROL\_TERMINATION.
- ncycle** = Overrides **ENDCYC** defined in \*CONTROL\_TERMINATION.

# File Organization

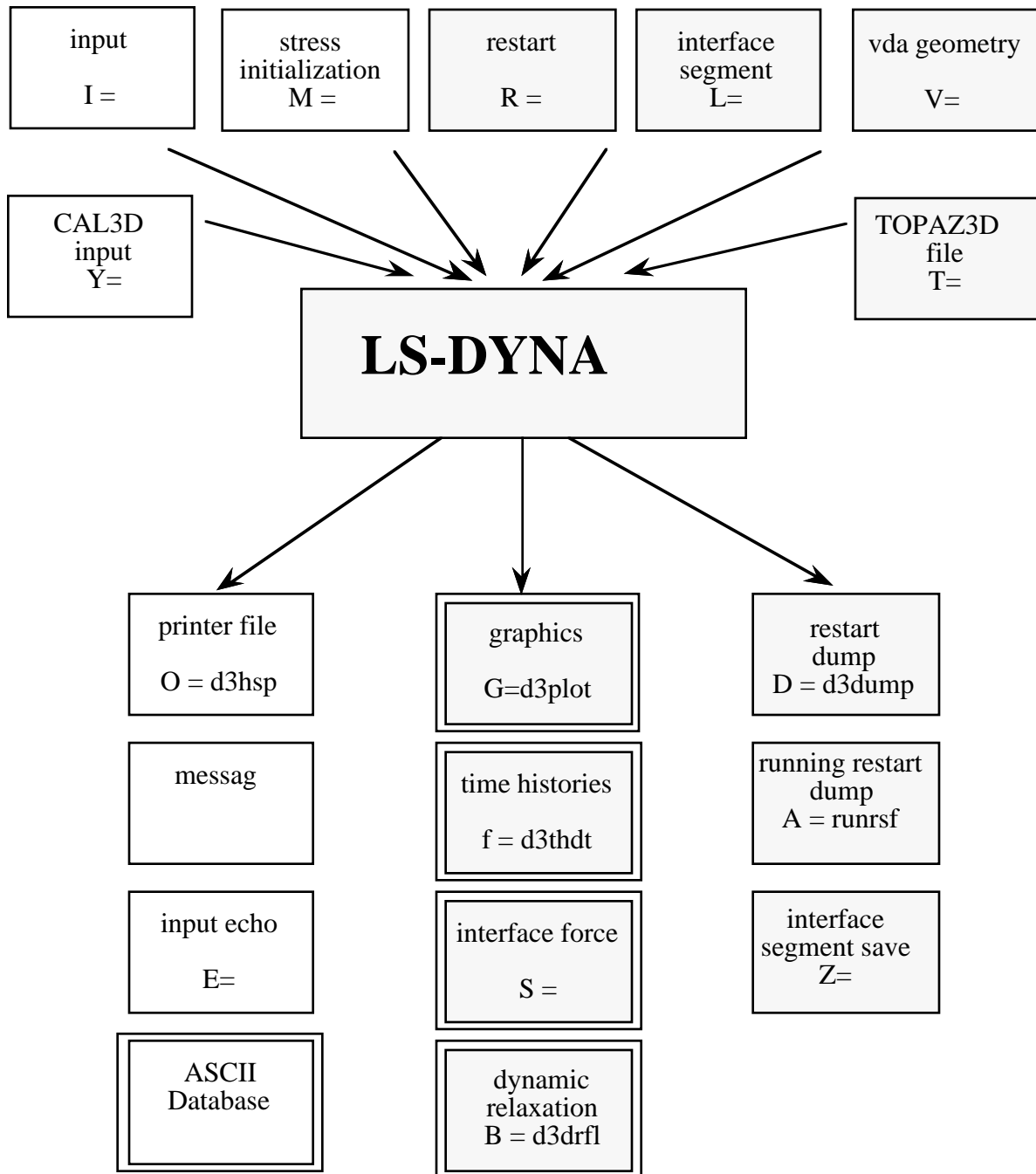


Figure I.3

# INTRODUCTION

---

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

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

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

The **INIT** (or **sw1.** can be used instead) command on the execution line causes the calculation to run just one cycle followed by termination with a full restart file. No editing of the input deck is required. The calculation can then be restarted with or without any additional input. Sometimes this option can be used to reduce the memory on restart if the required memory is given on the execution line and is specified too large in the beginning when the amount of required memory is unknown. Generally, this option would be used at the beginning of a new calculation.

If the word **MEMORY** is found anywhere on the execution line and if it is not set via (**=nwds**) LS-DYNA3D will give the default size of memory, request, and then read in the desired memory size. This option is necessary if the default value is insufficient memory and termination occurs as a result. Occasionally, the default value is too large for execution and this option can be used to lower the default size. Memory can also be specified on the **\*KEYWORD** card.

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

When restarting from a dump file, the execution line becomes

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

where

**rtf** = restart filename.

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

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

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

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

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

LS-DYNA I=**inf**

LS-DYNA R=**rtf**

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

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

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

and in the second by:

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

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

I=**inf**

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

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

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

# INTRODUCTION

---

## RESTART ANALYSIS

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

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

1. Create the restart input deck, if required, as described in the Restart Section of this manual. Call this file *restartinput*.
2. By invoking the execution line:

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

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

LS-DYNA R=*D3DUMPnn*

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

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

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

- Crash analysis continued with extra contact surfaces;



- Sheet metalforming continued with different tools for modeling a multi-stage forming process.

Assume an analysis is run using the input file, job1.inf, and a restart dump named d3dump01 is created. A new input file job2.inf is generated and submitted as a restart with R=d3dump01 as the dump file. The input file job2.inf contains the entire model in its original undeformed state but with more contact surfaces, new output databases, and so on. Since this is a restart job, information must be given to tell LS-DYNA which parts of the model should be initialized in the full deck restart. When the calculation begins the restart database contained in the file d3dump01 is read, and a new database is created to initialize the model in the input file, job2.inf. The data in file job2.inf is read and the LS-DYNA proceeds through the entire input deck and initialization. At the end of the initialization process, all the parts selected are initialized from the data saved from d3dump01. This means that the deformed position and velocities of the nodes on the elements of each part, and the stresses and strains in the elements (and, if the material of the part is rigid, the rigid body properties) will be assigned.

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

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

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

Points to note are:

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

# **INTRODUCTION**

---

## **VDA/IGES DATABASES**

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

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

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

## MESH GENERATION

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

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

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

Capabilities specialized to automotive applications:

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

---

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

# INTRODUCTION

---

## LS-POST POST-PROCESSING

LS-POST processes output from LS-DYNA. LS-POST reads the binary plot-files generated by LS-DYNA and plots contours, fringes, time histories, and deformed shapes. Color contours and fringes of a large number of quantities may be interactively plotted on meshes consisting of plate, shell, and solid type elements. LS-POST can compute a variety of strain measures, reaction forces along constrained boundaries, and momenta. LS-POST is operational on the CRAY, SUN, DEC, IBM RS6000, SGI, HP and PC computers.

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

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

# File Organization

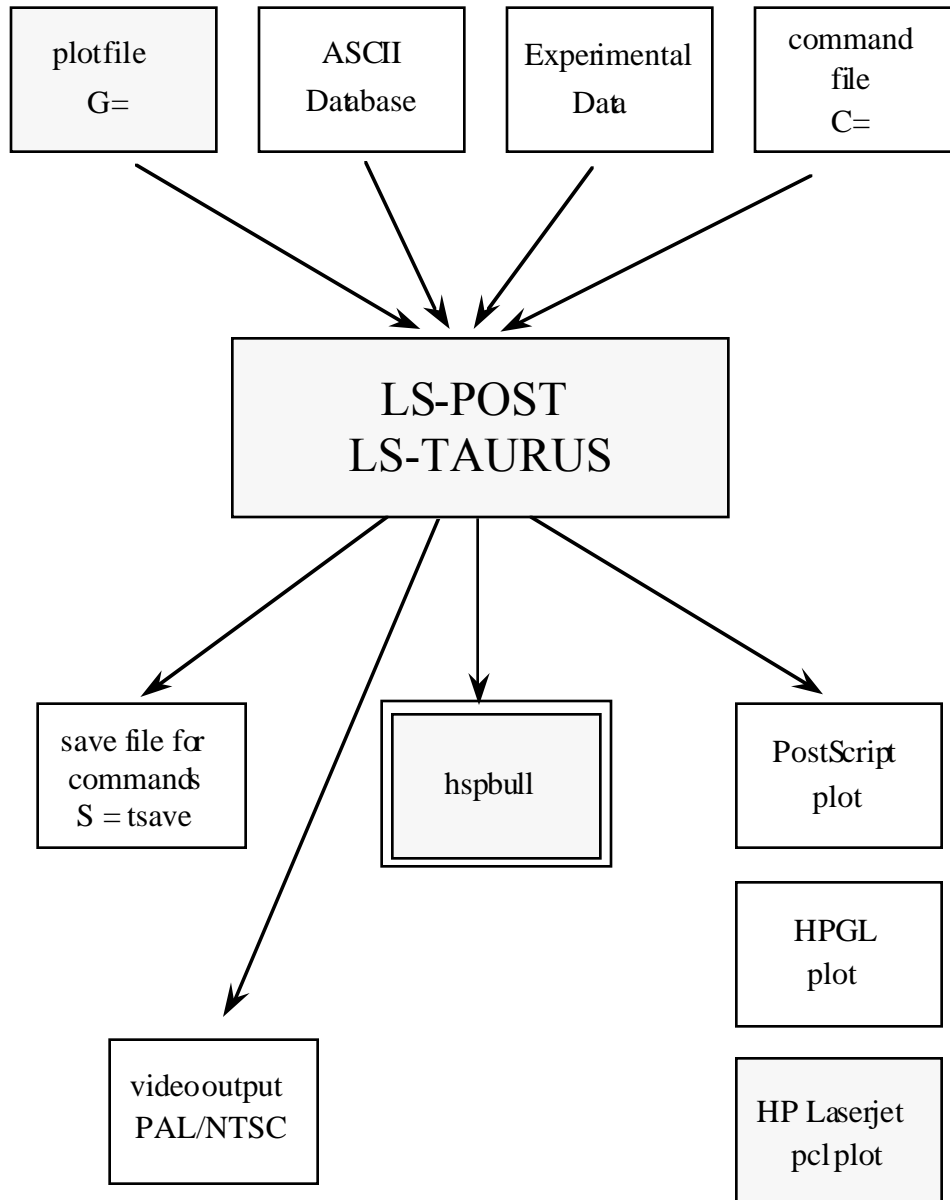


Figure I.4

# INTRODUCTION

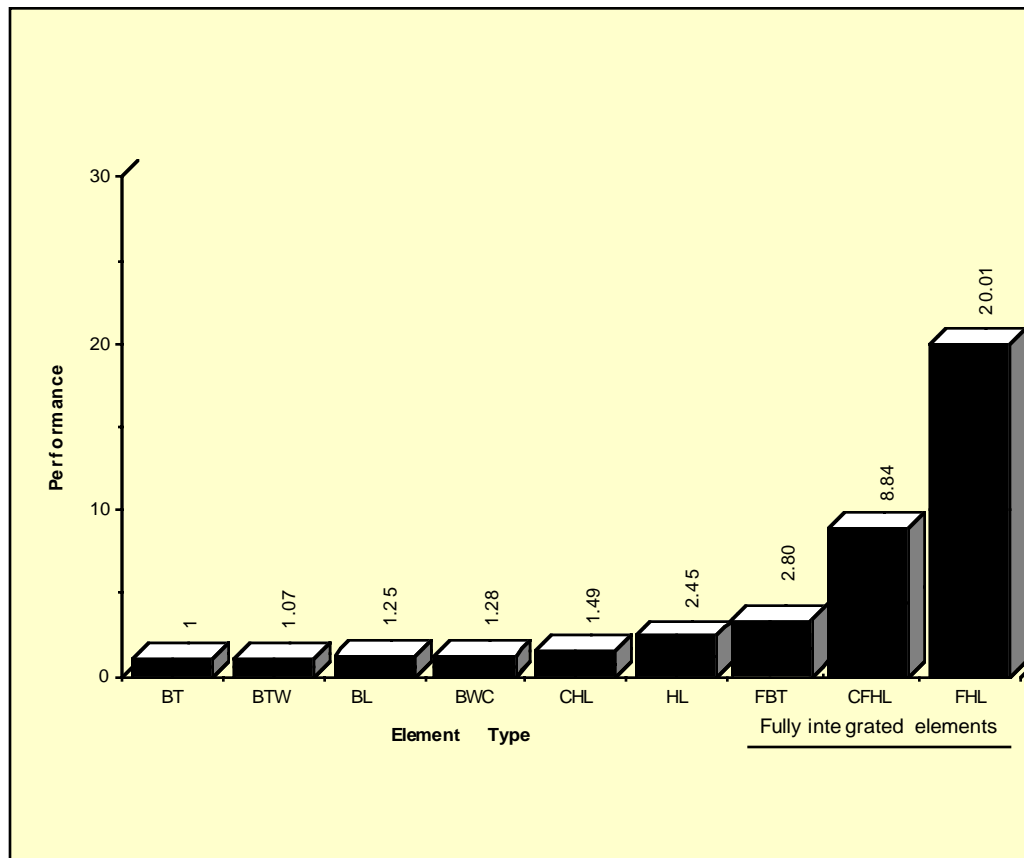
---

## EXECUTION SPEEDS

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

<u>Element Type</u>	<u>CPU Cost</u>
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 Belytschko-Tsay shell with four thickness integration points	11
4 node Belytschko-Tsay shell with resultant plasticity	9
BCIZ triangular shell with four thickness integration points	22
C <sup>0</sup> 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. Figure I.5 illustrates the relative cost of the various shell formulations in LS-DYNA3D.



**Figure I.5.** Relative cost of the four noded shells available in LS-DYNA where BT is the Belytschko-Tsay shell, BTW is the Belytschko-Tsay shell with the warping stiffness taken from the Belytschko-Wong-Chiang, BWC, shell. The BL shell is the Belytschko-Leviathan shell. CHL denotes the Hughes-Liu shell, HL, with one point quadrature and a co-rotational formulation. FBT is a Belytschko-Tsay like shell with full integration, FHL is the fully integrated Hughes-Liu shell, and the CFHL shell is its co-rotational version.

# INTRODUCTION

---

## UNITS

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

$$1 \text{ (force unit)} = 1 \text{ (mass unit)} \times 1 \text{ (acceleration unit)}$$

$$\text{and that } 1 \text{ (acceleration unit)} = \frac{1 \text{ (length unit)}}{[1 \text{ (time unit)}]^2}$$

Examples of sets of consistent units are:

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



## GENERAL CARD FORMAT

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

### Card Format

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

The type is the variable type and is either F, for floating point or I, for an integer. The default gives the value set if zero is specified, the field is left blank or the card is not defined. The remarks refer to comments at the end of the section. The card format is given above the card if it is other than eight fields of 10. Free formats may be used with the data separated by commas. When using comma format, the number of characters used to specify a number must not exceed the number which would fit into the equivalent rigid format field. An I8 number is limited to a number of 99999999 and larger numbers with more than eight characters are unacceptable. Rigid and free formats can be mixed throughout the deck but not within a card.

# INTRODUCTION

---

# **\*AIRBAG**

Purpose: Define an airbag or control volume.

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

**\*AIRBAG\_OPTION1\_{OPTION2}\_<OPTIONAL NUMERIC ID>**

**\*AIRBAG\_INTERACTION**

**\*AIRBAG\_REFERENCE\_GEOMETRY\_OPTION\_OPTION**

**\*AIRBAG\_OPTION1\_{OPTION2}**

*OPTION1* specifies one of the following thermodynamic relationships:

**SIMPLE\_PRESSURE\_VOLUME**

**SIMPLE\_AIRBAG\_MODEL**

**ADIABATIC\_GAS\_MODEL**

**WANG\_NEFSKE**

**WANG\_NEFSKE\_JETTING**

**WANG\_NEFSKE\_MULTIPLE\_JETTING**

**LOAD\_CURVE**

**LINEAR\_FLUID**

**HYBRID**

**HYBRID\_JETTING**

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

**POP**

The **OPTIONAL\_NUMERIC\_ID** is a unique number used only for identification of the airbag in the definition of airbag interaction via **\*AIRBAG\_INTERACTION**. The numeric ID is not used for any other purpose. To define an airbag using the Wang Nefske thermodynamic relationship with an ID of 25 the keyword is **\*AIRBAG\_WANG\_NEFSKE\_25**. For more information on these models the papers by Wang [1988, 1995] and Nusholtz [1991, 1996] are recommended.

# \*AIRBAG

---

## Card Format

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

### VARIABLE

### DESCRIPTION

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

### **Remarks:**

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

Lumped parameter control volumes are a mechanism for determining volumes of closed surfaces and applying a pressure based on some thermodynamic relationships. The volume is specified by a list of polygons similar to the pressure boundary condition cards or by specifying a

material subset which represents shell elements which form the closed boundary. All polygon normals must be oriented to face outwards from the control volume. If holes are detected, they are assumed to be covered by planar surfaces.

$V_{sca}$  and  $P_{sca}$  allow for unit system changes from the inflator to the finite element model. There are two sets of volume and pressure used for each control volume. First, the finite element model computes a volume ( $V_{femod\ell}$ ) and applies a pressure ( $P_{femod\ell}$ ). 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:

$$V_{cvolume} = (V_{sca} V_{femod\ell}) - V_{ini}$$

$$P_{femod\ell} = P_{sca} P_{cvolume}$$

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

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

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

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

<p style="text-align: center;"><b>Sensor Input to Activate Inflator</b> <b>Define if and only if <i>RBID</i> nonzero.</b></p>
---

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

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

# \*AIRBAG

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

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

**Card Format**

1            2            3            4            5            6            7            8

Variable	N							
Type	I							
Default	none							

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

1            2            3            4            5            6            7            8

Variable	C1	C2	C3	C4	C5			
Type	F	F	F	F	F			
Default	0.	0.	0.	0.	0.			

<u>VARIABLE</u>	<u>DESCRIPTION</u>
N	Number of input parameters (not to exceed 25)
C1,...CN	Up to 25 constants user subroutine.

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

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

**Acceleration/Velocity/Displacement Activation**

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

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

	1	2	3	4	5	6	7	8
Variable	UX	UY	UZ	UMAG				
Type	F	F	F	F				
Default	0.	0.	0.	0.				

# \*AIRBAG

---

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



**Additional card required for SIMPLE\_PRESSURE\_VOLUME option**

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

<u>VARIABLE</u>	<u>DESCRIPTION</u>
CN	Constant. Define if a load curve ID is not specified.
BETA	Scale factor, $\beta$ . Define if a load curve ID is not specified.
LCID	Optional load curve ID defining pressure versus relative volume.

**Remarks:**

The relationship is the following:

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

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

The pressure is then a function of the ratio of current volume to the initial volume. The constant, CN, is used to establish a relationship known from the literature. The scale factor  $\beta$  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.

# \*AIRBAG

## Additional cards required for SIMPLE\_AIRBAG\_MODEL option

Card 1            1            2            3            4            5            6            7            8

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

Card 2

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

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

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

**Remarks:**

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

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

where  $p$  is the pressure,  $\rho$  is the density,  $e$  is the specific internal energy of the gas, and  $\gamma$  is the ratio of the specific heats:

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

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

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

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

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

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

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

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

# \*AIRBAG

## Additional card required for ADIABATIC\_GAS\_MODEL option

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

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

### Remarks:

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

$$p = PSF p_0$$

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

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

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

where  $p$  is the pressure,  $\rho$  is the density,  $e$  is the specific internal energy of the gas, and  $\gamma$  is the ratio of the specific heats:

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

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

$$p_s = PSF(p - p_e)$$

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

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

# \*AIRBAG

---

**Additional 4 cards are required for all WANG\_NEFSKE models**

Card 1            1            2            3            4            5            6            7            8

Variable	CV	CP	T	LCT	LCMT	TVOL		
Type	F	F	F	I	I	F		
Default	none	none	0.	0	none	0.		

Card 2            1            2            3            4            5            6            7            8

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

Card 3            1            2            3            4            5            6            7            8

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

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

Card 4            1            2            3            4            5            6            7            8

Variable	IOC	IOA	IVOL	IRO	IT	LCBF		
Type	F	F	F	F	F	I		
Default	none	none	none	none	none	none		

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

Card 5            1            2            3            4            5            6            7            8

Variable	TEXT	A	B	MW	GASC			
Type	F	F	F	F	F			
Default	none	none	none	none	none			

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

Card 5            1            2            3            4            5            6            7            8

Variable	TDP	AXP	AYP	AZP	AMAGP	TDURP	TDA	RBIDP
Type	F	F	F	F	F	F	F	I
Default	0.0	0.0	0.0	0.0	0.0	0.0	0.0	none

**VARIABLE**

**DESCRIPTION**

- CV            Heat capacity at constant volume
- CP            Heat capacity at constant pressure

# \*AIRBAG

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
T	Temperature of input gas. For temperature variations a load curve, LCT, may be defined.
LCT	Optional load curve number defining temperature of input gas versus time. This overrides columns T.
LCMT	Load curve specifying input mass flow rate or tank pressure versus time. If the tank volume, TVOL, is nonzero the curve ID is assumed to be tank pressure versus time. If LCMT=0, then the inflator has to be modeled, see Card 4.
TVOL	Tank volume which is required only for the tank pressure versus time curve, LCMT.
C23	Vent orifice coefficient which applies to exit hole. Set to zero if LCC23 is defined below.
LCC23	Load curve number defining the vent orifice coefficient which applies to exit hole as a function of time. A nonzero value for C23 overrides LCC23.
A23	Vent orifice area which applies to exit hole. Set to zero if LCA23 is defined below.
LCA23	Load curve number defining the vent orifice area which applies to exit hole as a function of <u>absolute</u> pressure. A nonzero value for A23 overrides LCA23.
CP23	Orifice coefficient for leakage (fabric porosity). Set to zero if LCCP23 is defined below.
LCCP23	Load curve number defining the orifice coefficient for leakage (fabric porosity) as a function of time. A nonzero value for CP23 overrides LCCP23.
AP23	Area for leakage (fabric porosity)
LCAP23	Load curve number defining the area for leakage (fabric porosity) as a function of (absolute) pressure. A nonzero value for AP23 overrides LCAP23.
PE	Ambient pressure
RO	Ambient density
GC	Gravitational conversion constant (mandatory - no default). If consistent units are being used for all parameters in the airbag definition then unity should be input.
LCEFR	Optional curve for exit flow rate versus (gauge) pressure
POVER	Initial relative overpressure (gauge), $P_{over}$ in control volume



---

<b>VARIABLE</b>	<b>DESCRIPTION</b>
PPOP	Pop Pressure: relative pressure (gauge) for initiating exit flow, $P_{pop}$
OPT	Fabric venting option, if nonzero CP23, LCCP23, AP23, and LCAP23 are set to zero. EQ. 1: Wang-Nefske formulas for venting through an orifice are used. Blockage is not considered. EQ. 2: Wang-Nefske formulas for venting through an orifice are used. Blockage of venting area due to contact is considered. EQ. 3: Leakage formulas of Graefe, Krummheuer, and Siejak [1990] are used. Blockage is not considered. EQ. 4: Leakage formulas of Graefe, Krummheuer, and Siejak [1990] are used. Blockage of venting area due to contact is considered. EQ. 5: Leakage formulas based on flow through a porous media are used. Blockage is not considered. EQ. 6: Leakage formulas based on flow through a porous media are used. Blockage of venting area due to contact is considered.
KNKDN	<u>Optional</u> load curve ID defining the knock down pressure scale factor versus time. This option only applies to jetting. The scale factor defined by this load curve scales the pressure applied to airbag segments which do not have a clear line-of-sight to the jet. Typically, at very early times this scale factor will be less than unity and equal to unity at later times. The full pressure is always applied to segments which can see the jets.
IOC	Inflator orifice coefficient
IOA	Inflator orifice area
IVOL	Inflator volume
IRO	Inflator density
IT	Inflator temperature
LCBF	Load curve defining burn fraction versus time
TEXT	Ambient temperature.
A	First heat capacity coefficient of inflator gas (e.g., Joules/mole/ $^{\circ}$ K)
B	Second heat capacity coefficient of inflator gas, (e.g., Joules/mole/ $^{\circ}$ K <sup>2</sup> )
MW	Molecular weight of inflator gas (e.g., Kg/mole).
GASC	Universal gas constant of inflator gas (e.g., 8.314 Joules/mole/ $^{\circ}$ K)
TDP	Time delay before initiating exit flow after pop pressure is reached.
AXP	Pop acceleration magnitude in local x-direction. EQ. 0.0: Inactive.

## \*AIRBAG

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
AYP	Pop acceleration magnitude in local y-direction. EQ. 0.0: Inactive.
AZP	Pop acceleration magnitude in local z-direction. EQ. 0.0: Inactive.
AMAGP	Pap acceleration magnitude. EQ. 0.0: Inactive.
TDURP	Time duration pop acceleration must be exceeded to initiate exit flow. This is a cumulative time from the beginning of the calculation, i.e., it is not continuous.
TDA	Time delay before initiating exit flow after pop acceleration is exceeded for the prescribed time duration.
RBIDP	Part ID of the rigid body for checking accelerations against pop accelerations.

### **Remarks:**

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

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

where  $p$  is the pressure,  $\rho$  is the density,  $e$  is the specific internal energy of the gas, and  $\gamma$  is the ratio of the specific heats:

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

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

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

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

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

where  $\gamma$  is the ratio of specific heats:

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

If

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

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

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

and

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

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

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

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

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

$$\begin{aligned} p_e &= p_a \quad \text{if} \quad p_a \geq p_c \\ p_e &= p_c \quad \text{if} \quad p_a < p_c \end{aligned}$$

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

$$\dot{m}_{in} = C_o A_o \sqrt{2p_i \rho_i} \sqrt{\frac{g_c \gamma \left( Q^{\frac{2}{\gamma}} - Q^{\frac{\gamma+1}{\gamma}} \right)}{\gamma - 1}}$$

where  $C_o$ ,  $A_o$ , and  $\rho_i$  are the inflator orifice coefficient, area, and gas density, respectively.

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

## \*AIRBAG

---

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

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

If OPT set to 3 or 4 then:

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

and for OPT set to 5 or 6:

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

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

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

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

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

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

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

**Further additional 2 cards are required for JETTING models**

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

Define either card below but not both:

**1st additional card of 2 required for WANG\_NEFSKE\_JETTING option**

Card 1            1            2            3            4            5            6            7            8

Variable	XJFP	YJFP	ZJFP	XJVH	YJVH	ZJVH	CA	BETA
Type	F	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	none	1.0
Remark	1	1	1	1	1	1		

**1st additional card of 2 required for WANG\_NEFSKE\_MULTIPLE\_JETTING option**

Card 1            1            2            3            4            5            6            7            8

Variable	XJFP	YJFP	ZJFP	XJVH	YJVH	ZJVH	LCJRV	BETA
Type	F	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	none	1.0
Remark	1	1	1	1	1	1		

# \*AIRBAG

2nd additional card of 2 required for WANG\_NEFSKE\_JETTING and WANG\_NEFSKE\_MULTIPLE\_JETTING option

Card 2            1            2            3            4            5            6            7            8

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

## VARIABLE

## DESCRIPTION

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

---

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

**Remarks:**

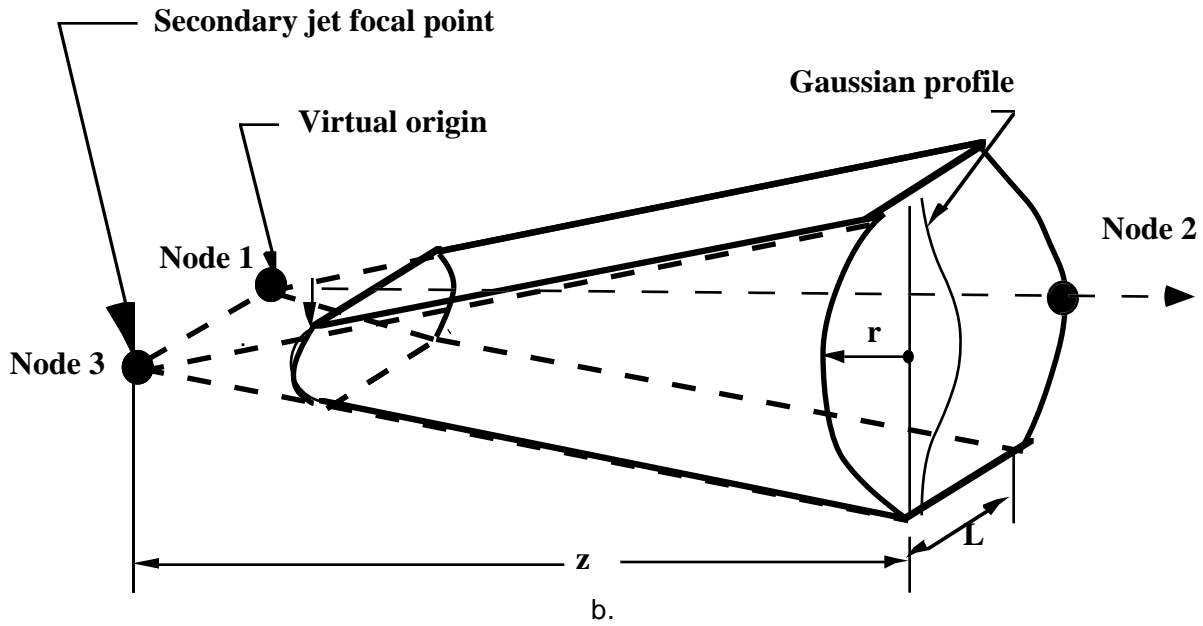
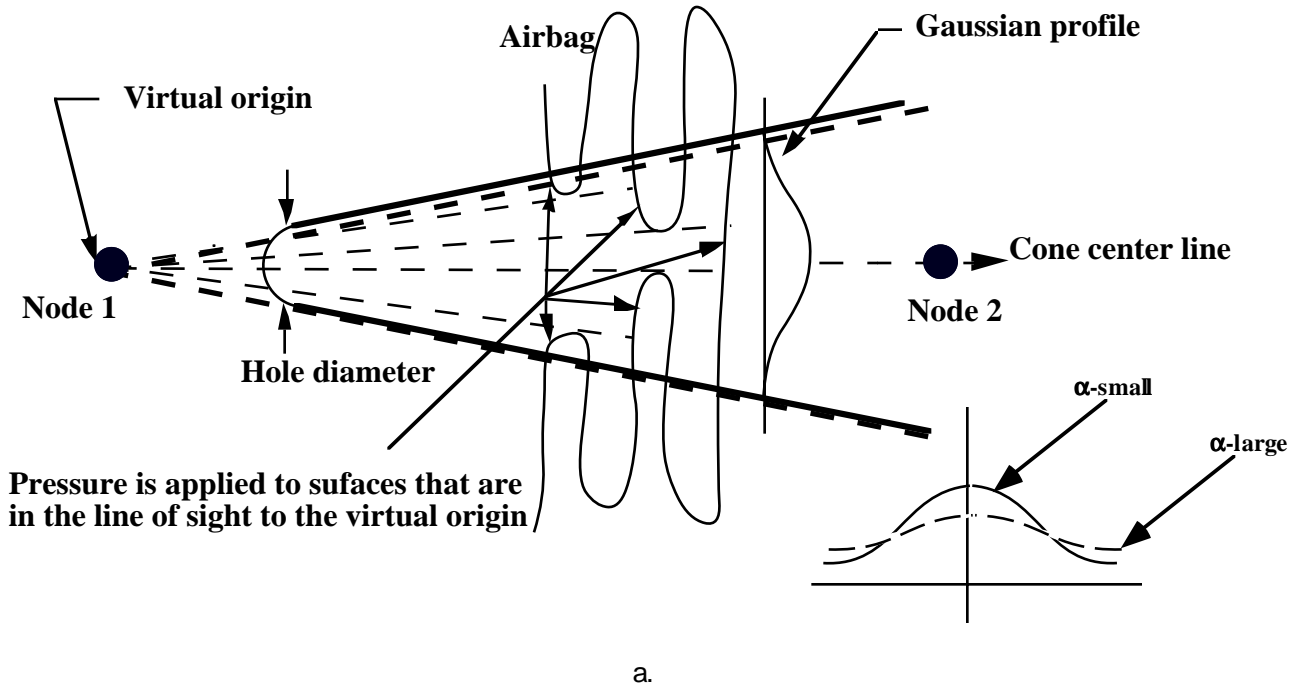
1. It is assumed that the jet direction is defined by the coordinate method (XJFP, YJFP, ZJFP) and (XJVH, YJVH, ZJVH) unless both NODE1 and NODE2 are defined. In which case the coordinates of the nodes give by NODE1, NODE2 and NODE3 will override (XJFP, YJFP, ZJFP) and (XJVH, YJVH, ZJVH). The use of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

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

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

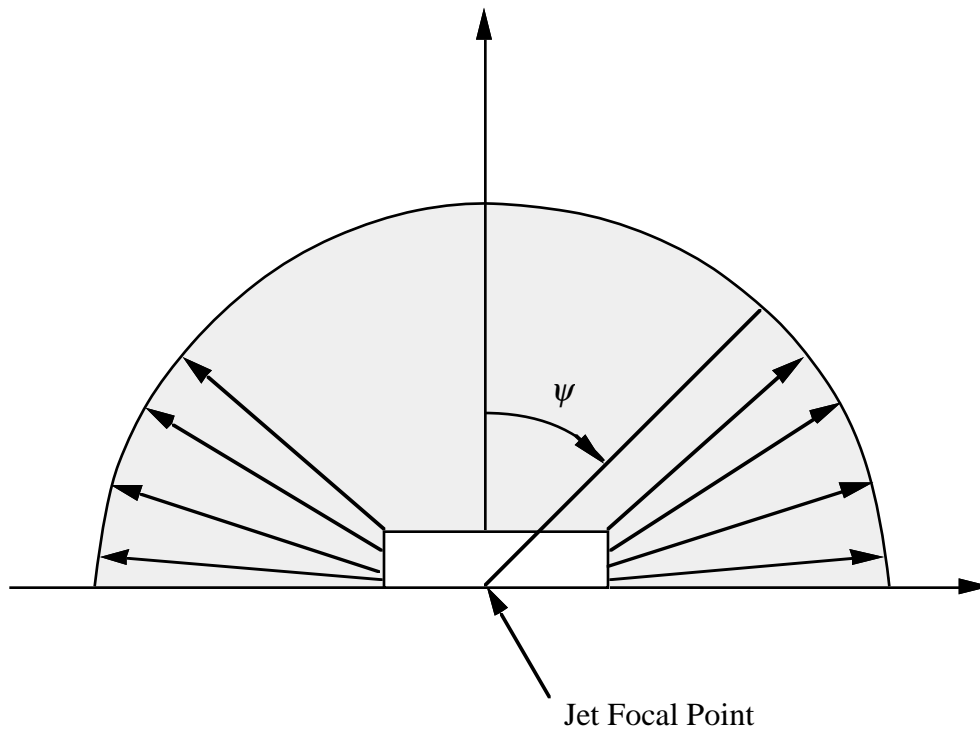
Instead of applying pressure to all surfaces in the line of sight of the virtual origin(s), a part set can be defined to which the pressure is applied.

# \*AIRBAG

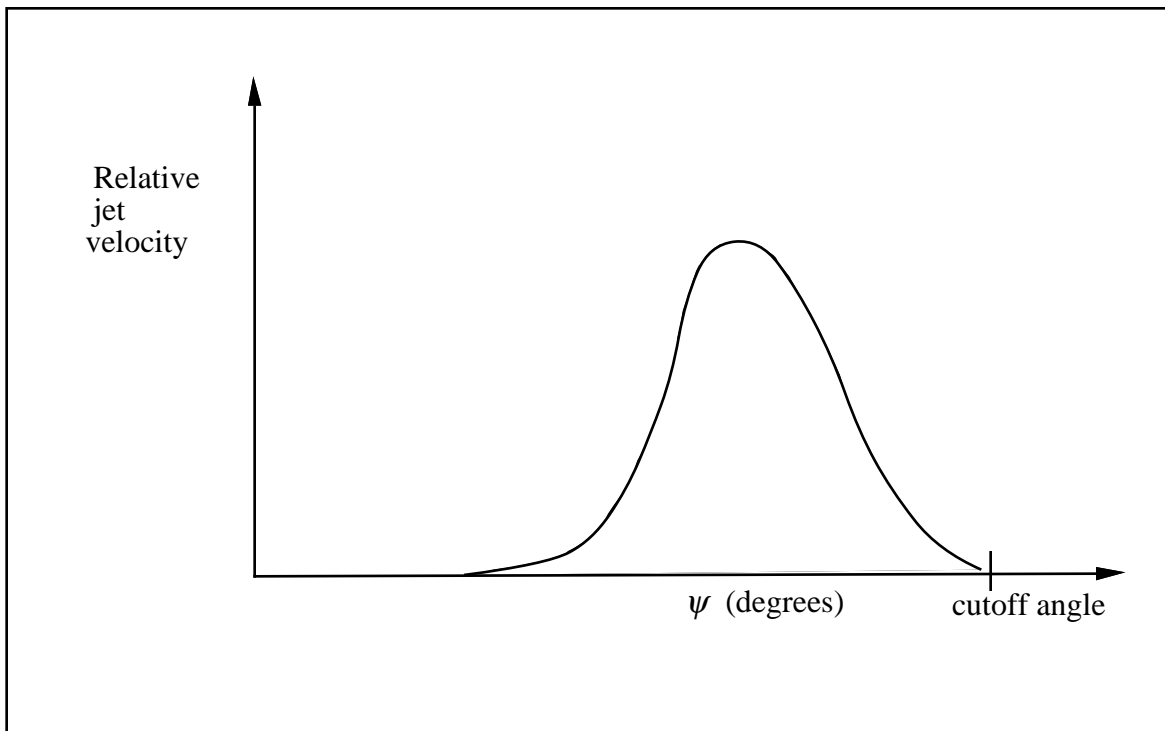


**Figure 1.1** Jetting configuration for (a.) driver's side airbag (pressure applied only if centroid of surface is in line-of-sight) and (b.) the passenger's side bag.





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



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

# \*AIRBAG

---

Additional card required for **LOAD\_CURVE** option

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

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

## **Remarks:**

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

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

$$P_{gauge} = P_{total} - P_{ambient}$$

The pressure is uniform throughout the control volume.

**Additional card required for LINEAR\_FLUID option**

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

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

**Remarks:**

If LCID = 0 then the pressure is determined from:

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

where

$P(t)$  Pressure,

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

## \*AIRBAG

---

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

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

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

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

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

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

**Additional cards required for HYBRID and HYBRID\_JETTING options**

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

	1	2	3	4	5	6	7	8
Variable	C23	LCC23	A23	LCA23	CP23	LCP23	AP23	LCAP23
Type	F	I	F	I	F	I	F	I
Default	none	0	none	0	none	0	none	0

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

# \*AIRBAG

---

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

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

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

<u>VARIABLE</u>	<u>DESCRIPTION</u>
ATMOST	Atmospheric temperature
ATMOSP	Atmospheric pressure
ATMOSD	Atmospheric density
GC	Universal molar gas constant
CC	Conversion constant EQ:0 Set to 1.0.
C23	Vent orifice coefficient which applies to exit hole. Set to zero if LCC23 is defined below.
LCC23	Load curve number defining the vent orifice coefficient which applies to exit hole as a function of time. A nonzero value for C23 overrides LCC23.

---

<b>VARIABLE</b>	<b>DESCRIPTION</b>
A23	Vent orifice area which applies to exit hole. Set to zero if LCA23 is defined below.
LCA23	Load curve number defining the vent orifice area which applies to exit hole as a function of <u>absolute</u> pressure. A nonzero value for A23 overrides LCA23.
CP23	Orifice coefficient for leakage (fabric porosity). Set to zero if LCCP23 is defined below.
LCCP23	Load curve number defining the orifice coefficient for leakage (fabric porosity) as a function of time. A nonzero value for CP23 overrides LCCP23.
AP23	Area for leakage (fabric porosity)
LCAP23	Load curve number defining the area for leakage (fabric porosity) as a function of (absolute) pressure. A nonzero value for AP23 overrides LCAP23.
OPT	Fabric venting option, if nonzero CP23, LCCP23, AP23, and LCAP23 are set to zero. EQ. 1: Wang-Nefske formulas for venting through an orifice are used. Blockage is not considered. EQ. 2: Wang-Nefske formulas for venting through an orifice are used. Blockage of venting area due to contact is considered. EQ. 3: Leakage formulas of Graefe, Krummheuer, and Siejak [1990] are used. Blockage is not considered. EQ. 4: Leakage formulas of Graefe, Krummheuer, and Siejak [1990] are used. Blockage of venting area due to contact is considered. EQ. 5: Leakage formulas based on flow through a porous media are used. Blockage is not considered. EQ. 6: Leakage formulas based on flow through a porous media are used. Blockage of venting area due to contact is considered.
PVENT	Gauge pressure when venting begins
NGAS	Number of gas inputs to be defined below. (Including initial air)

## \*AIRBAG

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
LCIDM	Load curve ID for inflator mass flow rate (eq. 0 for gas in the bag at time 0)
LCIDT	Load curve ID for inflator gas temperature (eq.0 for gas in the vag at time 0)
BLANK	(not used)
MW	Molecular weight
INITM	Initial mass fraction of gas component
A	Coefficient for molar heat capacity of inflator gas at constant pressure, (e.g., Joules/mole/°K)
B	Coefficient for molar heat capacity of inflator gas at constant pressure, (e.g., Joules/mole/°K)
C	Coefficient for molar heat capacity of inflator gas at constant pressure, (e.g., Joules/mole/°K)
FMASS	Fraction of additional aspirated mass.



**Further additional 2 cards are required for HYBRID\_JETTING models**

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

Card 1            1            2            3            4            5            6            7            8

Variable	XJFP	YJFP	ZJFP	XJVH	YJVH	ZJVH	CA	BETA
Type	F	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	none	none
Remark	1	1	1	1	1	1		

Card 2            1            2            3            4            5            6            7            8

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

**VARIABLE**

**DESCRIPTION**

- XJFP            x-coordinate of jet focal point, i.e., the virtual origin in Figure 1.1. See Remark 1 below.
- YJFP            y-coordinate of jet focal point, i.e., the virtual origin in Figure 1.1.
- ZJFP            z-coordinate of jet focal point, i.e., the virtual origin in Figure 1.1.
- XJVH            x-coordinate of jet vector head to defined code centerline

# \*AIRBAG

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
YJVH	y-coordinate of jet vector head to defined code centerline
ZJVH	z-coordinate of jet vector head to defined code centerline
CA	Cone angle, $\alpha$ , defined in radians. LT.0.0: $ \alpha $ is the load curve ID defining cone angle as a function of <i>time</i>
LCRJV	Load curve ID giving the spatial jet relative velocity distribution, see Figures 1.2 and 1.3. The jet velocity is determined from the inflow mass rate and scaled by the load curve function value corresponding to the value of the angle $\psi$ . Typically, the values on the load curve vary between 0 and unity. See *DEFINE_CURVE.
BETA	Efficiency factor, $\beta$ , which scales the final value of pressure obtained from Bernoulli's equation. LT.0.0: $ \beta $ is the load curve ID defining the efficiency factor as a function of <i>time</i>
XSJFP	x-coordinate of secondary jet focal point, passenger side bag. If the coordinate of the secondary point is (0,0,0) then a conical jet (driver's side airbag) is assumed.
YSJFP	y-coordinate of secondary jet focal point
ZSJFP	z-coordinate of secondary jet focal point
PSID	Optional part set ID, see *PART. If zero all elements are included in the airbag.
IDUM	Dummy field (Variable not used)
NODE1	Node ID located at the jet focal point, i.e., the virtual origin in Figure 1.1. See Remark 1 below.
NODE2	Node ID for node along the axis of the jet .
NODE3	Optional node ID located at secondary jet focal point.

## **Remarks:**

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

The jetting option provides a simple model to simulate the real pressure distribution in the airbag during the breakout and early unfolding phase. Only the surfaces that are in the line of

sight to the virtual origin have an increased pressure applied. With the optional load curve LCRJV, the pressure distribution with the code can be scaled according to the so-called relative jet velocity distribution.

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

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

2. This variable is not used and has been included to maintain the same format as the WANG\_NEFSKE\_JETTING options.

# \*AIRBAG

---

## \*AIRBAG\_INTERACTION

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

Define one card for each airbag interaction definition

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

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

## **Remarks:**

All input options are valid for the following airbag types:

- \*AIRBAG\_SIMPLE\_AIRBAG\_MODEL
- \*AIRBAG\_WANG\_NEFSKE
- \*AIRBAG\_WANG\_NEFSKE\_JETTING
- \*AIRBAG\_WANG\_NEFSKE\_MULTIPLE\_JETTING
- \*AIRBAG\_HYBRID
- \*AIRBAG\_HYBRID\_JETTING

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

- \*AIRBAG\_LOAD\_CURVE
- \*AIRBAG\_LINEAR\_FLUID

If the AREA, SF, and PID defined method is used to define the interaction then the airbags must contain the same gas, i.e.  $C_p$ ,  $C_v$  and  $g$  must be the same. The flow between bags is governed by formulas which are similar to those of Wang-Nefske, except that choked flow is currently ignored. This will be added later.

## \*AIRBAG

---

### \*AIRBAG\_REFERENCE\_GEOMETRY\_OPTION\_OPTION

Available options include:

#### **BIRTH**

#### **RDT**

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

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

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

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

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

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

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

1            2            3            4            5            6            7            8

Variable	BIRTH							
Type	F							
Default	0.0							

**Card Format (I8,3E16.0)**

Card 2,...            1            2            3            4            5            6            7            8            9            10

Variable	NID	X	Y	Z			
Type	I	F	F	F			
Default	none	0.	0.	0.			
Remarks							

**VARIABLE**

**DESCRIPTION**

- |       |  |
|-------|--|
| BIRTH | Time at which the reference geometry activates (default=0.0) |
| NID   | Node number  |
| X     | x coordinate   |
| Y     | y coordinate   |
| Z     | z coordinate   |

**\*AIRBAG**

---



# **\*ALE**

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

**\*ALE\_MULTI-MATERIAL\_GROUP**

**\*ALE\_SMOOTHING**

For other input information related to the ALE capability, see keywords: **\*CONTROL\_ALE**, **\*INITIAL\_VOID**, **\*SECTION\_SOLID\_ALE**, and **\*ALE\_SMOOTHING**.

# \*ALE

---

## \*ALE\_MULTI-MATERIAL\_GROUP\_OPTION

Available options are:

### **PART SET**

Purpose: The following input defines the PART 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 groups to be mixed within the same element.

For each group define the follow cards.

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

### **Card Format**

	1	2	3	4	5	6	7	8
Variable	PSID/PID							
Type	I							
Default	none							
Remarks	1							

### VARIABLE

### DESCRIPTION

PSID/PID

Part set ID or part ID, see also \*SET\_PART:

### **Remarks:**

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

Example

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

The above example defines a mixture of three groups of materials, oil, water and air, that is, the number of ALE groups, NALEGP=3.

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

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

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

# \*ALE

---

## \*ALE\_SMOOTHING

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

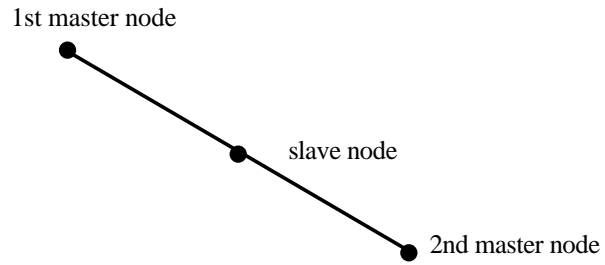
### Card Format

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

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

### **Remark:**

Arbitrary Lagrangian Eulerian meshes are defined via the choice of the element type-only solids elements can be used, and the \*CONTROL\_ALE card.



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



# **\*BOUNDARY**

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

**\*BOUNDARY\_CONVECTION\_OPTION**

**\*BOUNDARY\_CYCLIC**

**\*BOUNDARY\_ELEMENT\_METHOD\_OPTION**

**\*BOUNDARY\_FLUX\_OPTION**

**\*BOUNDARY\_NON\_REFLECTING**

**\*BOUNDARY\_NON\_REFLECTING\_2D**

**\*BOUNDARY\_PRESCRIBED\_MOTION\_OPTION**

**\*BOUNDARY\_PRESSURE\_OUTFLOW\_OPTION**

**\*BOUNDARY\_RADIATION\_OPTION**

**\*BOUNDARY\_SLIDING\_PLANE**

**\*BOUNDARY\_SPC\_OPTION**

**\*BOUNDARY\_SYMMETRY\_FAILURE**

**\*BOUNDARY\_TEMPERATURE\_OPTION**

**\*BOUNDARY\_USA\_SURFACE**

# \*BOUNDARY

---

## \*BOUNDARY\_CONVECTION\_OPTION

Available options are:

**SEGMENT**  
**SET**

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

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

### Card Format (Card 1 of 2)

Card 1            1            2            3            4            5            6            7            8

Variable	SSID								
Type	I								
Default	none								

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

### Card Format (Card 1 of 2)

Card 1            1            2            3            4            5            6            7            8

Variable	N1	N2	N3	N4					
Type	I	I	I	I					
Default	none	none	none	none					



Define the following card for both options:

**Card Format (Card 2 of 2)**

Card 1            1            2            3            4            5            6            7            8

Variable	HLCID	HMULT	TLCID	TMULT				
Type	I	F	I	F				
Default	none	1.0	none	1.0				

---

**VARIABLE**

---

**DESCRIPTION**

---

SSID	Segment set ID, see *SET_SEGMENT.
N1,N2...	Node ID's defining segment.
HLCID	Load curve ID for heat transfer coefficient, $h$ : GT.0: function versus time, EQ.0: use constant multiplier value, HMULT, LT.0: function versus temperature.
HMULT	Curve multiplier for $h$ .
TLCID	Load curve ID for $T_\infty$ versus time, see *DEFINE_CURVE: EQ.0: use constant multiplier value, TMULT.
TMULT	Curve multiplier for $T_\infty$

**Remarks:**

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

$h$                       heat transfer coefficient

$(T - T_\infty)$             temperature potential

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

# \*BOUNDARY

---

## \*BOUNDARY\_CYCLIC

Purpose: Define nodes in boundary planes for cyclic symmetry.

These boundary conditions can be used to model a segment of an object that has rotational symmetry such as an impeller, i.e., Figure 3.1. The segment 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 Format

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

---

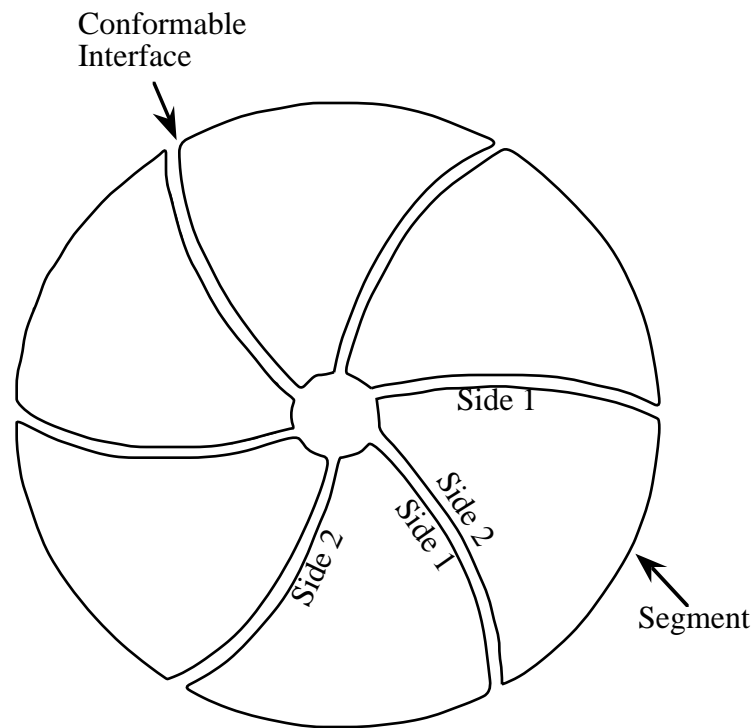
### VARIABLE

### DESCRIPTION

XC	x-component axis vector of axis of rotation
YC	y-component axis vector of axis of rotation
ZC	z-component axis vector of axis of rotation
NSID1	Node set ID for first boundary plane (side 1, see Figure 3.1).
NSID2	Node set ID for second boundary plane (side 2, see Figure 3.1). Each boundary node in this boundary plane is constrained to its corresponding node in the first node set. Node sets NSID1 and NSID2 must contain the same number of nodal points. Care has to be taken that the nodes in both node sets have a location which, if given in cylindrical coordinates, differ all by the same angle.

### **Remark:**

Only globally defined axes of rotation are possible.



**Figure 3.1** With cyclic symmetry only one segment is modeled.

## \*BOUNDARY

---

### \*BOUNDARY\_ELEMENT\_METHOD\_OPTION

Available options are:

**CONTROL**  
**FLOW**  
**NEIGHBOR**  
**WAKE**

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

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

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

The BOUNDARY\_ELEMENT\_METHOD\_FLOW card turns on the analysis, and is mandatory.

**\*BOUNDARY\_ELEMENT\_METHOD\_CONTROL**

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

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

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

**Define one card.**

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

# \*BOUNDARY

---

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

## **Remarks:**

1. Wakes convect with the free-stream velocity. The number of elements in the wake should be set to provide a total wake length equal to 5-10 times the characteristic streamwise length of the lifting surface to which the wake is attached. Note that each wake element has a streamwise length equal to the magnitude of the free stream velocity multiplied by the time increment between calls to the boundary element method routines. This time increment is controlled by DTBEM.
2. The most accurate results will be obtained with FARBEM set to 5 or more, while values as low as 2 will provides slightly reduced accuracy with a 50% reduction in the time required to compute the matrix of influence coefficients.

**\*BOUNDARY\_ELEMENT\_METHOD\_FLOW**

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

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

**Define one card.**

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

**VARIABLE****DESCRIPTION**

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

## \*BOUNDARY

---

### **Remarks:**

1. It is recommended that the shell segments in the SSID set use the NULL material (see \*MAT\_NULL). This will provide for the display of fluid pressures in the post-processor. For triangular shells the 4th node number should be the same as the 3rd node number. For fluid-structure interaction problems it is recommended that the boundary element shells use the same nodes and be coincident with the structural shell elements (or the outer face of solid elements) which define the surface of the body. This approach guarantees that the boundary element segments will move with the surface of the body as it deforms.
2. A pressure of PSTATIC is applied uniformly to all segments in the segment set. If the body of interest is hollow, then PSTATIC should be set to the free-stream static pressure minus the pressure on the inside of the body.
3. The effects of subsonic compressibility on gas flows can be included using a non-zero value for MACH. The pressures which arise from the fluid flow are increased using the Prandtl-Glauert compressibility correction. MACH should be set to zero for water or other liquid flows.



**\*BOUNDARY\_ELEMENT\_METHOD\_NEIGHBOR**

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

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

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

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

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

---

**VARIABLE**

---

**DESCRIPTION**

---

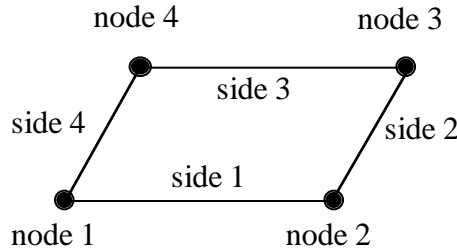
NELEM	Element number.
NABOR1	Neighbor for side 1 of NELEM.
NABOR2	Neighbor for side 2 of NELEM.
NABOR3	Neighbor for side 3 of NELEM.
NABOR4	Neighbor for side 4 of NELEM.

# \*BOUNDARY

---

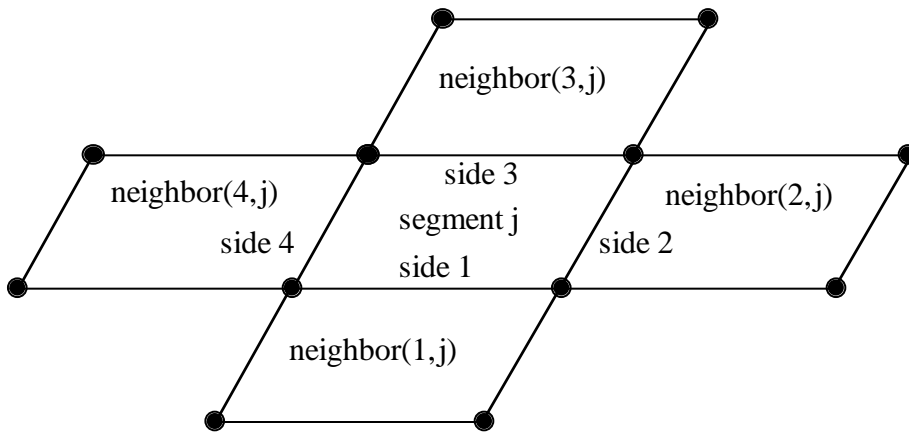
## Remarks:

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



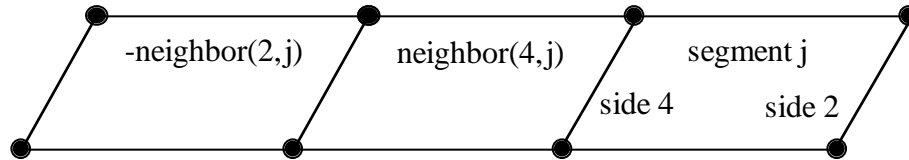
**Figure 3.2** Each segment has 4 sides.

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



**Figure 3.3** Typical neighbor specification.

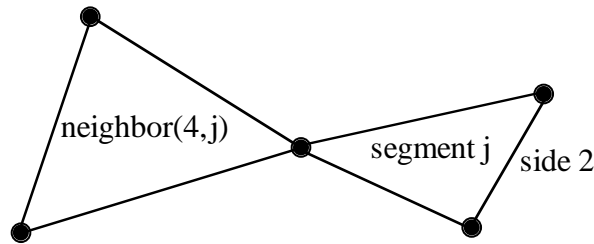
There are several situations for which the user may desire to directly specify the neighbor array for certain elements. For example, boundary element wakes result in discontinuous doublet distributions, and neighbors which cross a wake should not be used. Figure 3.4 illustrates a situation where a wake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



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

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

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



**Figure 3.5** Sometimes another triangular boundary element segment can be used as  $\text{neighbor}(4,j)$ .

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

# \*BOUNDARY

---

**Table 3.1** Surface pressure computation for element j.

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

**\*BOUNDARY\_ELEMENT\_METHOD\_WAKE**

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

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

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

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

---

**VARIABLE**

---

**DESCRIPTION**

NELEM

Element number to which a wake is attached.

NSIDE

The side of NELEM to which the wake is attached (see Fig. 3.2). This should be the "downstream" side of NELEM.

**Remarks:**

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

# \*BOUNDARY

---

## \*BOUNDARY\_FLUX\_OPTION

Available options are:

**SEGMENT**  
**SET**

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

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

### Card Format (Card 1 of 2)

Card 1            1            2            3            4            5            6            7            8

Variable	SSID								
Type	I								
Default	none								

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

### Card Format (Card 1 of 2)

Card 1            1            2            3            4            5            6            7            8

Variable	N1	N2	N3	N4					
Type	I	I	I	I					
Default	none	none	none	none					

Define the following card for both options:

**Card Format (Card 2 of 2)**

Card 1            1            2            3            4            5            6            7            8

Variable	LCID	MLC1	MLC2	MLC3	MLC4			
Type	I	F	F	F	F			
Default	none	1.0	1.0	1.0	1.0			

---

**VARIABLE**

---

**DESCRIPTION**

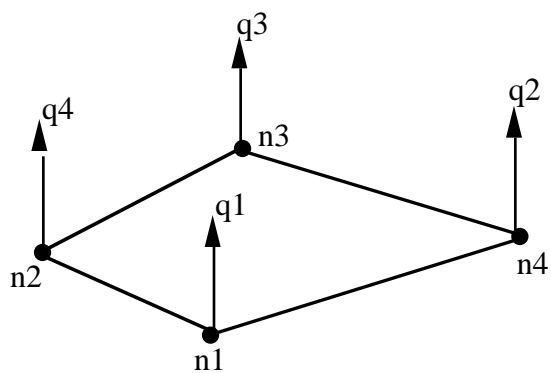
---

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

**Remarks:**

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

By convention, heat flow is negative in the direction of the surface outward normal vector. Surface definition is in accordance with the left hand rule. The outward normal vector points to the left as one progresses from node  $N_1$ - $N_2$ - $N_3$ - $N_4$ . See Figure 3.6.



**Figure 3.6.** Nodal number determines outward normal.



**\*BOUNDARY\_NON\_REFLECTING**

Purpose: Define a non-reflecting boundary. This option applies to continuum domains modeled with solid elements, as indefinite domains are usually not modelled. For geomechanical problems this option is important for limiting the size of the models.

**Card Format**

	1	2	3	4	5	6	7	8
Variable	SSID	AD	AS					
Type	I	F	F					
Default	none	0.0	0.0					
Remarks	1, 2	3	3					

---

**VARIABLE**

---

**DESCRIPTION**

SSID	Segment set ID, see *SET_SEGMENT.
AD	Default activation flag for dilatational waves. (on.eq.0.0, off.ne.0.0)
AS	Default activation flag for shear waves. (on.eq.0.0, off.ne.0.0)

**Remarks:**

1. Non-reflecting boundaries defined with this keyword are only used with three-dimensional solid elements. Boundaries are defined as a collection of segments, and segments are equivalent to element faces on the boundary. Segments are defined by listing the corner nodes in either a clockwise or counterclockwise order.
2. Non-reflecting boundaries are used on the exterior boundaries of an analysis model of an infinite domain, such as a half-space to prevent artificial stress wave reflections generated at the model boundaries from reentering the model and contaminating the results. Internally, LS-DYNA computes an impedance matching function for all non-reflecting boundary segments based on an assumption of linear material behavior. Thus, the finite element mesh should be constructed so that all significant nonlinear behavior is contained within the discrete analysis model.
3. With the two optional switches, the influence of reflecting waves can be studied.

# \*BOUNDARY

---

## \*BOUNDARY\_NON\_REFLECTING\_2D

Purpose: Define a non-reflecting boundary. This option applies to continuum domains modeled with two dimensional solid elements in the xy plane, as indefinite domains are usually not modelled. For geomechanical problems, this option is important for limiting the size of the models.

### Card Format

	1	2	3	4	5	6	7	8
Variable	NSID							
Type	I							
Default	none							
Remarks	1, 2							

---

#### VARIABLE

---

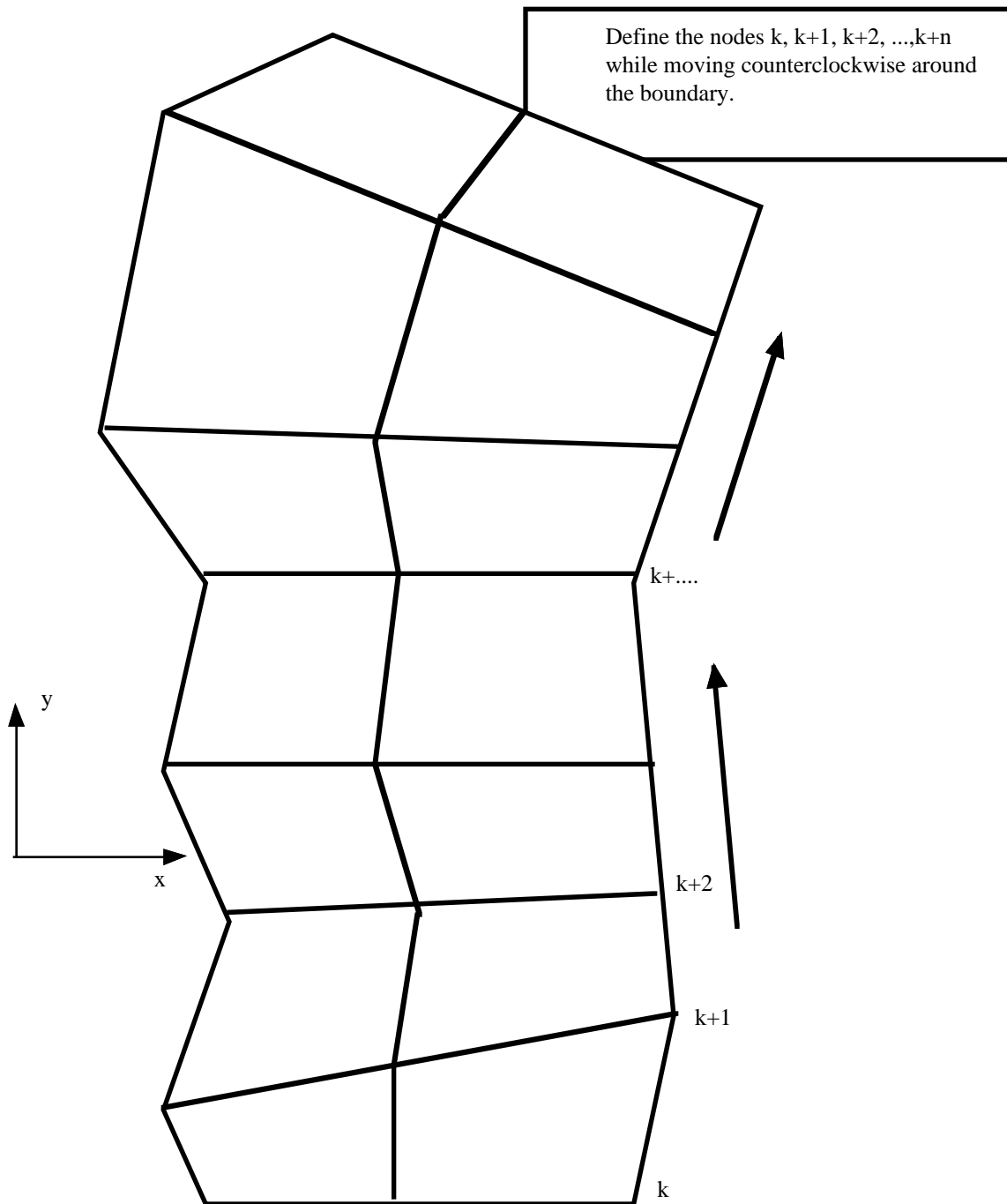
#### DESCRIPTION

NSID

Node set ID, see \*SET\_NODE. See Figure 3.7.

### **Remarks:**

1. Non-reflecting boundaries defined with this keyword are only used with two-dimensional solid elements in either plane strain or axisymmetric geometries. Boundaries are defined as a sequential string of nodes moving counterclockwise around the boundary.
2. Non-reflecting boundaries are used on the exterior boundaries of an analysis model of an infinite domain, such as a half-space to prevent artificial stress wave reflections generated at the model boundaries from reentering the model and contaminating the results. Internally, LS-DYNA computes an impedance matching function for all non-reflecting boundary segments based on an assumption of linear material behavior. Thus, the finite element mesh should be constructed so that all significant nonlinear behavior is contained within the discrete analysis model.



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

# \*BOUNDARY

---

## \*BOUNDARY\_PRESCRIBED\_MOTION\_OPTION

Available options include:

**NODE**  
**SET**  
**RIGID**  
**RIGID\_LOCAL**

Purpose: Define an imposed nodal motion (velocity, acceleration, or displacement) on a node or a set of nodes. Also velocities and displacements can be imposed on rigid bodies. If the local option is active the motion is prescribed with respect to the local coordinate system for the rigid body (See variable LCO for keyword \*MAT\_RIGID). Translational nodal velocity and acceleration specifications for rigid body nodes are allowed and are applied as described at the end of this section. For nodes on rigid bodies use the **NODE** option.

### Card Format

Card 1            1            2            3            4            5            6            7            8

Variable	<i>typeID</i>	DOF	VAD	LCID	SF	VID	DEATH	BIRTH
Type	I	I	I	I	F	I	F	F
Default	none	none	0	none	1.	0	1.E+28	0.0

**Card is required if DOF=9,10,11 on the first card or VAD=4. If DOF<9 and VAD<4, skip this card.**

Card 2            1            2            3            4            5            6            7            8

Variable	OFFSET1	OFFSET2	MRB	NODE1	NODE2			
Type	F	F	I	I	I			
Default	0.	0.	0	0	0			

---

<b>VARIABLE</b>	<b>DESCRIPTION</b>
NID, NSID, PID	Node ID (NID), nodal set ID (NSID), SEE *SET_NODE, or part ID (PID), see *PART, for a rigid body.
DOF	Applicable degrees-of-freedom: EQ. 1: x-translational degree-of-freedom, EQ. 2: y-translational degree-of-freedom, EQ. 3: z-translational degree-of-freedom, EQ. 4: translational motion in direction given by the VID. Movement on plane normal to the vector is permitted. EQ.-4: translational motion in direction given by the VID. Movement on plane normal to the vector is <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 motion about the vector given by the VID. Rotation about the normal axes is permitted. EQ.-8: rotational motion about the vector given by the VID. Rotation about the normal axes is <u>not</u> permitted. This option does not apply to rigid bodies. EQ. 9: y/z degrees-of-freedom for node rotating about the x-axis at location (OFFSET1,OFFSET2) in the yz-plane, <i>point</i> (y,z). Radial motion is NOT permitted. EQ.-9: y/z degrees-of-freedom for node rotating about the x-axis at location (OFFSET1,OFFSET2) in the yz-plane, <i>point</i> (y,z). Radial motion is permitted. EQ.10: z/x degrees-of-freedom for node rotating about the y-axis at location (OFFSET1,OFFSET2) in the zx-plane, <i>point</i> (z,x). Radial motion is NOT permitted. EQ.-10: z/x degrees-of-freedom for node rotating about the y-axis at location (OFFSET1,OFFSET2) in the zx-plane, <i>point</i> (z,x). Radial motion is permitted. EQ.11: x/y degrees-of-freedom for node rotating about the z-axis at location (OFFSET1,OFFSET2) in the xy-plane, <i>point</i> (x,y). Radial motion is NOT permitted. EQ.-11: x/y degrees-of-freedom for node rotating about the z-axis at location (OFFSET1,OFFSET2) in the xy-plane, <i>point</i> (x,y). Radial motion is permitted.
VAD	Velocity/Acceleration/Displacement flag: EQ.0: velocity (rigid bodies and nodes), EQ.1: acceleration (nodes only), EQ.2: displacement (rigid bodies and nodes). EQ.3: velocity versus displacement (rigid bodies) EQ.4: relative displacement (rigid bodies only)
LCID	Load curve ID to describe motion value versus time, see *DEFINE_CURVE.
SF	Load Curve Scale Factor

## \*BOUNDARY

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
VID	Vector ID for DOF values of 4 or 8, see *DEFINE_VECTOR.
DEATH	Time imposed motion/constraint is removed: EQ.0.0: default set to $10^{28}$ .
BIRTH	Time imposed motion/constraint is activated.
OFFSET1	Offset for DOF types 9-11 (y, z, x direction)
OFFSET2	Offset for DOF types 9-11 (z, x, y direction)
MRB	Master rigid body for measuring the relative displacement.
NODE1	Optional orientation node, n1, for relative displacement
NODE2	Optional orientation node, n2, for relative displacement

### Remarks:

Arbitrary translations and rotations are possible. Rotations around local axis can be defined either by setting DOF=8 or by using the offset option of DOF>8. The load curve scale factor can be used for simple modifications or unit adjustments.

The relative displacement can be measured in either of two ways:

1. Along a straight line between the mass centers of the rigid bodies,
2. Along a vector beginning at node n1 and terminating at node n2.

With option 1, a positive displacement will move the rigid bodies further apart, and, likewise a negative motion will move the rigid bodies closer together. The mass centers of the rigid bodies must not be coincident when this option is used. With option 2 the relative displacement is measured along the vector, and the rigid bodies may be coincident. Note that the motion of the master rigid body is not directly affected by this option, i.e., no forces are generated on the master rigid body.

The activation time, BIRTH, is the time during the solution that the constraint begins to act. Until this time, the prescribed motion card is ignored. The function value of the load curves will be evaluated at the offset time given by the difference of the solution time and BIRTH, i.e., (solution time-BIRTH). Relative displacements that occur prior to reaching BIRTH are ignored. Only relative displacements that occur after BIRTH are prescribed.

When the constrained node is on a rigid body, the translational motion is imposed without altering the angular velocity of the rigid body by calculating the appropriate translational velocity for the center of mass of the rigid body using the equation:

$$v_{cm} = v_{node} - \omega \times (x_{cm} - x_{node})$$

where  $v_{cm}$  is the velocity of the center of mass,  $v_{node}$  is the specified nodal velocity,  $\omega$  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. Extreme care must be used when prescribing motion of a rigid body

**\*BOUNDARY**

node. Typically, for nodes on a given rigid body, the motion of no more than one node should be prescribed or unexpected results may be obtained.

```

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

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

```

# \*BOUNDARY

---

## \*BOUNDARY\_PRESSURE\_OUTFLOW\_OPTION

Available options are

### SEGMENT SET

Purpose: Define pressure outflow boundary conditions. These boundary conditions are attached to solid elements using the eulerian ambient formulation (7) and defined to be pressure outflow ambient elements (3). See \*SECTION\_SOLID\_OPTION.

For the SET option define the following card

#### Card Format

Card 1            1            2            3            4            5            6            7            8

Variable	SSID								
Type	I								
Default	none								

For the SEGMENT option define the following card

#### Card Format

Card 1            1            2            3            4            5            6            7            8

Variable	N1	N2	N3	N4					
Type	I	I	I	I					
Default	none	none	none	none					



<u>VARIABLE</u>	<u>DESCRIPTION</u>
SSID	Segment set ID
N1,N2...	Node ID's defining segment

# \*BOUNDARY

---

## \*BOUNDARY\_RADIATION\_OPTION

Available options are:

**SEGMENT**  
**SET**

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

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

### Card Format (Card 1 of 2)

Card 1            1            2            3            4            5            6            7            8

Variable	SSID								
Type	I								
Default	none								

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

### Card Format (Card 1 of 2)

Card 1            1            2            3            4            5            6            7            8

Variable	N1	N2	N3	N4					
Type	I	I	I	I					
Default	none	none	none	none					

Define the following card for both options :

**Card Format (Card 2 of 2)**

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

<u>VARIABLE</u>	<u>DESCRIPTION</u>
SSID	Segment set ID, see *SET_SEGMENT
N1,N2...	Node ID's defining segment
RFLCID	Load curve ID for radiation factor $f$ , see *DEFINE_CURVE: GT.0: function versus time, EQ.0: use constant multiplier value, FMULT, LT.0: function versus temperature.
RFMULT	Curve multiplier for $f$ , see *DEFINE_CURVE
TILCID	Load curve ID for $T_\infty$ versus time, see *DEFINE_CURVE: EQ.0: use constant multiplier, TIMULT.
TIMULT	Curve multiplier for $T_\infty$

**Remarks:**

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

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

The exchange factor,  $F$ , is a characterization of the effect of the system geometry, emissivity and reflectivity on the capability of radiative transport between surfaces. The radiation boundary condition data cards require specification of the product,  $f = F\sigma$ , and  $T_\infty$  for the boundary surface.

# \*BOUNDARY

---

## \*BOUNDARY\_SLIDING\_PLANE

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

### Card Format

	1	2	3	4	5	6	7	8
Variable	NSID	VX	VY	VZ	COPT			
Type	I	F	F	F	I			
Default	none	0	0	0	0			

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
-----------------	--------------------

---

NSID	Nodal set ID, see *SET_NODE
VX	x-coordinate of vector defining normal or vector
VY	y-coordinate of vector defining normal or vector
VZ	z-coordinate of vector defining normal or vector
COPT	Option: EQ.0: node moves on normal plane, EQ.1: node moves only in vector direction.

### Remarks:

Any node may be constrained to move on an arbitrarily oriented plane or line depending on the choice of COPT. Each boundary condition card defines a vector originating at (0,0,0) and terminating at the coordinates defined above. Since an arbitrary magnitude is assumed for this vector, the specified coordinates are non-unique and define only a direction.

**\*BOUNDARY\_SPC\_OPTION**

Available options include:

**NODE  
SET**

Purpose: Define nodal single point constraints.

**Card Format**

	1	2	3	4	5	6	7	8
Variable	NID/NSID	CID	DOFX	DOFY	DOFZ	DOFRX	DOFRY	DOFRZ
Type	I	I	I	I	I	I	I	I
Default	none	0	0	0	0	0	0	0

**VARIABLE****DESCRIPTION**

NID/NSID	Node ID or nodal set ID, see *SET_NODE.
CID	Coordinate system ID, see *DEFINE_COORDINATE_SYSTEM.
DOFX	Insert 1 for translational constraint in local x-direction.
DOFY	Insert 1 for translational constraint in local y-direction.
DOFZ	Insert 1 for translational constraint in local z-direction.
DOFRX	Insert 1 for rotational constraint about local x-axis.
DOFRY	Insert 1 for rotational constraint about local y-axis.
DOFRZ	Insert 1 for rotational constraint about local z-axis.

**Remark:**

Constraints are applied if a value of 1 is given for DOFxx. A value of zero means no constraint.

# \*BOUNDARY

---

```
$  
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$  
$  
$$$ *BOUNDARY_SPC_NODE  
$  
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$  
$  
$ Make boundary constraints for nodes 6 and 542.  
$  
*BOUNDARY_SPC_NODE  
$  
$.>...1.>...2.>...3.>...4.>...5.>...6.>...7.>...8  
$ nid cid dofx dofy dofz dofrx dofry dofrz  
6 0 1 1 1 1 1 1  
542 0 0 1 0 1 0 1  
$  
$ Node 6 is fixed in all six degrees of freedom (no motion allowed).  
$  
$ Node 542 has a symmetry condition constraint in the x-z plane,  
$ no motion allowed for y translation, and x & z rotation.  
$  
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$  
$
```

**\*BOUNDARY\_SYMMETRY\_FAILURE**

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

**Card Format**

	1	2	3	4	5	6	7	8
Variable	SSID	FS	VTX	VTY	VTZ	VHX	VHY	VHZ
Type	I	F	F	F	F	F	F	F
Default	none	0	0	0	0	0	0	0

<u>VARIABLE</u>	<u>DESCRIPTION</u>
SSID	Segment set ID, see *SET_SEGMENT
FS	Tensile failure stress > 0.0. The average stress in the elements surrounding the boundary nodes in a direction perpendicular to the boundary is used.
VTX	x-coordinate of tail of a normal vector originating on the wall (tail) and terminating in the body (head) (i.e., vector points from the symmetry plane into the body).
VTY	y-coordinate of tail
VTZ	z-coordinate of tail
VHX	x-coordinate of head
VHY	y-coordinate of head
VHZ	z-coordinate of head

**Remarks:**

A plane of symmetry is assumed for the nodes on the boundary at the tail of the vector given above. Only the motion perpendicular to the symmetry plane is constrained. After failure the nodes are set free.

# \*BOUNDARY

---

## \*BOUNDARY\_TEMPERATURE\_OPTION

Available options are:

**NODE**  
**SET**

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

### Card Format

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

<u>VARIABLE</u>	<u>DESCRIPTION</u>
NID/SID	Node ID/Node Set ID, see *SET_NODE_OPTION
LCID	Load curve ID for temperature versus time: EQ.0: use the constant multiplier value given below by CMULT.
CMULT	Curve multiplier for temperature

### Remarks:

If no load curve ID is given, then a constant boundary temperature is assumed. CMULT is also used to scale the load curve values.



**\*BOUNDARY\_USA\_SURFACE**

Purpose: Define a surface for coupling with the USA boundary element code [DeRuntz, 1993]. The outward normal vectors should point into the fluid media.

**Card Format**

	1	2	3	4	5	6	7	8
Variable	SSID	WETDRY	NBEAM					
Type	I	I	I					
Default	none	0	0					

<u>VARIABLE</u>	<u>DESCRIPTION</u>
SSID	Segment set ID, see *SET_SEGMENT
WETDRY	Wet surface flag: EQ.0: dry, no coupling, EQ.1: wet, coupled with USA.
NBEAM	The number of nodes touched by USA Surface-of-Revolution (SOR) elements. It is not necessary that the LS-DYNA model has beams where USA has beams (i.e., SOR elements), merely that the LS-DYNA model has nodes to receive the forces that USA will return.

**Remarks:**

The wet surface of 3 and 4-noded USA General boundary elements is defined in LS-DYNA with a segment set of 4-noded surface segments, where the fourth node can duplicate the third node to form a triangle. The segment normals should be directed into the USA fluid. If USA overlays are going to be used to reduce the size of the DAA matrices, the user should nonetheless define the wet surface here as if no overlay were being used. If Surface-of-Revolution elements (SORs) are being used in USA, then NBEAM should be non zero on one and only one card in this section.

When running a coupled problem with USA, the procedure involves several steps. First, LS-DYNA is executed to create a LS-DYNA dump file "d3dump" and a linking file "strnam" which contains the nodal grid point data and wet segment connectivity data for the FLUMAS processor, and the dof-equation table and structural mass vector for the AUGMAT processor. "Dyna.pre" is denoted "grdnam" in the FLUMAS manual and "strnam" in the AUGMAT manual. The execution line in the first step is:

## \*BOUNDARY

---

**LS-DYNA** memory=nwds i=inputfilename > outputfilename

where "inputfilename" is the LS-DYNA input file.

In the second step, the DAA fluid mass matrix is created through execution of the USA FLUMAS processor:

**FLUMAS** -m nwds < flumasinputfilename > flumasoutputfilename

In the third step, the modified augmented DAA equations for the coupled problem are calculated and saved through execution of the USA AUGMAT processor:

**AUGMAT** -m nwds < augmatinputfilename > augmatoutputfilename

This step is repeated whenever one wishes to change DAA formulations.

In the fourth step the actual coupled time-integration is conducted using the execution line:

**LS-DYNA** memory=nwds r=d3dump usa=usainputfilename > outputfilename

The input files, flumasinputfilename, augmatinputfilename, and usainputfilename, are prepared in accordance with the USA code documentation.

It is advisable when running coupled problems to check the ASCII output files to ensure that each run completed normally.

# **\*COMPONENT**

The keyword **\*COMPONENT** provides a way of incorporating specialized components and features. The keyword control cards in this section are defined in alphabetical order:

**\*COMPONENT\_GEBOD\_OPTION**

**\*COMPONENT\_GEBOD\_JOINT\_OPTION**

# \*COMPONENT

---

## \*COMPONENT\_GEBOD\_OPTION

Purpose: Generate a rigid body dummy based on dimensions and mass properties from the GEBOD database. The motion of the dummy is governed by equations integrated within LS-DYNA separately from the finite element model. Default joint characteristics (stiffnesses, stop angles, etc.) are set internally and should give reasonable results, however, they may be altered using the \*COMPONENT\_GEBOD\_JOINT command. Contact between the segments of the dummy and the finite element model is defined using the \*CONTACT\_GEBOD command. See Appendix K for further detail on these dummies. NOTE: This feature is available only in keyword input format.

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

**MALE**  
**FEMALE**  
**CHILD**

### Card Format (Card 1 of 2)

1            2            3            4            5            6            7            8

Variable	DID	UNITS	SIZE					
Type	I	I	F					
Default	none	none	none					

#### VARIABLE

#### DESCRIPTION

DID            Dummy ID. Unique numbers must be specified.

UNITS            System of units used in the finite element model.  
 EQ.1: lbf\*sec<sup>2</sup>/in - inch - sec  
 EQ.2: kg - meter - sec  
 EQ.3: kgf\*sec<sup>2</sup>/mm - mm - sec  
 EQ.4: metric ton - mm - sec  
 EQ.5: kg - mm - msec

SIZE            Size of the dummy. This represents a combined height and weight percentile ranging from 0 to 100 for the male and female types. For the child the number of months of age is input with an admissible range from 24 to 240.

# \*COMPONENT

## Card Format (Card 2 of 2)

Card 1                    1                    2                    3                    4                    5                    6                    7                    8

Variable	VX	VY	VZ	GX	GY	GZ		
Type	F	F	F	F	F	F		
Default	0.	0.	0.	0.	0.	0.		

### VARIABLE

### DESCRIPTION

VX,VY,VZ

Initial velocity of the dummy in the global x, y and z directions.

GX,GY,GZ

Global x,y, and z components of gravitational acceleration applied to the dummy.

```

$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$ *COMPONENT_GEBOD_MALE
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$ A 50th percentile male dummy with the ID number of 7 is generated in the
$ lbf*sec^2-inch-sec system of units. The dummy is given an initial velocity of
$ 616 in/sec in the negative x direction and gravity acts in the negative z
$ direction with a value 386 in/sec^2.
$
*COMPONENT_GEBOD_MADE
$
$.>....1.>....2.>....3.>....4.>....5.>....6.>....7.>....8
$   did   units   size
$     7     1     50
$   vx    vy    vz    gx    gy    gz
$  -616    0     0     0     0   -386
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$

```

# \*COMPONENT

---

## \*COMPONENT\_GEBOD\_JOINT\_OPTION

Purpose : Alter the joint characteristics of a GEBOD rigid body dummy. Setting a joint parameter value to zero retains the default value set internally. See Appendix K for further details.

The following options are available.

- PELVIS
- WAIST
- LOWER\_NECK
- UPPER\_NECK
- LEFT\_SHOULDER
- RIGHT\_SHOULDER
- LEFT\_ELBOW
- RIGHT\_ELBOW
- LEFT\_HIP
- RIGHT\_HIP
- LEFT\_KNEE
- RIGHT\_KNEE
- LEFT\_ANKLE
- RIGHT\_ANKLE

### Card 1 - Required.

	1	2	3	4	5	6	7	8
Variable	DID	LC1	LC2	LC3	SCF1	SCF2	SCF3	
Type	F	I	I	I	F	F	F	

<u>VARIABLE</u>	<u>DESCRIPTION</u>
DID	Dummy ID, see *COMPONENT_GEBOD_OPTION.
LCi	Load curve ID specifying the loading torque versus rotation (in radians) for the i-th degree of freedom of the joint.
SCFi	Scale factor applied to the load curve of the i-th joint degree of freedom.

# \*COMPONENT

## Card 2 - Required.

	1	2	3	4	5	6	7	8
Variable	C1	C2	C3	NEUT1	NEUT2	NEUT3		
Type	F	F	F	F	F	F		

### VARIABLE

### DESCRIPTION

Ci	Linear viscous damping coefficient applied to the i-th DOF of the joint. Units are torque*time/radian, where the units of torque and time depend on the choice of UNITS in card 1 of *COMPONENT_GEBOD_OPTION.
NEUTi	Neutral angle (degrees) of joint's i-th DOF.

## Card 3 - Required.

	1	2	3	4	5	6	7	8
Variable	LOSA1	HISA1	LOSA2	HISA2	LOSA3	HISA3		
Type	F	F	F	F	F	F		

### VARIABLE

### DESCRIPTION

LOSAi	Value of the low stop angle (degrees) for the i-th DOF of this joint.
HISAi	Value of the high stop angle (degrees) for the i-th DOF of this joint.

# \*COMPONENT

---

## Card 4 - Required.

1            2            3            4            5            6            7            8

Variable	UNK1	UNK2	UNK3					
Type	F	F	F					
Default	0.	0.	0.					

---

**VARIABLE**

---

**DESCRIPTION**

---

UNKi

Unloading stiffness (torque/radian) for the i-th degree of freedom of the joint. This must be a positive number. Units of torque depend on the choice of UNITS in card 1 of \*COMPONENT\_GEBOD\_OPTION.





# \*COMPONENT

---

# **\*CONSTRAINED**

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

- \*CONSTRAINED\_EXTRA\_NODES\_OPTION**
- \*CONSTRAINED\_GENERALIZED\_WELD\_OPTION**
- \*CONSTRAINED\_GLOBAL**
- \*CONSTRAINED\_JOINT\_OPTION**
- \*CONSTRAINED\_JOINT\_STIFFNESS\_OPTION**
- \*CONSTRAINED\_LAGRANGE\_IN\_SOLID**
- \*CONSTRAINED\_LINEAR**
- \*CONSTRAINED\_NODAL\_RIGID\_BODY\_{OPTION}**
- \*CONSTRAINED\_NODE\_SET**
- \*CONSTRAINED\_RIGID\_BODIES**
- \*CONSTRAINED\_RIGID\_BODY\_STOPPERS**
- \*CONSTRAINED\_RIVET**
- \*CONSTRAINED\_SHELL\_TO\_SOLID**
- \*CONSTRAINED\_SPOTWELD\_{OPTION}**
- \*CONSTRAINED\_TIE-BREAK**
- \*CONSTRAINED\_TIED\_NODES\_FAILURE**

# \*CONSTRAINED

---

## \*CONSTRAINED\_EXTRA\_NODES\_OPTION

Available options include:

**NODE**

**SET**

Purpose: Define extra nodes for rigid body.

### Card Format

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

---

#### VARIABLE

#### DESCRIPTION

PID	Part ID of rigid body to which the nodes will be added, see *PART.
NID/NSID	Node (option: _NODE) or node set ID (option: _SET), see *SET_NODE, of added nodes.

### **Remarks:**

Extra nodes for rigid bodies may be placed anywhere, even outside the body, and they are assumed to be part of the rigid body. They have many uses including:

1. The definition of draw beads in metal forming applications by listing nodes along the draw bead.
2. Placing nodes where joints will be attached between rigid bodies.
3. Defining a nodes where point loads are to be applied or where springs may be attached.
4. Defining a lumped mass at a particular location.

and so on. The coordinates of the extra nodes are updated according to the rigid body motion.

```

$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$ *CONSTRAINED_EXTRA_NODES_NODE
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$ Rigidly attach nodes 285 and 4576 to part 14. (Part 14 MUST be a rigid body.)
$
*CONSTRAINED_EXTRA_NODES_NODE
$
$...>...1...>...2...>...3...>...4...>...5...>...6...>...7...>...8
$ pid nid
$ 14 285
$ 14 4576
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$

```

```

$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$ *CONSTRAINED_EXTRA_NODES_SET
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$ Rigidly attach all nodes in set 4 to part 17. (Part 17 MUST be a rigid body.)
$
$ In this example, four nodes from a deformable body are attached
$ to rigid body 17 as a means of joining the two parts.
$
*CONSTRAINED_EXTRA_NODES_SET
$
$...>...1...>...2...>...3...>...4...>...5...>...6...>...7...>...8
$ pid nsid
$ 17 4
$
$
*SET_NODE_LIST
$ sid
$ 4
$ nid1 nid2 nid3 nid4 nid5 nid6 nid7 nid8
$ 665 778 896 827
$
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$

```

# \*CONSTRAINED

---

## \*CONSTRAINED\_GENERALIZED\_WELD\_OPTION

Then the following options are available:

- SPOT**
- FILLET**
- BUTT**
- CROSS\_FILLET**
- COMBINED**

Purpose: Define spot and fillet welds. Coincident nodes are permitted if the local coordinate ID is defined. For the spot weld a local coordinate ID is not required if the nodes are offset. Failures can include both the plastic and brittle failures. These can be used either independently or together. Failure occurs when either criteria is met. The welds may undergo large rotations since the equations of rigid body mechanics are used to update their motion.

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

	1	2	3	4	5	6	7	8
Variable	NSID	CID	FILTER	WINDOW	NPR	NPRT		
Type	I	I	I	E	I	I		
Default	none	none						

### VARIABLE

### DESCRIPTION

- NSID            Nodal set ID, see \*SET\_NODE\_OPTION.
- CID            Coordinate system ID for output of data in local system, see \*DEFINE\_COORDINATE\_OPTION. CID is not required for spotwelds if the nodes are not coincident.

---

<b>VARIABLE</b>	<b>DESCRIPTION</b>
FILTER	<p>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.</p> <p>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.</p>
WINDOW	<p>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.</p> <p>EQ.0: time window is not used</p>
NPR	<p>NFW, number of individual nodal pairs in the cross fillet or combined general weld.</p> <p>and general welds.</p>
NPRT	<p>Print option in file RBDOUT.</p> <p>EQ.0: default from Control Card is used EQ.1: data is printed EQ.2: data is not printed</p>

# \*CONSTRAINED

**Additional Card required for the CONSTRAINED\_GENERALIZED\_WELD\_SPOT option:**

Card 2            1            2            3            4            5            6            7            8

Variable	TFAIL	EPSF	SN	SS	N	M		
Type	F	F	F	F	F	F		

**VARIABLE**

**DESCRIPTION**

TFAIL	Failure time for constraint set, $t_f$ . (default=1.E+20)
EPSF	Effective plastic strain at failure, $\epsilon_{fail}^p$ defines ductile failure.
SN	$S_n$ , normal force at failure, only for the brittle failure of spotwelds.
SS	$S_s$ , shear force at failure, only for the brittle failure of spotwelds.
N	n, exponent for normal force, only for the brittle failure of spotwelds.
M	m, exponent for shear force, only for the brittle failure of spotwelds.

**Remarks:**

Spotweld failure due to plastic straining occurs when the effective nodal plastic strain exceeds the input value,  $\epsilon_{fail}^p$ . This option can model the tearing out of a spotweld from the sheet metal since the plasticity is in the material that surrounds the spotweld, not the spotweld itself. A least squares algorithm is used to generate the nodal values of plastic strains at the nodes from the element integration point values. The plastic strain is integrated through the element and the average value is projected to the nodes via a least square fit. This option should only be used for the material models related to metallic plasticity and can result in slightly increased run times.

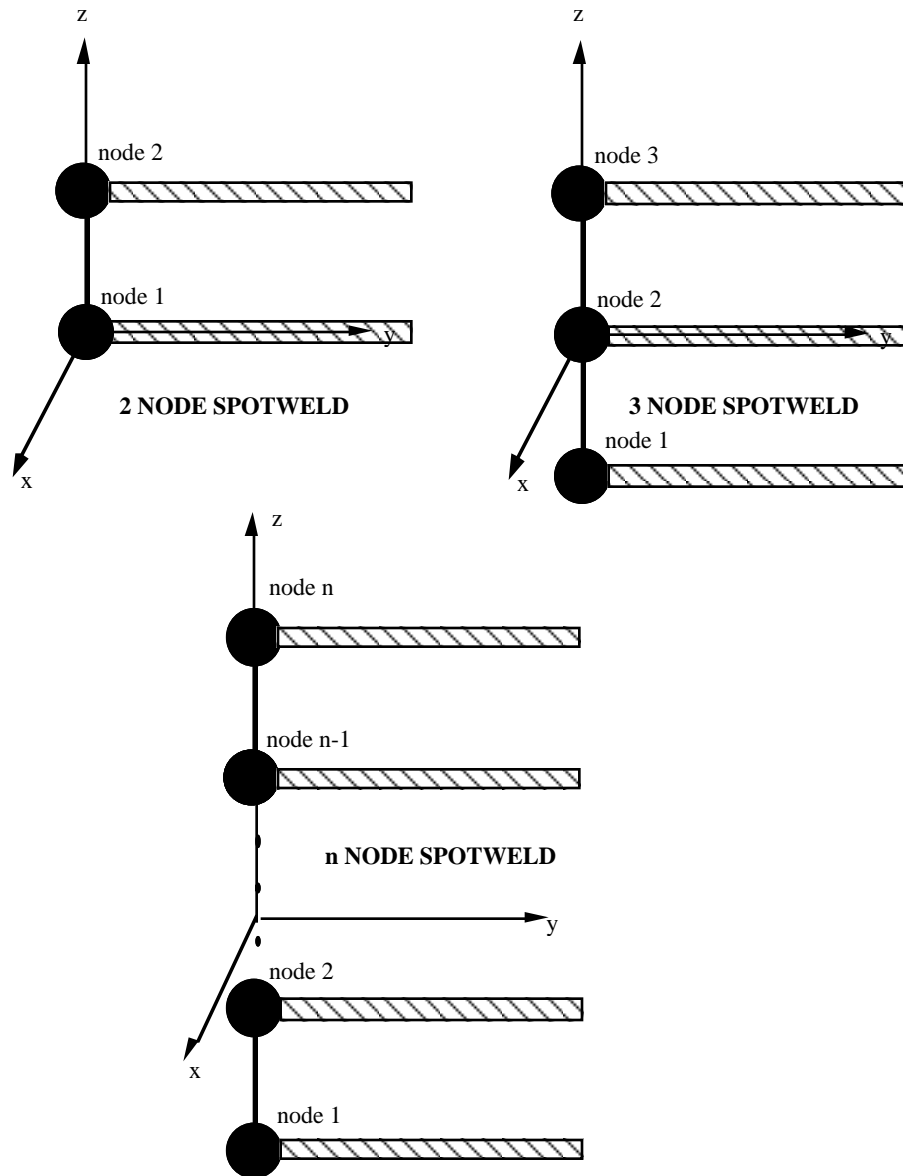
Brittle failure of the spotwelds occurs when:

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

where  $f_n$  and  $f_s$  are the normal and shear interface force. Component  $f_n$  contributes for tensile values only. When the failure time,  $t_f$ , is reached the nodal rigid body becomes inactive and the constrained nodes may move freely. In Figure 5.1 the ordering of the nodes is shown for the 2 node and 3 node spotwelds. This order is with respect to the local coordinate system where the local z axis determines the tensile direction. The nodes in the spotweld may coincide. The failure of the 3 node spotweld may occur gradually with first one node failing and later the second node may fail. For n noded spotwelds the failure is progressive starting with the outer nodes (1 and n) and then moving inward



to nodes 2 and n-1. Progressive failure is necessary to preclude failures that would create new rigid bodies.



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

# \*CONSTRAINED

## Additional Card required for the FILLET option:

Card 2	1	2	3	4	5	6	7	8
Variable	TFAIL	EPSF	SIGY	BETA	L	W	A	ALPHA
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
TFAIL	Failure time for constraint set, $t_f$ . (default=1.E+20)
EPSF	Effective plastic strain at failure, $\epsilon_{fail}^p$ defines ductile failure.
SIGY	$\sigma_f$ , stress at failure for brittle failure.
BETA	$\beta$ , failure parameter for brittle failure.
L	L, length of fillet/butt weld (see Figure 5.2 and 5.3).
W	w, width of flange (see Figure 5.2).
A	a, width of fillet weld (see Figure 5.2).
ALPHA	$\alpha$ , weld angle (see Figure 5.2) in degrees.

### Remarks:

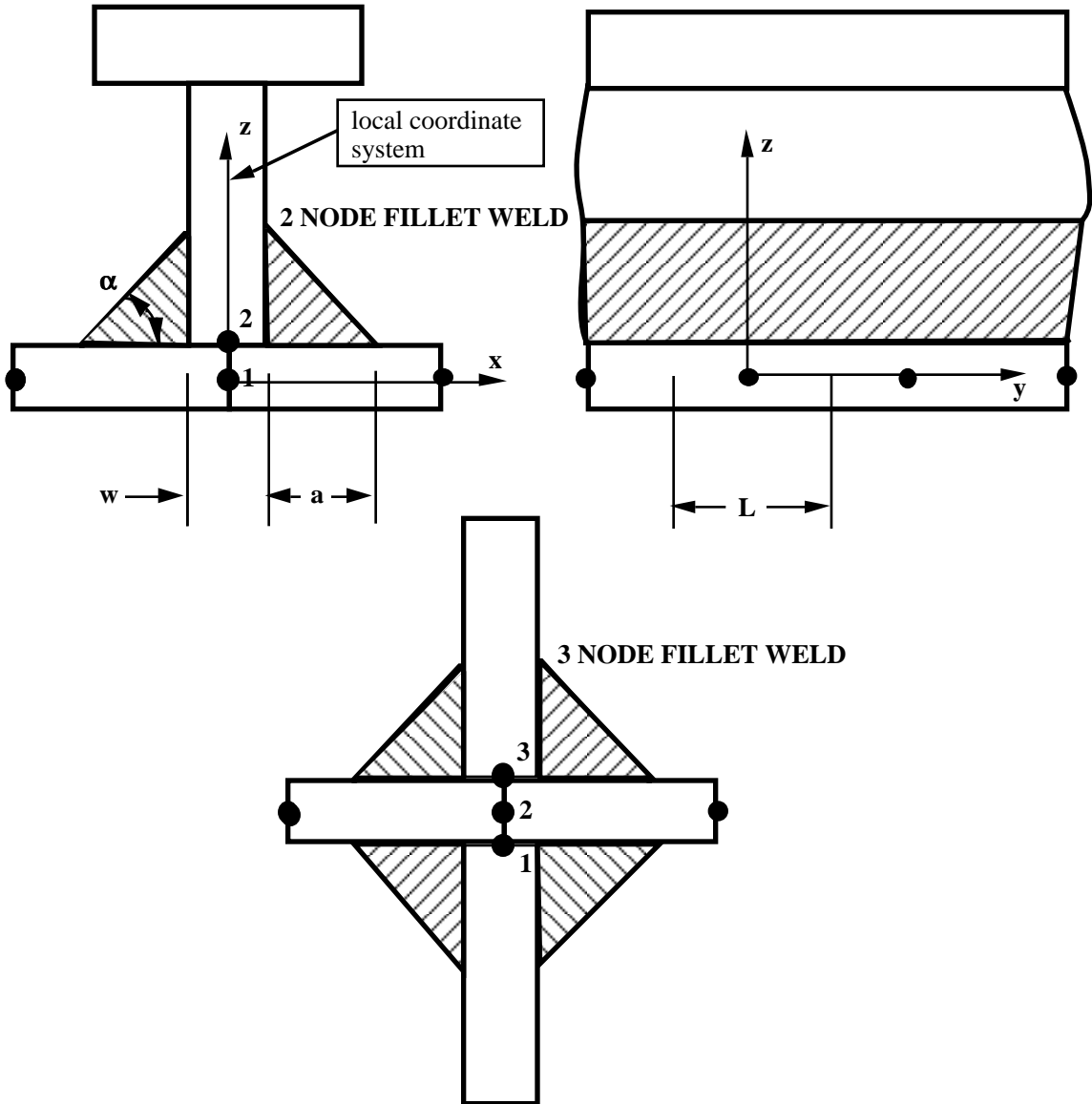
Ductile fillet weld failure, due to plastic straining, is treated identically to spotweld failure. Brittle failure of the fillet welds occurs when:

$$\beta \sqrt{\sigma_n^2 + 3(\tau_n^2 + \tau_t^2)} \geq \sigma_f$$

where

- $\sigma_n$  = normal stress
- $\tau_n$  = shear stress in direction of weld (local y)
- $\tau_t$  = shear stress normal to weld (local x)
- $\sigma_f$  = failure stress
- $\beta$  = failure parameter

Component  $\sigma_n$  is nonzero for tensile values only. When the failure time,  $t_f$ , is reached the nodal rigid body becomes inactive and the constrained nodes may move freely. In Figure 5.2 the ordering of the nodes is shown for the 2 node and 3 node fillet welds. This order is with respect to the local coordinate system where the local z axis determines the tensile direction. The nodes in the fillet weld may coincide. The failure of the 3 node fillet weld may occur gradually with first one node failing and later the second node may fail.



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

# \*CONSTRAINED

---

## Additional Card required for the BUTT option:

Card 2	1	2	3	4	5	6	7	8
Variable	TFAIL	EPSF	SIGY	BETA	L	D	LT	
Type	F	F	F	F	F	F	F	

<u>VARIABLE</u>	<u>DESCRIPTION</u>
TFAIL	Failure time for constraint set, $t_f$ . (default=1.E+20)
EPSF	Effective plastic strain at failure, $\epsilon_{fail}^p$ defines ductile failure.
SIGY	$\sigma_f$ , stress at failure for brittle failure.
BETA	$\beta$ , failure parameter for brittle failure.
L	L, length of fillet/butt weld (see Figure 5.2 and 5.3).
D	d, thickness of butt weld (see Figure 5.3).
LT	$L_t$ , transverse length of butt weld (see Figure 5.3).

### Remarks:

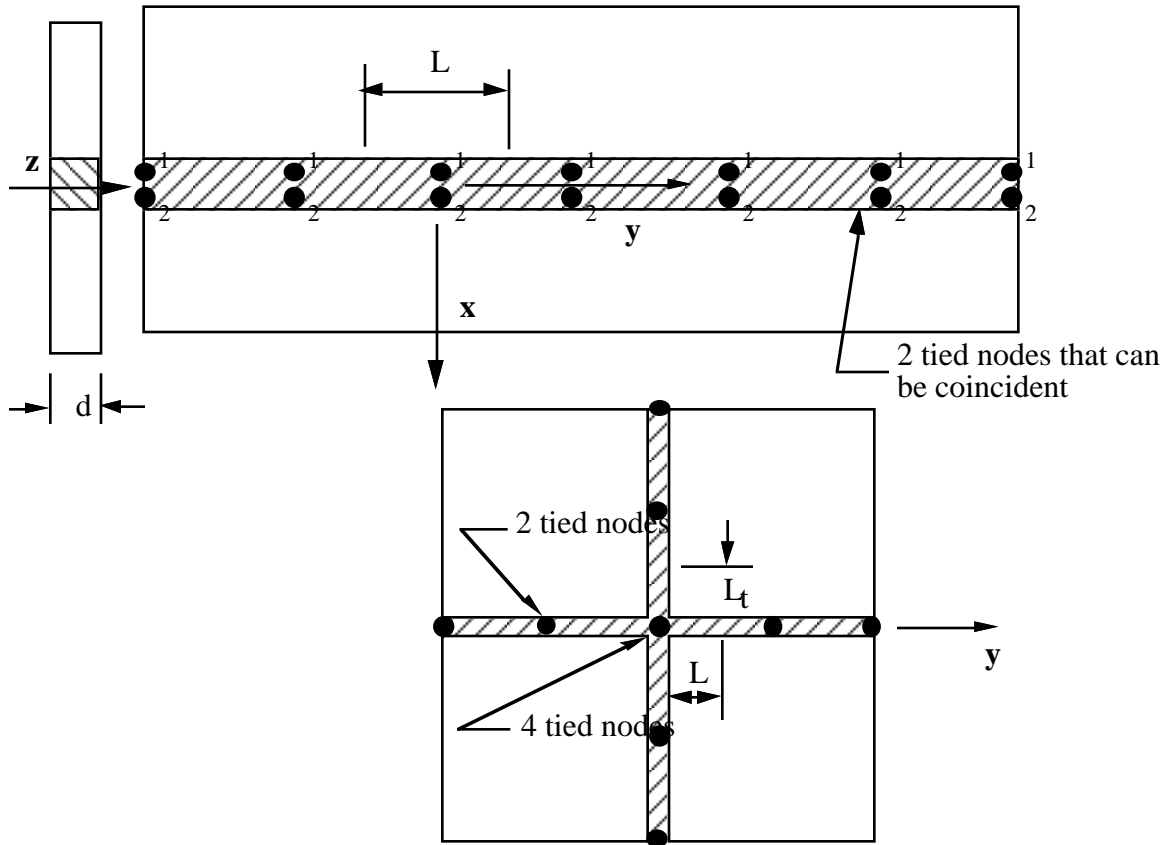
Ductile butt weld failure, due to plastic straining, is treated identically to spotweld failure. Brittle failure of the butt welds occurs when:

$$\beta \sqrt{\sigma_n^2 + 3(\tau_n^2 + \tau_t^2)} \geq \sigma_f$$

where

- $\sigma_n$  = normal stress
- $\tau_n$  = shear stress in direction of weld (local y)
- $\tau_t$  = shear stress normal to weld (local z)
- $\sigma_f$  = failure stress
- $\beta$  = failure parameter

Component  $\sigma_n$  is nonzero for tensile values only. When the failure time,  $t_f$ , is reached the nodal rigid body becomes inactive and the constrained nodes may move freely. The nodes in the butt weld may coincide.



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

# \*CONSTRAINED

---

```

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

```

## \*CONSTRAINED

### Additional Cards (1+NPR) required for the CROSS\_FILLET option:

Card 2            1            2            3            4            5            6            7            8

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

Cards 3,4,  
...,2+NPR

Variable	NODEA	NODEB	NCID					
Type	I	I	I					

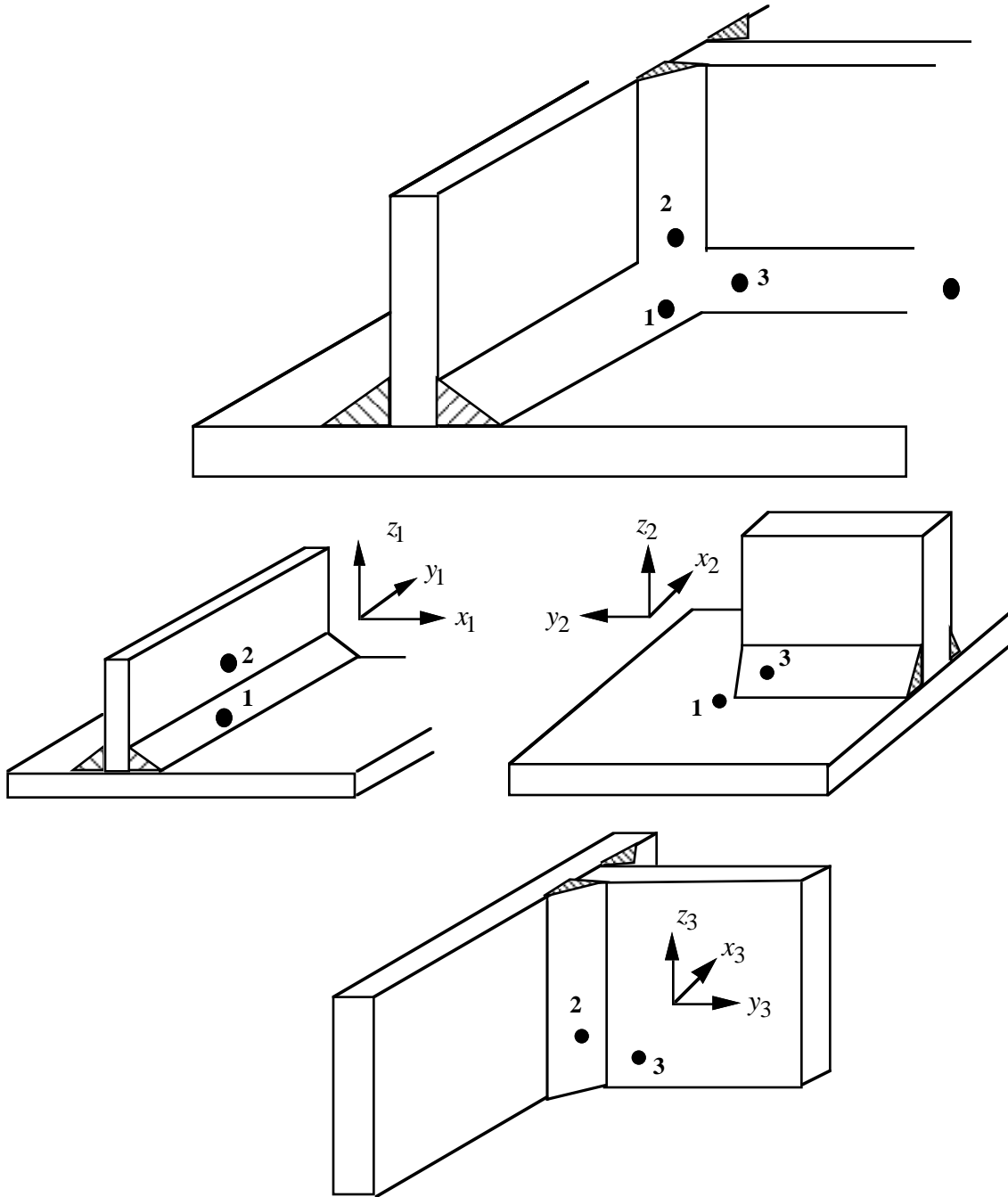
#### VARIABLE

#### DESCRIPTION

TFAIL	Failure time for constraint set, $t_f$ . (default=1.E+20)
EPSF	Effective plastic strain at failure, $\epsilon_{fail}^p$ defines ductile failure.
SIGY	$\sigma_f$ , stress at failure for brittle failure.
BETA	$\beta$ , failure parameter for brittle failure.
L	L, length of fillet/butt weld (see Figure 5.2 and 5.3).
W	w, width of flange (see Figure 5.2).
A	a, width of fillet weld (see Figure 5.2).
ALPHA	$\alpha$ , weld angle (see Figure 5.2) in degrees.
NODEA	Node ID, A, in weld pair (CROSS or COMBINED option only). See Figure 5.4.
NODEB	Node ID, B, in weld pair (CROSS or COMBINED option only).
NCID	Local coordinate system ID (CROSS or COMBINED option only).

# \*CONSTRAINED

---



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



## \*CONSTRAINED

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

Card 2            1            2            3            4            5            6            7            8

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

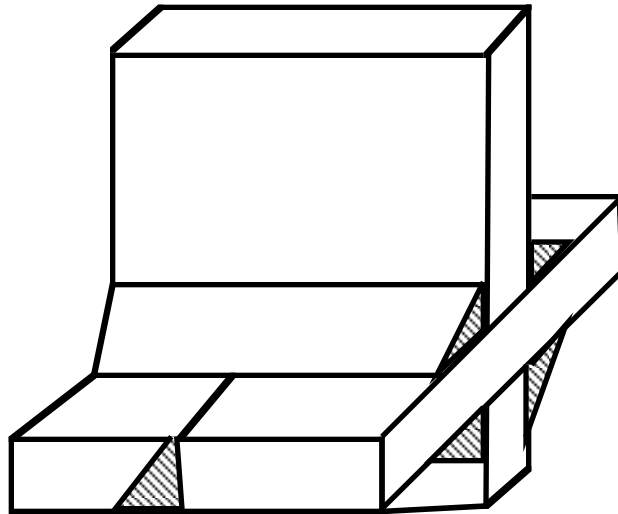
Card 3

Variable	NODEA	NODEB	NCID	WTYP				
Type	I	I	I	I				

**VARIABLE**

**DESCRIPTION**

TFAIL	Failure time for constraint set, $t_f$ . (default=1.E+20)
EPSF	Effective plastic strain at failure, $\epsilon_{fail}^p$ defines ductile failure.
SIGY	$\sigma_f$ , stress at failure for brittle failure.
BETA	$\beta$ , failure parameter for brittle failure.
L	L, length of fillet/butt weld (see Figure 5.2 and 5.3).
W	w, width of flange (see Figure 5.2).
A	a, width of fillet weld (see Figure 5.2).
ALPHA	$\alpha$ , weld angle (see Figure 5.2) in degrees.
NODEA	Node ID, A, in weld pair (CROSS or COMBINED option only).
NODEB	Node ID, B, in weld pair (CROSS or COMBINED option only).
NCID	Local coordinate system ID (CROSS or COMBINED option only).
WTYP	Weld pair type (GENERAL option only). See Figure 5.5. EQ.0: fillet weld EQ.1: butt weld



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

**\*CONSTRAINED\_GLOBAL**

Purpose: Define a global boundary constraint plane.

**Card Format**

	1	2	3	4	5	6	7	8
Variable	TC	RC	DIR	X	Y	Z		
Type	I	I	I	F	F	F		
Default	0	0	0	0	0	0		

---

**VARIABLE**

---

**DESCRIPTION**

TC	Translational Constraint: EQ. 1: constrained x translation, EQ. 2: constrained y translation, EQ. 3: constrained z translation, EQ. 4: constrained x and y translations, EQ. 5: constrained y and z translations, EQ. 6: constrained x and z translations, EQ. 7: constrained x, y, and z translations,
RC	Rotational Constraint: EQ. 1: constrained x-rotation, EQ. 2: constrained y-rotation, EQ. 3: constrained z-rotation, EQ. 4: constrained x and y rotations, EQ. 5: constrained y and z rotations, EQ. 6: constrained z and x rotations, EQ. 7: constrained x, y, and z rotations.
DIR	Direction of normal EQ. 1: global x, EQ. 2: global y, EQ. 3: global z.
X	x-offset coordinate
Y	y-offset coordinate
Z	z-offset coordinate

## \*CONSTRAINED

---

### **Remarks:**

Nodes within a tolerance are constrained on a global plane. This option is recommended for use with r-method adaptive remeshing where nodal constraints are lost during the remeshing phase.

## \*CONSTRAINED\_JOINT\_OPTION\_{OPTION}

Available forms include (one is mandatory):

**CONSTRAINED\_JOINT\_SPHERICAL**  
**CONSTRAINED\_JOINT\_REVOLUTE**  
**CONSTRAINED\_JOINT\_CYLINDRICAL**  
**CONSTRAINED\_JOINT\_PLANAR**  
**CONSTRAINED\_JOINT\_UNIVERSAL**  
**CONSTRAINED\_JOINT\_TRANSLATIONAL**  
**CONSTRAINED\_JOINT\_LOCKING**  
**CONSTRAINED\_JOINT\_TRANSLATIONAL\_MOTOR**  
**CONSTRAINED\_JOINT\_ROTATIONAL\_MOTOR**  
**CONSTRAINED\_JOINT\_GEAR**  
**CONSTRAINED\_JOINT\_RACK\_AND\_PINION**  
**CONSTRAINED\_JOINT\_CONSTANT\_VELOCITY**  
**CONSTRAINED\_JOINT\_PULLEY**  
**CONSTRAINED\_JOINT\_SCREW**

If the force output data is to be transformed into a local coordinate then an additional option is available:

### **LOCAL**

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

### **Card Format:**

Card 1 is required for all joint types.

Card 2 is required for joint types: MOTOR, GEARS, RACK\_AND\_PINION, PULLEY, and SCREW

Optional Card 3 is required only if LOCAL is specified in the keyword.

In the first seven joint types above excepting the Universal joint, the nodal points within the nodal pairs (1,2), (3,4), and (5,6) (see Figure 4.4) should coincide in the initial configuration, and the nodal pairs should be as far apart as possible to obtain the best behavior. For the Universal Joint the nodes within the nodal pair (3,4) do not coincide, but the lines drawn between nodes (1,3) and (2,4) must be perpendicular.

## \*CONSTRAINED

---

The geometry of joints is defined in Figure 5.4. When the penalty method is used (see \*CONTROL\_RIGID), at each time step, the relative penalty stiffness is multiplied by a function dependent on the step size to give the maximum stiffness that will not destroy the stability of the solution. Instabilities can result in the explicit time integration scheme if the penalty stiffness is too large. If instabilities occur, the recommended way to eliminate these problems is to decrease the time step or reduce the scale factor on the penalties..

For cylindrical joints, by setting node 3 to zero, it is possible to use a cylindrical joint to join a node that is not on a rigid body (node 1) to a rigid body (nodes 2 and 4).

### Card 1 - Required

Card 1            1            2            3            4            5            6            7            8

Variable	N1	N2	N3	N4	N5	N6	RPS	DAMP
Type	I	I	I	I	I	I	F	F
Default	0	0	0	0	0	0	1.0	1.0

#### VARIABLE

#### DESCRIPTION

N1	Node 1, in rigid body A. Define for all joint types.
N2	Node 2, in rigid body B. Define for all joint types.
N3	Node 3, in rigid body A. Define for all joint types except SPHERICAL
N4	Node 4, in rigid body B. Define for all joint types except SPHERICAL.
N5	Node 5, in rigid body A. Define for joint types TRANSLATIONAL, LOCKING, ROTATIONAL_MOTOR, CONSTANT_VELOCITY, GEARS, RACK_AND_PINION, PULLEY, and SCREW
N6	Node 6, in rigid body B. Define for joint types TRANSLATIONAL, LOCKING, ROTATIONAL_MOTOR, CONSTANT_VELOCITY, GEARS, RACK_AND_PINION, PULLEY, and SCREW
RPS	Relative penalty stiffness (default = 1.0).
DAMP	Damping scale factor on default damping value. (Revolute and Spherical Joints): EQ.0.0: default is set to 1.0, LE.0.01 and GT.0.0: no damping is used.

**\*CONSTRAINED**

**Card 2. Required for joint types MOTOR, GEARS, RACK\_AND\_PINION, PULLEY, and SCREW only.**

Card 1            1

Variable	PARM	LCID	TYPE					
Type	F	I	I					
Default	none							

**VARIABLE**

**DESCRIPTION**

PARM            Parameter which a function of joint type. Leave blank for MOTORS.  
                   Gears: define  $\frac{R_2}{R_1}$   
                   Rack and Pinion: define  $h$   
                   Pulley: define  $\frac{R_2}{R_1}$   
                   Screw: define  $\frac{\dot{x}}{\omega}$

LCID            Define load curve ID for MOTOR joints.

TYPE            Define integer flag for MOTOR joints as follows:  
                   EQ.0: translational/rotational velocity  
                   EQ.1: translational/rotational acceleration  
                   EQ.2: translational/rotational displacement

**Optional Card Format: Required only if LOCAL is specified after the keyword.**

Card 1            1            2            3            4            5            6            7            8

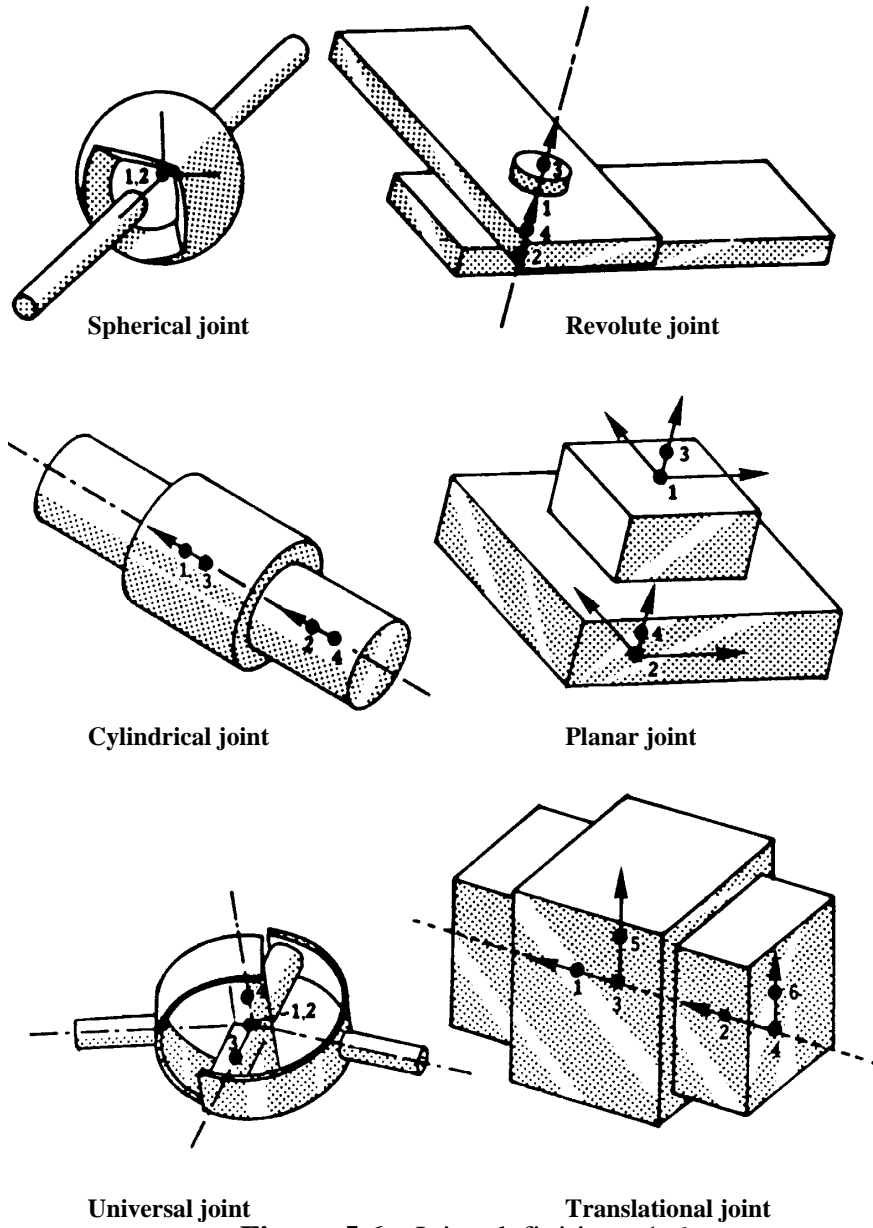
Variable	RAID	LST						
Type	I	I						
Default	0	0						

# \*CONSTRAINED

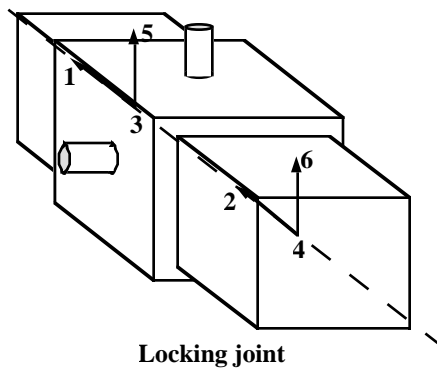
---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
RAID	Rigid body or accelerometer ID. The force resultants are output in the local system of the rigid body or accelerometer.
LST	Flag for local system type: EQ. 0: rigid body EQ. 1: accelerometer





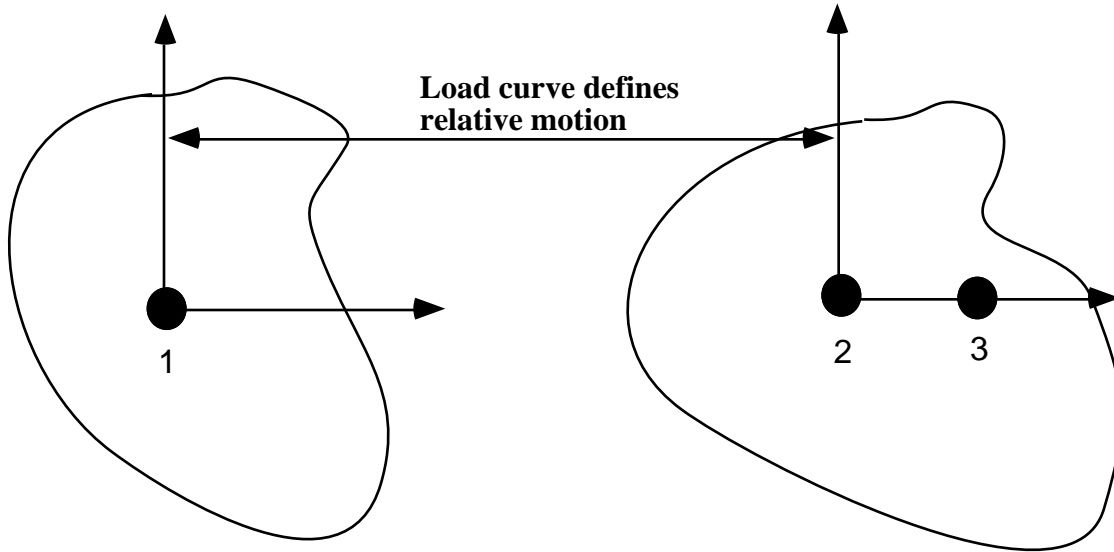
**Figure 5.6.** Joint definitions 1-6



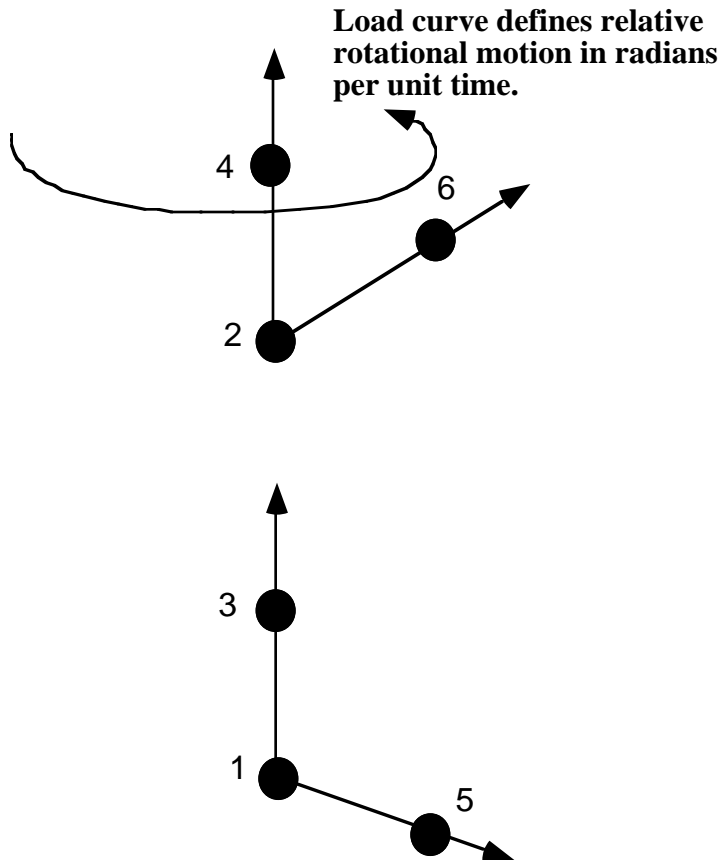
**Figure 5.7.** Locking joint.

# \*CONSTRAINED

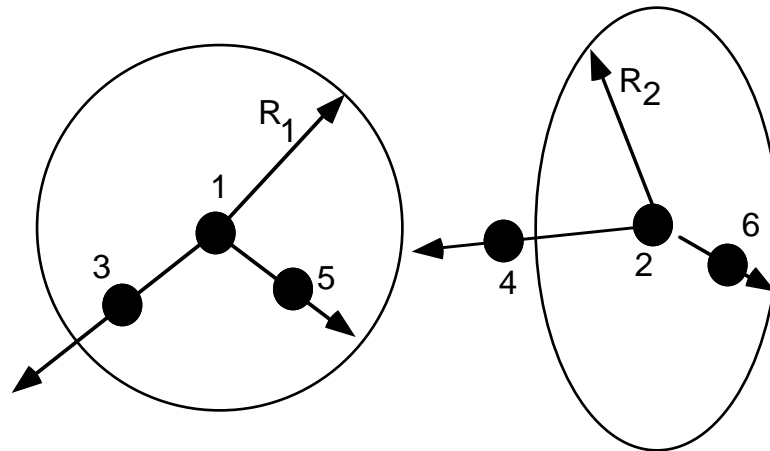
---



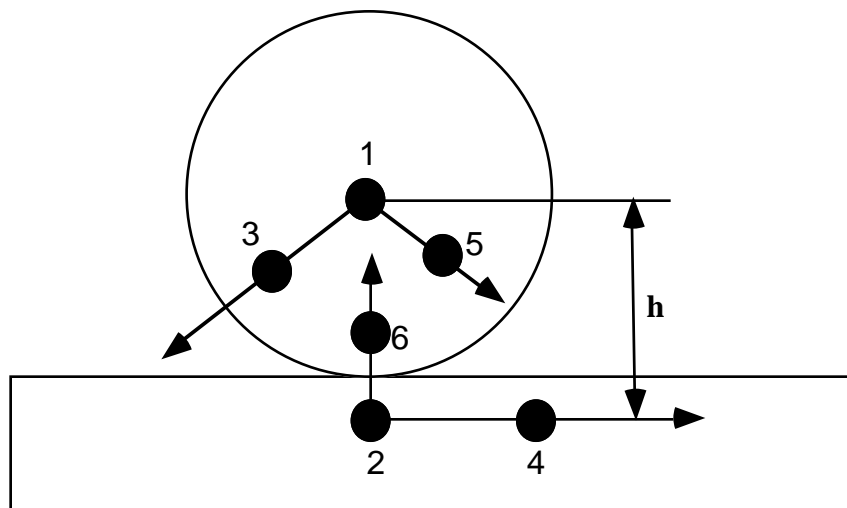
**Figure 5.8.** Translational motor joint. This joint can be used in combination with the translational or the cylindrical joint.



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



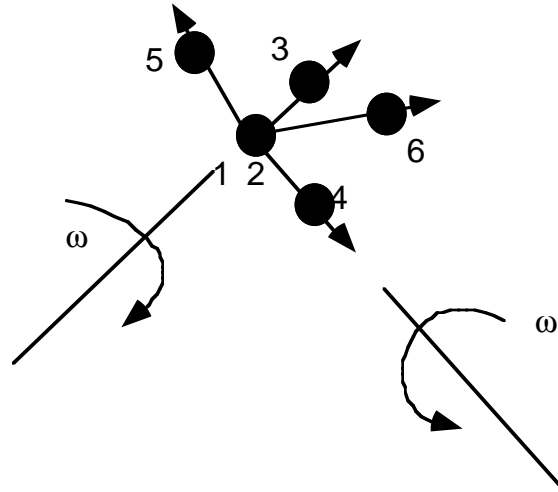
**Figure 5.10.** Gear joint. Nodal pairs (1,3) and (2,4) define axes that are orthogonal to the gears. Nodal pairs (1,5) and (2,6) define vectors in the plane of the gears. The ratio  $\frac{R_2}{R_1}$  is specified.



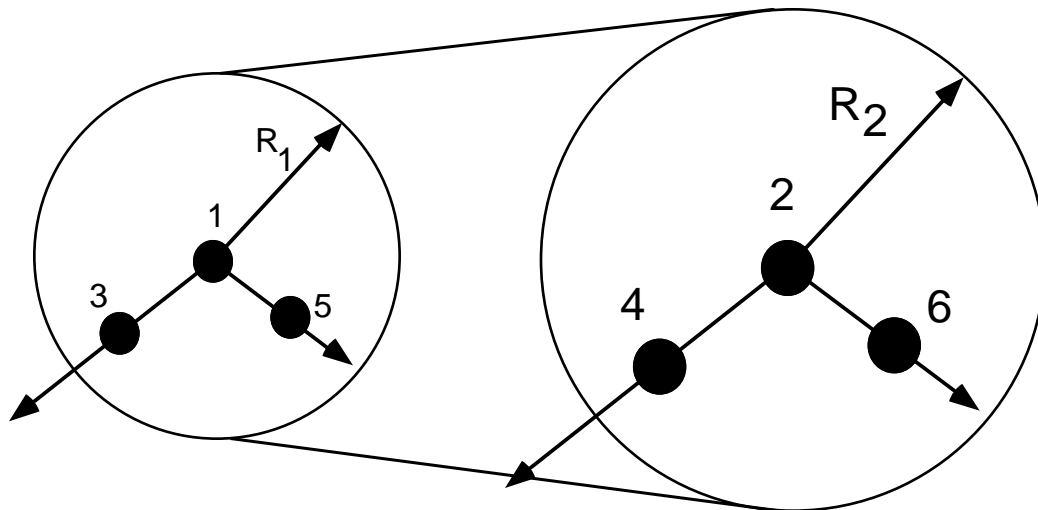
**Figure 5.11.** Rack and pinion joint. Nodal pair (1,3) defines an axes that is orthogonal to the gear. Nodal pair (1,5) is a vector in the plane of the gear. The value h is specified.

# \*CONSTRAINED

---



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



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



# \*CONSTRAINED

```

*SET_NODE_LIST
$   sid
$     6
$   nid1
$    150
$
$$$ request output for joint force data
$
*DATABASE_JNTFORC
$ dt/cycl      lcdt
$   0.0001
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$

$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$ *CONSTRAINED_JOINT_REVOLUTE
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$ Create a revolute joint between two rigid bodies. The rigid bodies must
$ share a common edge to define the joint along. This edge, however, must
$ not have the nodes merged together. Rigid bodies A and B will rotate
$ relative to each other along the axis defined by the common edge.
$
$ Nodes 1 and 2 are on rigid body A and coincide with nodes 9 and 10
$ on rigid body B, respectively. (This defines the axis of rotation.)
$
$ The relative penalty stiffness on the revolute joint is to be 1.0,
$ the joint is well lubricated, thus no damping at the joint is supplied.
$
*CONSTRAINED_JOINT_REVOLUTE
$
$.>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$   n1      n2      n3      n4      n5      n6      rps      damp
$     1      9      2      10           1.0
$
$ Note: A joint stiffness is not mandatory for this joint to work.
$ However, to see how a joint stiffness can be defined for this
$ particular joint, see the corresponding example listed in:
$ *CONSTRAINED_JOINT_STIFFNESS_GENERALIZED
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$

```

**\*CONSTRAINED\_JOINT\_STIFFNESS\_OPTION**

Options include:

**GENERALIZED  
FLEXION-TORSION**

Purpose: Define optional rotational joint stiffnesses for joints defined by \*CONSTRAINED\_JOINT\_OPTION. These definitions apply to all joints even though degrees of freedom that are considered in the joint stiffness capability may be constrained out in some joint types. The energy that is dissipated with the joint stiffness option is written for each joint in joint force file with the default name, JNTFORC. In the global energy balance this energy is included with the energy of the discrete elements, i.e., the springs and dampers.

**Card Format:**

Card 1 is common to all joint stiffness types.

Cards 2 to 4 are unique for each stiffness type.

**Card 1 - Required for all joint stiffness types.**

Card 1            1            2            3            4            5

Variable	JSID	PIDA	PIDB	CIDA	CIDB			
Type	I	I	I	I	I			
Default	none	none	none	none	CIDA			

**VARIABLE****DESCRIPTION**

JSID	Joint stiffness ID
PIDA	Part ID for rigid body A, see *PART.
PIDB	Part ID for rigid body B, see *PART.
CIDA	Coordinate ID for rigid body A, see *DEFINE_COORDINATE_OPTION.
CIDB	Coordinate ID for rigid body B. If zero, the coordinate ID for rigid body A is used, see *DEFINE_COORDINATE_OPTION.

# \*CONSTRAINED

---

## Card 2 of 4 - Required for GENERALIZED stiffness.

Card 2            1            2            3            4            5            6

Variable	LCIDPH	LCIDT	LCIDPS	DLCIDPH	DLCIDT	DLCIDPS		
Type	I	I	I	I	I	I		
Default	none	none	none	none	none	none		

VARIABLE	DESCRIPTION
LCIDPH	Load curve ID for $\phi$ -moment versus rotation in radians. See Figure 5.15. If zero, the applied moment is set to 0.0. See *DEFINE_CURVE.
LCIDT	Load curve ID for $\theta$ -moment versus rotation in radians. If zero, the applied moment is set to 0.0. See *DEFINE_CURVE.
LCIDPS	Load curve ID for $\psi$ -moment versus rotation in radians. If zero, the applied moment is set to 0.0. See *DEFINE_CURVE.
DLCIDPH	Load curve ID for $\phi$ -damping moment versus rate of rotation in radians per unit time. If zero, damping is not considered. See *DEFINE_CURVE.
DLCIDT	Load curve ID for $\theta$ -damping moment versus rate of rotation in radians per unit time. If zero, damping is not considered. See *DEFINE_CURVE.
DLCIDPS	Load curve ID for $\psi$ -damping torque versus rate of rotation in radians per unit time. If zero, damping is not considered. See *DEFINE_CURVE.



**Card 3 of 4 - Required for GENERALIZED stiffness.**

Card 3            1            2            3            4            5            6

Variable	ESPH	FMPH	EST	FMT	ESPS	FMPS		
Type	F	F	F	F	F	F		
Default	0.0	0.0	0.0	0.0	0.0	0.0		

---

**VARIABLE**

---

**DESCRIPTION**

ESPH	Elastic stiffness per unit radian for friction and stop angles for $\phi$ rotation. See Figure 5.16. If zero, friction and stop angles are inactive for $\phi$ rotation.
FMPH	Frictional moment limiting value for $\phi$ rotation. If zero, friction is inactive for $\phi$ 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 $\phi$ rotation. See Figure 5.8.
EST	Elastic stiffness per unit radian for friction and stop angles for $\theta$ rotation. See Figure 5.16. If zero, friction and stop angles are inactive for $\theta$ rotation.
FMT	Frictional moment limiting value for $\theta$ rotation. If zero, friction is inactive for $\theta$ 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 $\theta$ rotation. See Figure 5.16.
ESPS	Elastic stiffness per unit radian for friction and stop angles for $\psi$ rotation. See Figure 5.16. If zero, friction and stop angles are inactive for $\psi$ rotation..
FMPS	Frictional moment limiting value for $\psi$ rotation. If zero, friction is inactive for $\psi$ 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 $\psi$ rotation. See Figure 5.16.

# \*CONSTRAINED

---

## Card 4 of 4 - Required for GENERALIZED stiffness.

Card 4            1            2            3            4            5            6

Variable	NSAPH	PSAPH	NSAT	PSAT	NSAPS	PSAPS		
Type	F	F	F	F	F	F		
Default	not used	not used	not used	not used	not used	not used		

---

### VARIABLE

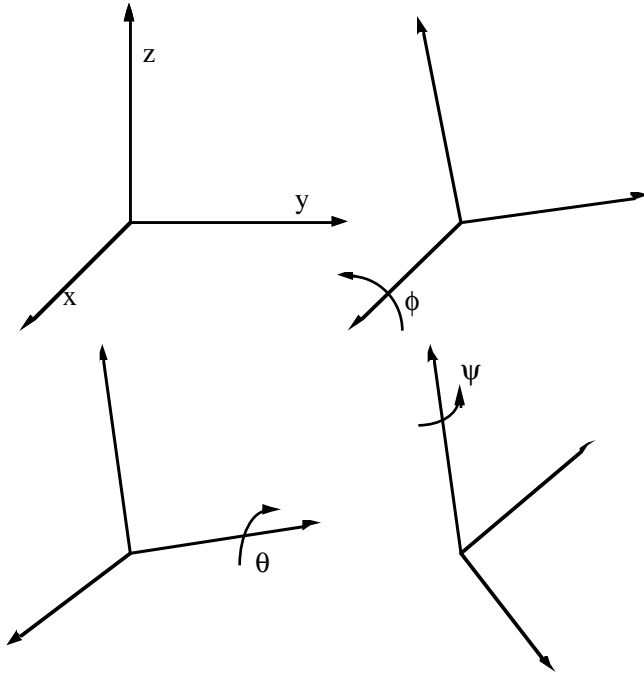
### DESCRIPTION

---

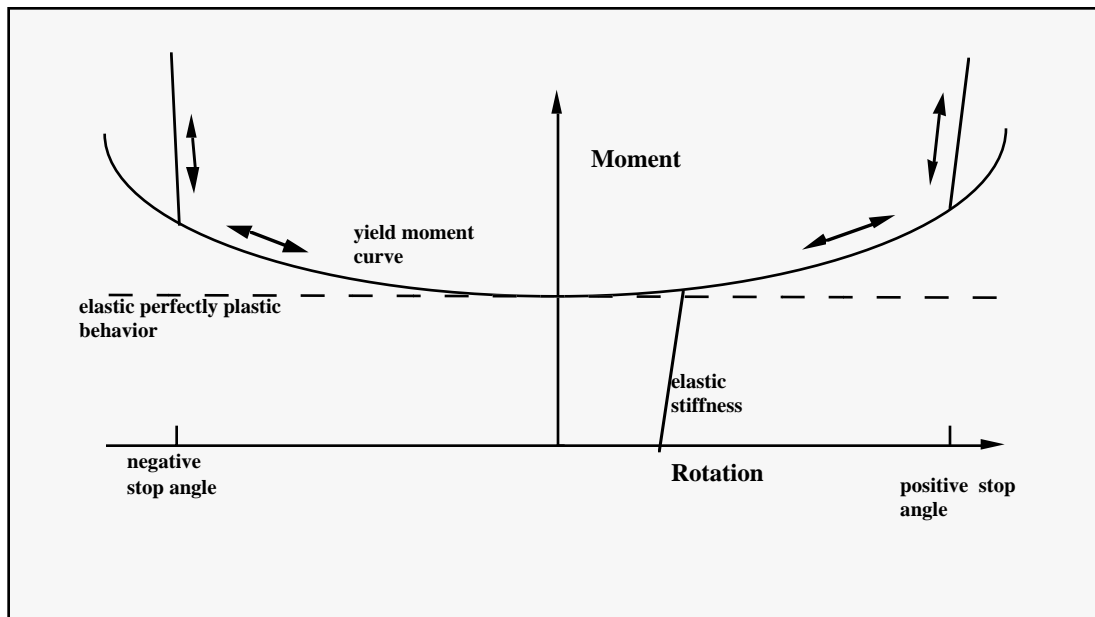
NSAPH	Stop angle in degrees for negative $\phi$ rotation. Ignored if zero.
PSAPH	Stop angle in degrees for positive $\phi$ rotation. Ignored if zero.
NSAT	Stop angle in degrees for negative $\theta$ rotation. Ignored if zero.
PSAT	Stop angle in degrees for positive $\theta$ rotation. Ignored if zero.
NSAPS	Stop angle in degrees for negative $\psi$ rotation. Ignored if zero.
PSAPS	Stop angle in degrees for positive $\psi$ rotation. Ignored if zero.

### **Remarks:**

After the stop angles are reached the torques increase linearly to resist further angular motion using the stiffness values on Card 3. Reasonable stiffness values have to be chosen. If the stiffness values are too low or zero, the stop will be violated.



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



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

# \*CONSTRAINED

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

Card 2            1            2            3            4            5            6

Variable	LCIDAL	LCIDG	LCIDBT	DLCIDAL	DLCIDG	DLCIDBT		
Type	I	I	I	I	I	I		
Default	none	1.0	none	none	1.0	none		

VARIABLE	DESCRIPTION
LCIDAL	Load curve ID for $\alpha$ -moment versus rotation in radians. See Figure 5.9 where it should be noted that $0 \leq \alpha \leq \pi$ . If zero, the applied moment is set to zero. See *DEFINE_CURVE.
LCIDG	Load curve ID for $\gamma$ versus a scale factor which scales the bending moment due to the $\alpha$ rotation. This load curve should be defined in the interval $-\pi \leq \gamma \leq \pi$ . If zero the scale factor defaults to 1.0. See *DEFINE_CURVE.
LCIDBT	Load curve ID for $\beta$ -torsion moment versus twist in radians. If zero the applied twist is set to zero. See *DEFINE_CURVE.
DLCIDAL	Load curve ID for $\alpha$ -damping moment versus rate of rotation in radians per unit time. If zero, damping is not considered. See *DEFINE_CURVE.
DLCIDG	Load curve ID for $\gamma$ -damping scale factor versus rate of rotation in radians per unit time. This scale factor scales the $\alpha$ -damping moment. If zero, the scale factor defaults to one. See *DEFINE_CURVE.
DLCIDBT	Load curve ID for $\beta$ -damping torque versus rate of twist. If zero damping is not considered. See *DEFINE_CURVE.

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

Card 3            1            2            3            4

Variable	ESAL	FMAL	ESBT	FMBT				
Type	F	F	F	F				
Default	0.0	0.0	0.0	0.0				

**VARIABLE**

**DESCRIPTION**

ESAL	Elastic stiffness per unit radian for friction and stop angles for $\alpha$ rotation, see Figure 5.17. If zero, friction and stop angles are inactive for $\alpha$ rotation..
FMAL	Frictional moment limiting value for $\alpha$ rotation. If zero, friction is inactive for $\alpha$ 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 $\alpha$ rotation, see Figure 5.17.
ESBT	Elastic stiffness per unit radian for friction and stop angles for $\beta$ twist, see Figure 5.17. If zero, friction and stop angles are inactive for $\beta$ twist.
FMBT	Frictional moment limiting value for $\beta$ twist. If zero, friction is inactive for $\beta$ twist. This option may also be thought of as an elastic-plastic spring. If a negative value is input then the absolute value is taken as the load curve ID defining the yield moment versus $\beta$ rotation, see Figure 5.17.

# \*CONSTRAINED

---

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

Card 4            1            2            3

Variable	SAAL	NSABT	PSABT					
Type	F	F	F					
Default	not used	not used	not used					

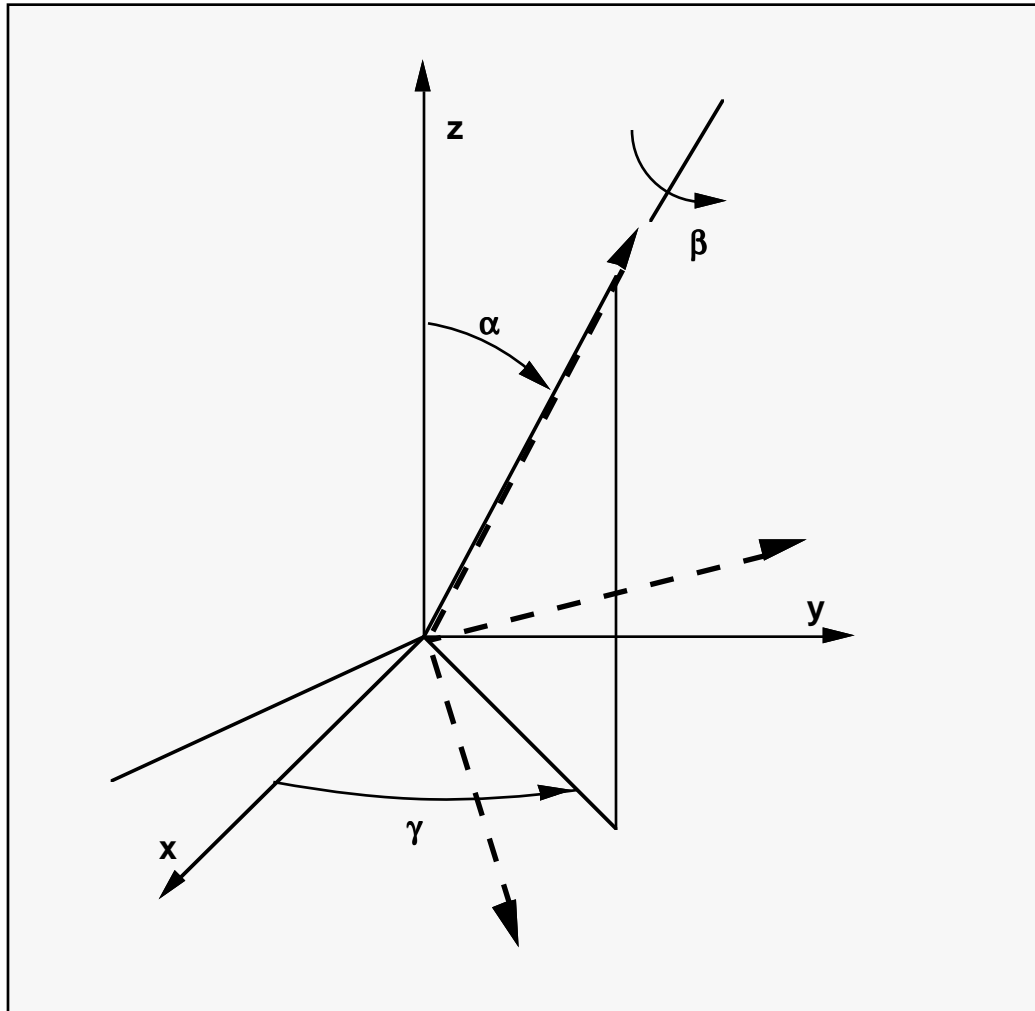
<u>VARIABLE</u>	<u>DESCRIPTION</u>
SAAL	Stop angle in degrees for $\alpha$ rotation where $0 \leq \alpha \leq \pi$ . Ignored if zero.
NSABT	Stop angle in degrees for negative $\beta$ rotation. Ignored if zero.
PSABT	Stop angle in degrees for positive $\beta$ rotation. Ignored if zero.

### **Remarks:**

This option simulates a flexion-torsion behavior of a joint in a slightly different fashion than with the generalized joint option.

After the stop angles are reached the torques increase linearly to resist further angular motion using the stiffness values on Card 3. If the stiffness value is too low or zero, the stop will be violated.

The moment resultants generated from the moment versus rotation curve, damping moment versus rate-of-rotation curve, and friction are evaluated independently and are added together.



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

# \*CONSTRAINED

---

```

$$$
$$$
$$$ *CONSTRAINED_JOINT_STIFFNESS_GENERALIZED
$$$
Define a joint stiffness for the revolute joint described in
*CONSTRAINED_JOINT_REVOLUTE
Attributes of the joint stiffness:
- Used for defining a stop angle of 30 degrees rotation
  (i.e., the joint allows a positive rotation of 30 degrees and
  then imparts an elastic stiffness to prevent further rotation)
- Define between rigid body A (part 1) and rigid body B (part 2)
- Define a local coordinate system along the revolute axis
  on rigid body A - nodes 1, 2 and 3 (cid = 5). This is used to
  define the revolute angles phi (PH), theta (T), and psi (PS).
- The elastic stiffness per unit radian for the stop angles
  are 100, 10, 10 for PH, T, and PS, respectively.
- Values not specified are not used during the simulation.
*CONSTRAINED_JOINT_STIFFNESS_GENERALIZED
$.>...1.>...2.>...3.>...4.>...5.>...6.>...7.>...8
$  jsid      pida      pidb      cida      cidb
$        1         1         2         5         5
$
$  lcidph    lcidt    lcidps    dlcidph    dlcidt    dlcidps
$
$  esph      fmps      est       fmt       esps      fmps
$        100.0      10.0     10.0     10.0
$
$  nsaph      psaph      nsat      psat      nsaps      psaps
$           30.0
$
$
*DEFINE_COORDINATE_NODES
$  cid      n1      n2      n3
$        5       1       2       3
$
$$$

```



# \*CONSTRAINED

## \*CONSTRAINED\_LAGRANGE\_IN\_SOLID

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

Note: This option will not work for RIGID slave PARTS.

### Card Format

Card 1            1            2            3            4            5            6            7            8

Variable	SLAVE	MASTER	SSTYP	MSTYP	NQUAD	CTYPE	DIREC	MCOUP
Type	I	I	I	I	I	I	I	I
Default	none	none	0	0	0	2	1	0

Card 2

Variable	START	END	PFAC	FRIC				
Type	F	F	F	F				
Default	0	1.0E10	0.1	0				

### VARIABLE

### DESCRIPTION

SLAVE            Part, part set ID or Segment set ID of slaves see \*PART , \*SET\_PART or \*SET\_SEGMENT.

MASTER           Part or part set ID of master solid elements, see \*PART or \*SET\_PART.

SSTYP            Slave type:  
                  EQ.0: part set ID,  
                  EQ.1: part ID.  
                  EQ.2: segment set ID

# \*CONSTRAINED

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MSTYP	Master type: EQ.0: part set ID, EQ.1: part ID.
NQUAD	Quadrature rule for coupling slaves to solids (CTYPE 2 only). EQ.0: at nodes only, EQ.n: use a rectangular grid of n*n points, EQ.-n: at nodes and a rectangular grid of n*n points.
CTYPE	Coupling type EQ.1: constrained acceleration, EQ.2: constrained acceleration and velocity (default), EQ.3: constrained acceleration and velocity, normal direction only, EQ.4: penalty coupling.
DIREC	Coupling direction (CTYPE 4 only). EQ.1: normal direction, compression and tension (default), EQ.2: normal direction, compression only, EQ.3: all directions.
MCOUP	Multi-material option (CTYPE 4 only). EQ.0: couple with all multi-material groups, EQ.1: couple with material with highest density.
START	Start time for coupling.
END	End time for coupling.
PFAC	Penalty factor (CTYPE 4 only).
FRIC	Coefficient of friction (DIREC 2 only).

**\*CONSTRAINED\_LINEAR**

Purpose: Define linear constraint equations between displacements/rotations, which can be defined in global coordinate systems.

**Card Format:**

Card 1 is required.

Card 2: One card 2 required for each node in the constraint equation.

**Remark:**

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.

**Card 1 - Required**

Card 1            1            2            3            4            5            6            7            8

Variable	NUM								
Type	I								
Default	none								

---

**VARIABLE**

---

**DESCRIPTION**

NUM

Number of nodes in equation

# \*CONSTRAINED

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

Card 2            1            2            3            4            5            6            7            8

Variable	NID	DOFX	DOFY	DOFZ	DOFRX	DOFRY	DOFRZ	COEF
Type	I	I	I	I	I	I	I	F
Default	none	0	0	0	0	0	0	0
Remark	1							

**VARIABLE**

**DESCRIPTION**

NID	Node ID
DOFX	Insert 1 (0) for (no) translational constraint in x-direction.
DOFY	Insert 1 (0) for (no) translational constraint in y-direction.
DOFZ	Insert 1 (0) for (no) translational constraint in z-direction.
DOFRX	Insert 1 (0) for (no) rotational constraint about x-axis.
DOFRY	Insert 1 (0) for (no) rotational constraint about y-axis.
DOFRZ	Insert 1 (0) for (no) rotational constraint about z-axis.
COEF	Nonzero coefficient, $C_k$

**Remarks:**

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,  $L$ , is constructed relating the unconstrained,  $u$ , and constrained,  $u_{\sim c}$ , degrees-of-freedom. The constrained accelerations used in the above equation are given by:

$$\ddot{u}_{\sim c} = \left[ \begin{matrix} L^t & M & L \end{matrix} \right]^{-1} \begin{matrix} L^t \\ \sim \\ \sim \end{matrix} F_{\sim}$$

where  $M$  is the Diagonal lumped mass matrix and  $F$  is the right hand side force vector. This requires the inversion of the condensed mass matrix which is equal in size to the number of constrained degrees-of-freedom minus one.

# \*CONSTRAINED

---

```

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

```

**\*CONSTRAINED\_NODAL\_RIGID\_BODY\_{OPTION}**

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

**INERTIA**

Purpose: Define a nodal rigid body. This is a rigid body which consists of the defined nodes. If the INERTIA option is not used, then the inertia tensor is computed from the nodal masses. Arbitrary motion of this rigid body is allowed. If the INERTIA option is used, constant translational and rotational velocities can be defined in a global or local coordinate system.

**Card Format:**

Card 1 is required.

Cards 2 - 4 are required for the INERTIA option.

Card 5 is required if a local coordinate system is used to specify the inertia tensor when the INERTIA option is used.

**Remarks:**

1. Unlike the \*CONSTRAINED\_NODE\_SET which permits only translational motion, here the equations of rigid body dynamics are used to update the motion of the nodes and therefore rotations of the nodal sets are admissible. Mass properties are determined from the nodal masses and coordinates. Inertial properties are defined if and only if the INERTIA option is specified.
2. Currently, no part number is assigned to the nodal rigid body, thus other options that use rigid bodies can not be used with these. If other rigid body options are desired, this can be accomplished by creating a part with no elements with a rigid material definition, and then add nodes to that part using the \*CONSTRAINED\_EXTRA\_NODES option.

# \*CONSTRAINED

---

## Card 1 - Required.

Card 1      1            2            3            4            5            6            7            8

Variable	NSID	CID						
Type	I	I						
Default	none	none						

---

### VARIABLE

---

### DESCRIPTION

NSID            Nodal set ID, see \*SET\_NODE\_OPTION. This nodal set defines the rigid body.

CID            Coordinate system ID for output of data in local system, see \*DEFINE\_COORDINATE\_OPTION. Only necessary if no local system is defined below.

## Card 2 of 4 - Required for the INERTIA option.

Card 2      1            2            3            4            5            6            7            8

Variable	XC	YC	ZC	TM	IRCS			
Type	F	F	F	F	I			
Default	0	0	0	0	0			

---

### VARIABLE

---

### DESCRIPTION

XC            x-coordinate of center of mass

YC            y-coordinate of center of mass

ZC            z-coordinate of center of mass

TM            Translational mass

IRCS            Flag for inertia tensor reference coordinate system:  
                   EQ.0: global inertia tensor,  
                   EQ.1: principal moments of inertias with orientation vectors as given below.



**Card 3 of 4 - Required for the INERTIA option.**

Card 3	1	2	3	4	5	6	7	8
Variable	IXX	IXY	IXZ	IYY	IYZ	IZZ		
Type	F	F	F	F	F	F		
Default	none	0	0	none	0	0		

**VARIABLE****DESCRIPTION**

IXX	$I_{xx}$ , xx component of inertia tensor
IXY	$I_{xy}$ (set to zero if IRCS=1)
IXZ	$I_{xz}$ (set to zero if IRCS=1)
IYY	$I_{yy}$ , yy component of inertia tensor
IYZ	$I_{yz}$ (set to zero if IRCS=1)
IZZ	$I_{zz}$ , zz component of inertia tensor

**Card 4 of 4 - Required for the INERTIA option.**

Card 4	1	2	3	4	5	6	7	8
Variable	VTX	VTY	VTZ	VRX	VRX	VRZ		
Type	F	F	F	F	F	F		
Default	0	0	0	0	0	0		

**VARIABLE****DESCRIPTION**

VTX	x-rigid body initial translational velocity in global coordinate system.
VTY	y-rigid body initial translational velocity in global coordinate system.
VTZ	z-rigid body initial translational velocity in global coordinate system.
VRX	x-rigid body initial rotational velocity in global coordinate system.
VRX	y-rigid body initial rotational velocity in global coordinate system.
VRZ	z-rigid body initial rotational velocity in global coordinate system.

# \*CONSTRAINED

---

Optional card required for IRCS=1. Define two local vectors or a local coordinate system ID.

Card 5            1            2            3            4            5            6            7            8

Variable	XL	YL	ZL	XLIP	YLIP	ZLIP	CID	
Type	F	F	F	F	F	F	I	
Default	none	none	none	none	none	none	none	

---

**VARIABLE**

---

---

**DESCRIPTION**

---

XL	x-coordinate of local x-axis. Origin lies at (0,0,0).
YL	y-coordinate of local x-axis
ZL	z-coordinate of local x-axis
XLIP	x-coordinate of local in-plane vector
YLIP	y-coordinate of local in-plane vector
ZLIP	z-coordinate of local in-plane vector
CID	Local coordinate system ID, see *DEFINE_COORDINATE_.... With this option leave fields 1-6 blank.

**Remark:**

The local coordinate system is set up in the following way. After the local x-axis is defined, the local z-axis is computed from the cross-product of the local x-axis vector with the given in-plane vector. Finally, the local y-axis is determined from the cross-product of the local z-axis with the local x-axis. The local coordinate system defined by CID has the advantage that the local system can be defined by nodes in the rigid body which makes repositioning of the rigid body in a preprocessor much easier since the local system moves with the nodal points.



# \*CONSTRAINED

---

## \*CONSTRAINED\_NODE\_SET

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

### Card Format

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

---

#### VARIABLE

#### DESCRIPTION

NSID	Nodal set ID, see *SET_NODE_OPTION.
DOF	Applicable degrees-of-freedom: EQ. 1: x-translational degree-of-freedom, EQ. 2: y-translational degree-of-freedom, EQ. 3: z-translational degree-of-freedom, EQ. 4: x and y-translational degrees-of-freedom, EQ. 5: y and z-translational degrees-of-freedom, EQ. 6: z and x-translational degrees-of-freedom, EQ. 7: x, y, and z-translational degrees-of-freedom.
TF	Failure time for nodal constraint set.

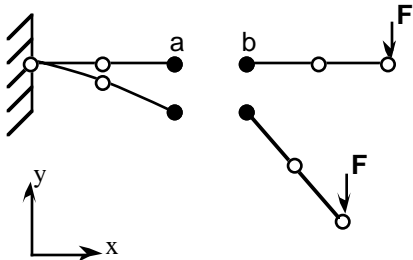
### Remarks:

1. The masses of the nodes are summed up to determine the total mass of the constrained set. It must be noted that the definition of a nodal rigid body is not possible with this input. For nodal rigid bodies the keyword input: \*CONSTRAINED\_NODAL\_RIGID\_BODY\_OPTION, must be used.
2. When the failure time, *TF*, is reached the nodal constraint becomes inactive and the constrained nodes may move freely.

## \*CONSTRAINED

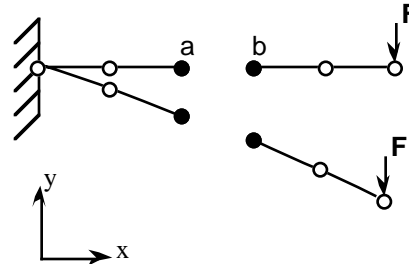
### \*CONSTRAINED\_NODE\_SET

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



### \*CONSTRAINED\_NODAL\_RIGID\_BODY \*CONSTRAINED\_SPOTWELD

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



**Offset nodes a and b are constrained to move together.**

**Figure 5.18.** \*CONSTRAINED\_NODE\_SET can lead to nonphysical responses.

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

# \*CONSTRAINED

---

## \*CONSTRAINED\_RIGID\_BODIES

Purpose: Merge two rigid bodies. One rigid body, called slave rigid body, is merged to the other one called a master rigid body.

### Card Format

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

<u>VARIABLE</u>	<u>DESCRIPTION</u>
PIDM	Master rigid body part ID, see *PART.
PIDS	Slave rigid body part ID, see *PART.

### Remarks:

The slave rigid body is merged to the master rigid body. The inertial properties computed by LS-DYNA are based on the combination of the master rigid body plus all the rigid bodies which are slaved to it unless the inertial properties of the master rigid body are defined via the \*PART\_INERTIA keyword in which case those properties are used for the combination of the master and slave rigid bodies. Note that a master rigid body may have many slaves.

Rigid bodies must not share common nodes since each rigid body updates the motion of its nodes independently of the other rigid bodies. If common nodes exist between rigid bodies the rigid bodies sharing the nodes must be merged.

It is also possible to merge rigid bodies that are completely separated and share no common nodal points or boundaries.

All actions valid for the master rigid body, e.g., constraints, given velocity, are now also valid for the newly-created rigid body.

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$ $ *CONSTRAINED_RIGID_BODIES
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$ Rigidly connect parts 35, 70, 71, and 72 to part 12.
$ All parts must be defined as rigid.
$
$ This example is used to make a single rigid body out of the five parts
$ that compose the back end of a vehicle. This was done to save cpu time
$ and was determined to be valid because the application was a frontal
$ impact with insignificant rear end deformations. (The cpu time saved
$ was from making the parts rigid, not from merging them - merging was
$ more of a convenience in this case for post processing, for checking
$ inertial properties, and for joining the parts.)
$
*CONSTRAINED_RIGID_BODIES
$...>...1...>...2...>...3...>...4...>...5...>...6...>...7...>...8
$   pidm    pids
$       12    35
$       12    70
$       12    71
$       12    72
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

# \*CONSTRAINED

---

## \*CONSTRAINED\_RIGID\_BODY\_STOPPERS

Purpose: Rigid body stoppers provide a convenient way of controlling the motion of rigid tooling in metalforming applications. The motion of a “master” rigid body is limited by load curves. This option will stop the motion based on a time dependent constraint. The stopper overrides prescribed velocity and displacement boundary conditions for both the master and slaved rigid bodies. See Figure 5.19.

### Card Format

Card 1            1            2            3            4            5            6            7            8

Variable	PID	LCMAX	LCMIN	PSIDMX	PSIDMN	LCVMNX	DIR	VID
Type	I	I	I	I	I	I	I	I
Default	required	0	0	0	0	0	required	0

Card 2            1            2            3            4            5            6            7            8

Variable	TB	TD						
Type	F	F						
Default	0	10 <sup>21</sup>						

### VARIABLE

### DESCRIPTION

PID	Part ID of master rigid body, see *PART.
LCMAX	Load curve ID defining the maximum coordinate or displacement as a function of time. See *DEFINE_CURVE: LT.0: Load Curve ID  LCMAX  provides an upper bound for the displacement of the rigid body EQ.0: no limitation of the maximum displacement. GT 0: Load Curve ID LCMAX provides an upper bound for the position of the rigid body center of mass



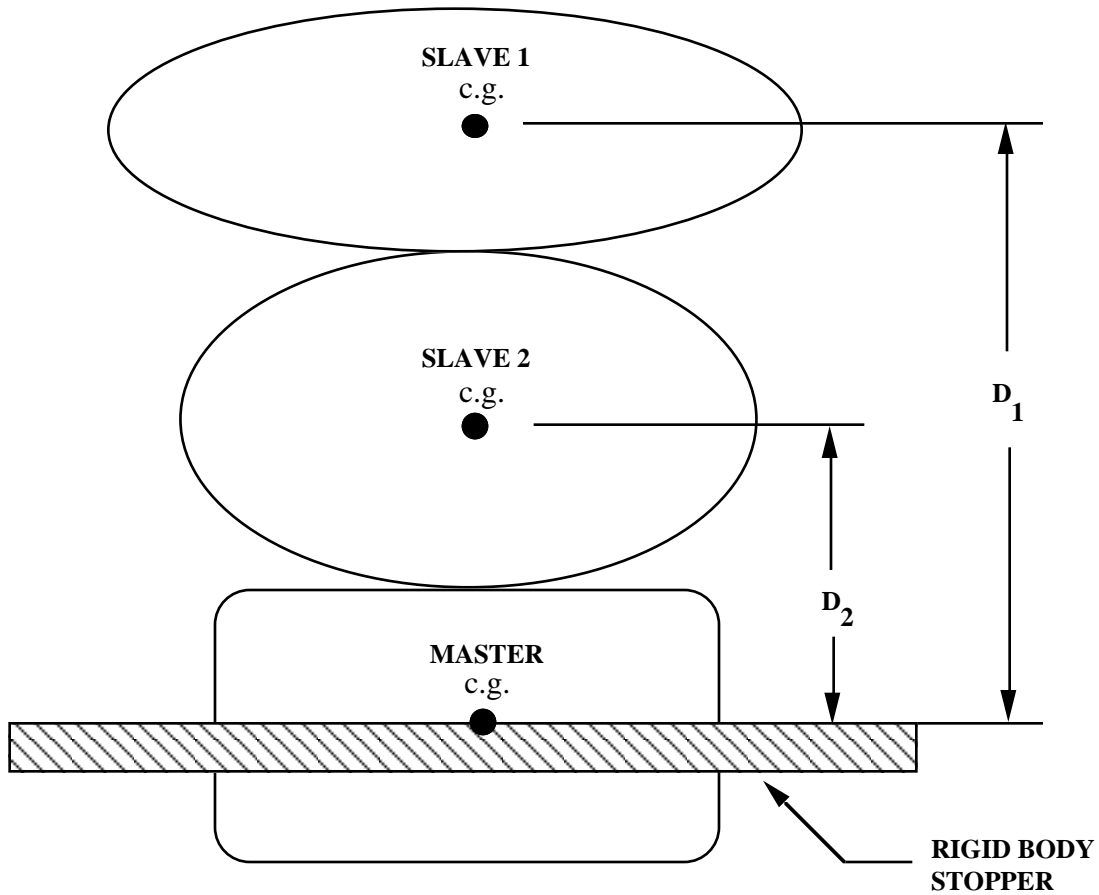
VARIABLE	DESCRIPTION
LCMIN	Load curve ID defining the minimum coordinate or displacement as a function of time. See *DEFINE_CURVE: LT.0: Load Curve ID  LCMIN  defines a lower bound for the displacement of the rigid body EQ.0: no limitation of the minimum displacement. GT.0: Load Curve ID LCMIN defines a lower bound for the position of the rigid body center of mass
PSIDMX	Optional part set ID of rigid bodies that are slaved in the maximum coordinate direction to the master rigid body. In the part set, see *SET_PART_OPTION, definition the "COLUMN" option may be used to defined as a part attribute the closure distance ( $D_1$ and $D_2$ in Figure 5.19) which activates the constraint. The constraint does not begin to act until the master rigid body stops. If the distance between the master rigid body is less than or equal to the closure distance, the slave rigid body motion towards the master rigid body also stops. However, the slaved rigid body is free to move away from the master. If the closure distance is input as zero (0.0) then the slaved rigid body stops when the master stops.
PSIDMN	Optional part set ID of rigid bodies that are slaved in the minimum coordinate direction to the master rigid body. In the part set, see *SET_PART_DEFINITION, definition the "COLUMN" option may be used to defined as a part attribute the closure distance ( $D_1$ and $D_2$ in Figure 5.11) which activates the constraint. The constraint does not begin to act until the master rigid body stops. If the distance between the master rigid body is less than or equal to the closure distance, the slave rigid body motion towards the master rigid body also stops. However, the slaved rigid body is free to move away from the master. If the closure distance is input as zero (0.0) then the slaved rigid body stops when the master stops.
LCVMNX	Load curve ID which defines the maximum absolute value of the velocity as a function of time that is allowed within the stopper. See *DEFINE_CURVE: EQ.0: no limitation on the velocity.
DIR	Direction stopper acts in: EQ.1: x-translation, EQ.2: y-translation, EQ.3: z-translation, EQ.4: arbitrary, defined by vector VID (see below), EQ.5: x-axis rotation , EQ.6: y-axis rotation, EQ.7: z-axis rotation, EQ.8: arbitrary, defined by vector VID (see below).
VID	Vector for arbitrary orientation of stopper, see *DEFINE_VECTOR.
TB	Time at which stopper is activated.
TD	Time at which stopper is deactivated.

# \*CONSTRAINED

---

## **Remark:**

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



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

**\*CONSTRAINED\_RIVET**

Purpose: Define massless rivets between non-contiguous nodal pairs. The nodes must not have the same coordinates. The action is such that the distance between the two nodes is kept constant throughout any motion. No failure can be specified.

**Card Format**

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

**VARIABLE****DESCRIPTION**

N1	Node ID
N2	Node ID
TF	Failure time for nodal constraint set.

**Remarks:**

1. Nodes connected by a rivet cannot be members of another constraint set that constrain the same degrees-of-freedom, a tied interface, or a rigid body, i.e., nodes cannot be subjected to multiple, independent, and possibly conflicting constraints. Also care must be taken to ensure that single point constraints applied to nodes in a constraint set do not conflict with the constraint sets constrained degrees-of-freedom.
2. When the failure time, *TF*, is reached the rivet becomes inactive and the constrained nodes may move freely.

# \*CONSTRAINED

---

```

$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$   *CONSTRAINED_RIVET
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$   Connect node 382 to node 88471 with a massless rivet.
$
*CONSTRAINED_RIVET
$
$.>...1.>...2.>...3.>...4.>...5.>...6.>...7.>...8
$
   n1      n2      tf
   382     88471    0.0
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$

```

**\*CONSTRAINED\_SHELL\_TO\_SOLID**

Purpose: Define a tie between a shell edge and solid elements. Nodal rigid bodies can perform the same function and may also be used.

**Card Format**

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

---

**VARIABLE**

---

**DESCRIPTION**

NID

Shell node ID

NSID

Solid nodal set ID, see *\*SET\_NODE\_OPTION*.**Remarks:**

The shell-brick interface, an extension of the tied surface capability, ties regions of hexahedron elements to regions of shell elements. A shell node may be tied to up to nine brick nodes lying along the tangent vector to the nodal fiber. See Figure 5.20. During the calculation, nodes thus constrained must lie along the fiber but can move relative to each other in the fiber direction. The brick nodes must be input in the order in which they occur, in either the plus or minus direction, as one moves along the shell node fiber.

This feature is intended to tie four node shells to eight node shells or solids; it is not intended for tying eight node shells to eight node solids.



**\*CONSTRAINED\_SPOTWELD\_{OPTION}**

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

**FILTERED\_FORCE**

and one additional card must be defined below.

Purpose: Define massless spot welds between non-contiguous nodal pairs. The spot weld is a rigid beam that connects the nodal points of the nodal pairs; thus, nodal rotations and displacements are coupled. The spot welds must be connected to nodes having rotary inertias, i.e., beams or shells. If this is not the case, for example, if the nodes belong to solid elements, use the option: \*CONSTRAINED\_RIVET. Note that shell elements do not have rotary stiffness in the normal direction and, therefore, this component cannot be transmitted. Spot welded nodes must not have the same coordinates. Coincident nodes in spot weld can be handeled by the \*CONSTRAINED\_NODAL\_RIGID\_BODY option. Brittle and ductile failures can be specified. Brittle failure is based on the resultant forces acting on the weld, and ductile failure is based on the average plastic strain value of the shell elements which include the spot welded node. Spot welds, which are connected to massless nodes, are automatically deleted in the initialization phase and a warning message is printed in the MESSAG file and the D3HSP file.

**Card 1 Format**

Card 1            1            2            3            4            5            6            7            8

Variable	N1	N2	SN	SS	N	M	TF	EP
Type	I	I	F	F	F	F	F	F
Default	none	none	optional	optional	optional	optional	1.E+20	1.E+20
Remarks	1.		2.				3	4

# \*CONSTRAINED

---

**Card 2 Format** Define if and only if the option **FILTERED\_FORCE** is specified.

Card 2            1            2            3            4            5            6            7            8

Variable	NF	TW						
Type	I	F						
Default	none	none						
Remarks								

<u>VARIABLE</u>	<u>DESCRIPTION</u>
N1	Node ID
N2	Node ID
SN	Normal force at spotweld failure (see Remark 2 below).
SS	Shear force at spotweld failure (see Remark 2 below).
N	Exponent for normal spotweld force (see Remark 2 below).
M	Exponent for shear spotweld force (see Remark 2 below).
TF	Failure time for nodal constraint set.
EP	Effective plastic strain at failure.
NF	Number of force vectors stored for filtering.
TW	Time window for filtering.

**Remarks:**

- Nodes connected by a spot weld cannot be members of another constraint set that constrain the same degrees-of-freedom, a tied interface, or a rigid body, i.e., nodes cannot be subjected to multiple, independent, and possibly conflicting constraints. Also, care must be taken to ensure that single point constraints applied to nodes in a constraint set do not conflict with the constraint sets constrained degrees-of-freedom.
- Failure of the spot welds occurs when:



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

where  $f_n$  and  $f_s$  are the normal and shear interface force. Component  $f_n$  is nonzero for tensile values only.

3. When the failure time,  $TF$ , is reached the spot weld becomes inactive and the constrained nodes may move freely.
4. Spot weld failure due to plastic straining occurs when the effective nodal plastic strain exceeds the input value,  $\epsilon_{fail}^p$ . This option can model the tearing out of a spotweld from the sheet metal since the plasticity is in the material that surrounds the spotweld, not the spotweld itself. A least squares algorithm is used to generate the nodal values of plastic strains at the nodes from the element integration point values. The plastic strain is integrated through the element and the average value is projected to the nodes via a least square fit. This option should only be used for the material models related to metallic plasticity and can result in slightly increased run times. Failures can include both the plastic and brittle failures.

# \*CONSTRAINED

---

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$ *CONSTRAINED_SPOTWELD
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$ Spotweld two nodes (34574 and 34383) with the approximate strength
$ of a 3/8" SAE Grade No 3 bolt.
$
*CONSTRAINED_SPOTWELD
$
$.>...1.>...2.>...3.>...4.>...5.>...6.>...7.>...8
$      n1      n2      sn      sf      n      m      tf      ps
$      34574    34383    36.0    18.0    2.0    2.0    10.    1.0
$
$      sn = 36.0 normal failure force is 36 kN
$      sf = 18.0 shear failure force is 18 kN
$      n = 2.0  normal failure criteria is raised to the power of 2
$      m = 2.0  shear failure criteria is raised to the power of 2
$      tf = 10.0 failure occurs at time 10 unless strain failure occurs
$      ps = 2.0 plastic strain at failure
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
```

**\*CONSTRAINED\_TIE-BREAK**

Purpose: Define a tied shell edge to shell edge interface that can release locally as a function of plastic strain of the shells surrounding the interface nodes. A rather ductile failure is achieved.

**Card Format**

	1	2	3	4	5	6	7	8
Variable	SNSID	MNSID	EPPF					
Type	I	I	F					
Default	none	none	0.					
Remarks		1, 2	3, 4					

---

**VARIABLE****DESCRIPTION**

---

SNSID	Slave node set ID, see *SET_NODE_OPTION.
MNSID	Master node set ID, see *SET_NODE_OPTION.
EPPF	Plastic strain at failure

**Remarks:**

1. Nodes in the master node set must be given in the order they appear as one moves along the edge of the surface.
2. Tie-breaks may not cross.
3. Tie-breaks may be used to tie shell edges together with a failure criterion on the joint. If the average volume-weighted effective plastic strain in the shell elements adjacent to a node exceeds the specified plastic strain at failure, the node is released. The default plastic strain at failure is defined for the entire tie-break but can be overridden in the slave node set to define a unique failure plastic strain for each node.
4. Tie-breaks may be used to simulate the effect of failure along a predetermined line, such as a seam or structural joint. When the failure criterion is reached in the adjoining elements, nodes along the slideline will begin to separate. As this effect propagates, the tie-breaks will appear to “unzip,” thus simulating failure of the connection.

# \*CONSTRAINED

---

## \*CONSTRAINED\_TIED\_NODES\_FAILURE

Purpose: Define a tied node set with failure based on plastic strain. The nodes must be coincident.

### Card Format

	1	2	3	4	5	6	7	8
Variable	NSID	EPPF	ETYPE					
Type	I	F						
Default	none	0.						
Remarks	1, 2, 3, 4							

<u>VARIABLE</u>	<u>DESCRIPTION</u>
NSID	Nodal set ID, see *SET_NODE_OPTION.
EPPF	Plastic strain at failure
ETYPE	Element type for nodal group: EQ:0: shell, EQ:1: solid element

### Remarks:

1. This feature applies only to deformable plastic three and four noded shell elements and to brick elements using the honeycomb material \*MAT\_HONEYCOMB. The specified nodes are tied together until the average volume weighted plastic strain exceeds the specified value. Entire regions of individual shell elements may be tied together unlike the tie-breaking shell slidelines. The tied nodes are coincident until failure. When the volume weighted average of the failure value is reached for a group of constrained nodes, the nodes of the elements that exceed the failure value are released to simulate the formation of a crack.
2. To use this feature to simulate failure, each shell element in the failure region should be generated with unique node numbers that are coincident in space with those of adjacent elements. Rather than merging these coincident nodes, the \*CONSTRAINED\_TIED\_NODES\_FAILURE option ties the nodal points together. As plastic strain develops and exceeds the failure strain, cracks will form and propagate through the mesh.
3. Entire regions of individual shell elements may be tied together, unlike the \*CONSTRAINED\_TIE-BREAK option. This latter option is recommended when the location of failure is known, e.g., as in the plastic covers which hide airbags in automotive structures.

## \*CONSTRAINED

4. When using surfaces of shell elements defined using the \*CONSTRAINED\_TIED\_NODES\_FAILURE option in contact, it is best to define each node in the surface as a slave node with the NODE\_TO\_SURFACE contact options. If this is not possible, the automatic contact algorithms beginning with \*CONTACT\_AUTOMATIC\_... all of which include thickness offsets are recommended.

```

$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$ *CONSTRAINED_TIED_NODES_FAILURE
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$ Tie shell elements together at the nodes specified in nodal set 101. The
$ constraint will be broken when the plastic strain at the nodes exceeds 0.085.
$
$ In this example, four shell elements come together at a common point.
$ The four corners of the shells are tied together with failure as opposed
$ to the more common method of merging the nodes in the pre-processing stage.
$
*CONSTRAINED_TIED_NODES_FAILURE
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$    nsid      eppf
$      101      0.085
$
$
*SET_NODE_LIST
$    sid
$      101
$    nid1     nid2     nid3     nid4     nid5     nid6     nid7     nid8
$      775      778      896      897
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$

```

**\*CONSTRAINED**

---

# **\*CONTACT**

The keyword **\*CONTACT** provides a way of treating interaction between disjoint parts. Different types of contact may be defined:

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

**\*CONTACT\_ENTITY**

**\*CONTACT\_GEBOD**

**\*CONTACT\_INTERIOR**

**\*CONTACT\_1D**

**\*CONTACT\_2D\_OPTION1\_{OPTION2}**

The first, **\*CONTACT\_...**, is the general 3D contact algorithms. The second, **\*CONTACT\_ENTITY**, treats contact using mathematical functions to describe the surface geometry for the master surface. The third, **\*CONTACT\_GEBOD** is a specialized form of the contact entity for use with the rigid body dummies (see **\*COMPONENT\_GEBOD**). The fourth, **\*CONTACT\_INTERIOR**, is under development and is used with soft foams where element inversion is sometimes a problem. Contact between layers of brick elements is treated to eliminate negative volumes. The fifth, **\*CONTACT\_1D**, remains in LS-DYNA for historical reasons, and is sometimes still used to model rebars which run along edges of brick elements. The last, **\*CONTACT\_2D**, is the general 2D contact algorithm based on those used previously in LS-DYNA2D.

# \*CONTACT

---

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

Purpose: Define a contact interface.

*OPTION1* specifies the contact type :

**AIRBAG\_SINGLE\_SURFACE**  
**AUTOMATIC\_GENERAL**  
**AUTOMATIC\_NODES\_TO\_SURFACE**  
**AUTOMATIC\_ONE\_WAY\_SURFACE\_TO\_SURFACE**  
**AUTOMATIC\_SINGLE\_SURFACE**  
**AUTOMATIC\_SURFACE\_TO\_SURFACE**  
**CONSTRAINT\_NODES\_TO\_SURFACE**  
**CONSTRAINT\_SURFACE\_TO\_SURFACE**  
**DRAWBEAD**  
**ERODING\_NODES\_TO\_SURFACE**  
**ERODING\_SINGLE\_SURFACE**  
**ERODING\_SURFACE\_TO\_SURFACE**  
**FORCE\_TRANSDUCER\_CONSTRAINT**  
**FORCE\_TRANSDUCER\_PENALTY**  
**FORMING\_NODES\_TO\_SURFACE**  
**FORMING\_ONE\_WAY\_SURFACE\_TO\_SURFACE**  
**FORMING\_SURFACE\_TO\_SURFACE**  
**NODES\_TO\_SURFACE**  
**ONE\_WAY\_SURFACE\_TO\_SURFACE**  
**RIGID\_NODES\_TO\_RIGID\_BODY**  
**RIGID\_BODY\_ONE\_WAY\_TO\_RIGID\_BODY**  
**RIGID\_BODY\_TWO\_WAY\_TO\_RIGID\_BODY**  
**SINGLE\_EDGE**  
**SINGLE\_SURFACE**  
**SLIDING\_ONLY**  
**SLIDING\_ONLY\_PENALTY**  
**SURFACE\_TO\_SURFACE**  
**TIEBREAK\_NODES\_TO\_SURFACE**  
**TIEBREAK\_SURFACE\_TO\_SURFACE**



**TIED\_NODES\_TO\_SURFACE**

**TIED\_SHELL\_EDGE\_TO\_SURFACE**

**TIED\_SURFACE\_TO\_SURFACE**

**TIED\_SURFACE\_TO\_SURFACE\_FAILURE**

*OPTION2* specifies a thermal contact and takes the single option:

**THERMAL**

Only the SURFACE\_TO\_SURFACE contact type may be used with this option.

*OPTION3* specifies that the first card to read defines the title and ID number of contact interface and takes the single option:

**TITLE**

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

**OFFSET**

Only contact types TIED\_NODES\_TO\_SURFACE, TIED\_SHELL\_EDGE\_TO\_SURFACE, and TIED\_SURFACE\_TO\_SURFACE may be used with this option. If this option is set, then offsets are permitted for these contact types, and, if not, the nodes are projected back to the contact surface during the initialization phase. The OFFSET option switches the formulation from a constraint type formulation to one that is penalty based.

## NOTES:

1. *OPTION1*, *OPTION2*, *OPTION3* and *OPTION4* may appear in any order in the keyword command line. The data must be in the order specified below.
2. *OPTION1* is mandatory.
3. *OPTION2*, *OPTION3* and *OPTION4* are optional.

## DISCUSSION AND EXAMPLES:

A brief discussion on the contact types and a few examples are provided at the end of this section. A theoretical discussion is provided in the LS-DYNA Theory Manual.

# \*CONTACT

---

## Card ordering is important in this section:

- **Card for the TITLE option is inserted here; otherwise, do not define this card.**

Define the title card first.

- **Cards 1 to 3 are mandatory for all contact types.**
- **Card 4 is mandatory for the following contact types:**

**\*CONTACT\_CONSTRAINT\_type**

**\*CONTACT\_DRAWBEAD**

**\*CONTACT\_ERODING\_type**

**\*CONTACT\_RIGID\_type**

**\*CONTACT\_TIEBREAK\_type**

Each of these types have different Card 4 formats. These card formats are presented in this manual after the optional cards specified above but, if used, Card 4 needs to be specified in your dyna deck before the optional cards.

- **Card for the THERMAL option is inserted here; otherwise, do not define this card.**

Additional parameters are required for thermal contact and are defined on this card.

- **Optional Card A**

Additional contact parameters that may be user specified. Default values have evolved over time to become pretty good values for most circumstances.

- **Optional Card B**

Additional contact parameters that may be user specified. Default values have evolved over time to become pretty good values for most circumstances. If Optional Card B is used, then Optional Card A is mandatory (use a blank line if no changes are desired for Card A parameters).



# \*CONTACT

---

Card 1 is mandatory for all contact types.

Card 1            1            2            3            4            5            6            7            8

Variable	SSID	MSID	SSTYP	MSTYP	SBOXID	MBOXID	SPR	MPR
Type	I	I	I	I	I	I	I	I
Default	none	none	none	none			0	0
Remarks	1	2			optional	optional	0=off	0=off

<u>VARIABLE</u>	<u>DESCRIPTION</u>
SSID	Slave segment, node set ID, partset ID, part ID, or shell element set ID, see *SET_SEGMENT, *SET_NODE_OPTION, *PART, *SET_PART or *SET_SHELL_OPTION: EQ.0: all segments are included for single surface contact.
MSID	Master segment set ID, partset ID, part ID, or shell element set ID, see *SET_SEGMENT, *SET_NODE_OPTION, *PART, *SET_PART, or *SET_SHELL_OPTION: EQ.0: for single surface contact.
SSTYP	Slave segment or node set type. The type must correlate with the number specified for SSID: EQ.0: segment set ID for surface to surface contact, EQ.1: shell element set ID for surface to surface contact, EQ.2: part set ID, EQ.3: part ID, EQ.4: node set ID for node to surface contact, EQ.5: include all for single surface definition. EQ.6: part set ID for exempted parts. All non exempted parts are included in the contact.
MSTYP	Master segment set type. The type must correlate with the number specified for MSID: EQ.0: segment set ID, EQ.1: shell element set ID, EQ.2: part set ID, EQ.3: part ID.

## Card 1 (continued)

<u>VARIABLE</u>	<u>DESCRIPTION</u>
SBOXID	BOXID, Include only slave nodes/segments within specified box, see *DEFINE_BOX, in contact definition. Only applies when SSID is defined by PART or PART SET.
MBOXID	BOXID, Include only master segments within specified box, see *DEFINE_BOX, in contact. Only applies when MSID is defined by PART or PART SET.
SPR	Include the slave side in the *DATABASE_NCFORC and the *DATABASE_BINARY_INTFOR interface force files: EQ.1: slave side forces included.
MPR	Include the master side in the *DATABASE_NCFORC and the *DATABASE_BINARY_INTFOR interface force files: EQ.1: master side forces included.

### **Remarks:**

1. Giving a slave set ID equal to zero is valid only for the single surface contact algorithms, i.e., the options SINGLE\_SURFACE, and the AUTOMATIC\_, AIRBAG\_, and ERODING\_SINGLE\_SURFACE options.
2. A master set ID is not defined for the single surface contact algorithms (including AUTOMATIC\_GENERAL) or FORCE\_TRANSDUCERS.

# \*CONTACT

Card 2 is mandatory for all contact types.

Card 2	1	2	3	4	5	6	7	8
Variable	FS	FD	DC	VC	VDC	PENCHK	BT	DT
Type	F	F	F	F	F	I	F	F
Default	0.	0.	0.	0.	0.	0	0.	1.0E20
Remarks								

<u>VARIABLE</u>	<u>DESCRIPTION</u>
FS	<p>Static coefficient of friction if FS IS &gt;0. and not equal to 2. The functional coefficient is assumed to be dependent on the relative velocity <math>v_{rel}</math> of the surfaces in contact <math>\mu_c = FD + (FS - FD)e^{-DC v_{rel} }</math>. The two other possibilities are:</p> <p>EQ.-1: If the frictional coefficients defined in the *PART section are to be used, set FS to a negative number (-1.0).</p> <p>WARNING: Please note that the FS=-1.0 option applies only to contact types: SINGLE_SURFACE, AUTOMATIC_GENERAL, AUTOMATIC_SINGLE_SURFACE, AUTOMATIC_NODES_TO_SURFACE, AUTOMATIC_SURFACE_TO_SURFACE, AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE, and ERODING_SINGLE_SURFACE.</p> <p>EQ. 2: For contact types SURFACE_TO_SURFACE and ONE_WAY_SURFACE_TO_SURFACE, the dynamic coefficient of friction points to the table, see DEFINE_TABLE (The table ID is give by FD below.), giving the coefficient of friction as a function of the relative velocity and pressure. This option must be used in combination with the thickness offset option. See Figure 6.1.</p>
FD	<p>Dynamic coefficient of friction. The functional coefficient is assumed to be dependent on the relative velocity <math>v_{rel}</math> of the surfaces in contact <math>\mu_c = FD + (FS - FD)e^{-DC v_{rel} }</math>. Give table ID if FS=2.</p> <p><b>Note:</b> For the special contact option "TIED_SURFACE_TO_SURFACE_FAILURE" only, the variables FS and FD act as failure stresses, i.e.,</p>

**Card 2 (continued)**

<u>VARIABLE</u>	<u>DESCRIPTION</u>
	failure occurs if $\left[ \frac{\max(0.0, \sigma_{normal})}{FS} \right]^2 + \left[ \frac{\sigma_{shear}}{FD} \right]^2 - 1 > 0$ where $\sigma_{normal}$ and $\sigma_{shear}$ are the interface normal and shear stresses.
FS	Normal tensile stress at failure
FD	Shear stress at failure
DC	Exponential decay coefficient. The functional coefficient is assumed to be dependent on the relative velocity $v_{rel}$ of the surfaces in contact $\mu_c = FD + (FS - FD)e^{-DC v_{rel} }$ .
VC	Coefficient for viscous friction. This is necessary to limit the friction force to a maximum. A limiting force is computed $F_{lim} = VC \cdot A_{cont} \cdot A_{cont}$ being the area of the segment contacted by the node in contact. The suggested value for $VC$ is to use the yield stress in shear $VC = \frac{\sigma_o}{\sqrt{3}}$ where $\sigma_o$ is the yield stress of the contacted material.
VDC	Viscous damping coefficient in percent of critical. In order to avoid undesirable oscillation in contact, e.g., for sheet forming simulation, a contact damping perpendicular to the contacting surfaces is applied. Damping coefficient $\xi = \frac{VDC}{100} \xi_{wd}$ , eg VDC = 20. $\xi_{crit}$ is determined in the following fashion by LS-DYNA. $\xi_{crit} = 2mw; \quad m = \min(m_{slave}, m_{master}) \quad \begin{matrix} \text{mass of master} \\ \text{resp. slave node} \end{matrix}$ $w = \sqrt{k \cdot \frac{m_{slave} + m_{master}}{m_{slave} \cdot m_{master}}} \quad k \text{ interface stiffness}$
PENCHK	Small penetration in contact search option. If the slave node penetrates more than the segment thickness times the factor XPENE, see *CONTROL_CONTACT, the penetration is ignored and the slave node is set free. The thickness is taken as the shell thickness if the segment belongs to a shell element or it is taken as 1/20 of its shortest diagonal if the segment belongs to a solid element. This option applies to the surface to surface contact algorithms: See table 6.1 for contact types and more details. EQ.0: check is turned off, EQ.1: check is turned on , EQ.2: check is on but shortest diagonal is used.
BT	Birth time (contact surface becomes active at this time).
DT	Death time (contact surface is deactivated at this time).

# \*CONTACT

---

Card 3 is mandatory for all contact types.

Card 3            1            2            3            4            5            6            7            8

Variable	SFS	SFM	SST	MST	SFST	SFMT	FSF	VSF
Type	F	F	F	F	F	F	F	F
Default	1.	1.	element thickness	element thickness	1.	1.	1.	1.

<u>VARIABLE</u>	<u>DESCRIPTION</u>
SFS	Scale factor on default slave penalty stiffness, see also *CONTROL_CONTACT.
SFM	Scale factor on default master penalty stiffness, see also *CONTROL_CONTACT.
SST	Optional thickness for slave surface (overrides true thickness). This option applies only to contact with shell elements. True thickness is the element thickness of the shell elements.
MST	Optional thickness for master surface (overrides true thickness). This option applies only to contact with shell elements. True thickness is the element thickness of the shell elements.
SFST	Scale factor for slave surface thickness (scales true thickness). This option applies only to contact with shell elements. True thickness is the element thickness of the shell elements.
SFMT	Scale factor for master surface thickness (scales true thickness). This option applies only to contact with shell elements. True thickness is the element thickness of the shell elements.
FSF	Coulomb friction scale factor. The Coulomb friction value is scaled as $\mu_{sc} = FSF \cdot \mu_c$ , see above.
VSF	Viscous friction scale factor. If this factor is defined then the limiting force becomes: $F_{lim} = VSF \cdot VC \cdot A_{cont}$ , see above.

## Remarks:

The variables FSF and VSF above can be overridden segment by segment on the \*SET\_SEGMENT or \*SET\_SHELL\_OPTION cards for the **slave surface only** as A3 and A4, and for the **master surface only** as A1 and A2. See \*SET\_SEGMENT and \*SET\_SHELL\_OPTION.



**This Card 4 is mandatory for:**

**\*CONTACT\_CONSTRAINT\_NODES\_TO\_SURFACE**

**\*CONTACT\_CONSTRAINT\_SURFACE\_TO\_SURFACE**

Card 4            1            2            3            4            5            6            7            8

Variable	KPF								
Type	F								
Default	0.0								

**VARIABLE**

**DESCRIPTION**

KPF

Kinematic partition factor for constraint:  
 EQ. 0.0: fully symmetric treatment.  
 EQ. 1.0: one way treatment with slave nodes constrained to master surface. Only the slave nodes are checked against contact.  
 EQ. -1.0: one way treatment with master nodes constrained to slave surface. Only the master nodes are checked against contact.

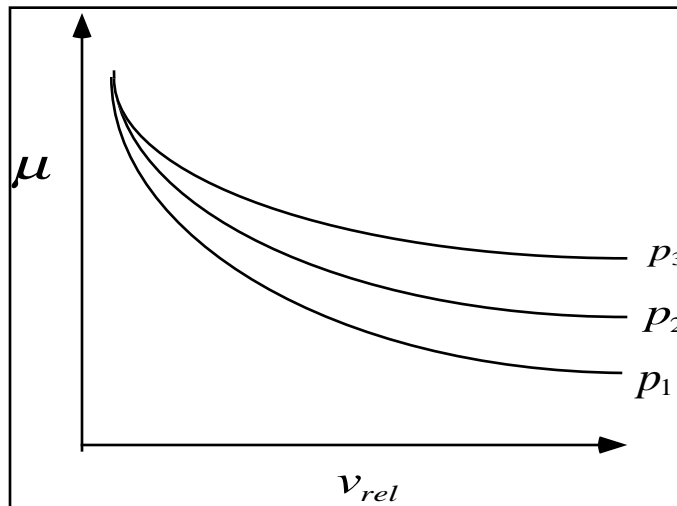


Figure 6.1. Friction coefficient,  $\mu$ , can be a function of relative velocity and pressure. Specify a flag for the static coefficient of friction, FS, and a table ID for the dynamic coefficient. This option only works with SURFACE\_TO\_SURFACE and ONE\_WAY\_SURFACE\_TO\_SURFACE with thickness offsets.

# \*CONTACT

---

This Card 4 is mandatory for:

## \*CONTACT\_ DRAWBEAD

Card 4            1            2            3            4            5            6            7            8

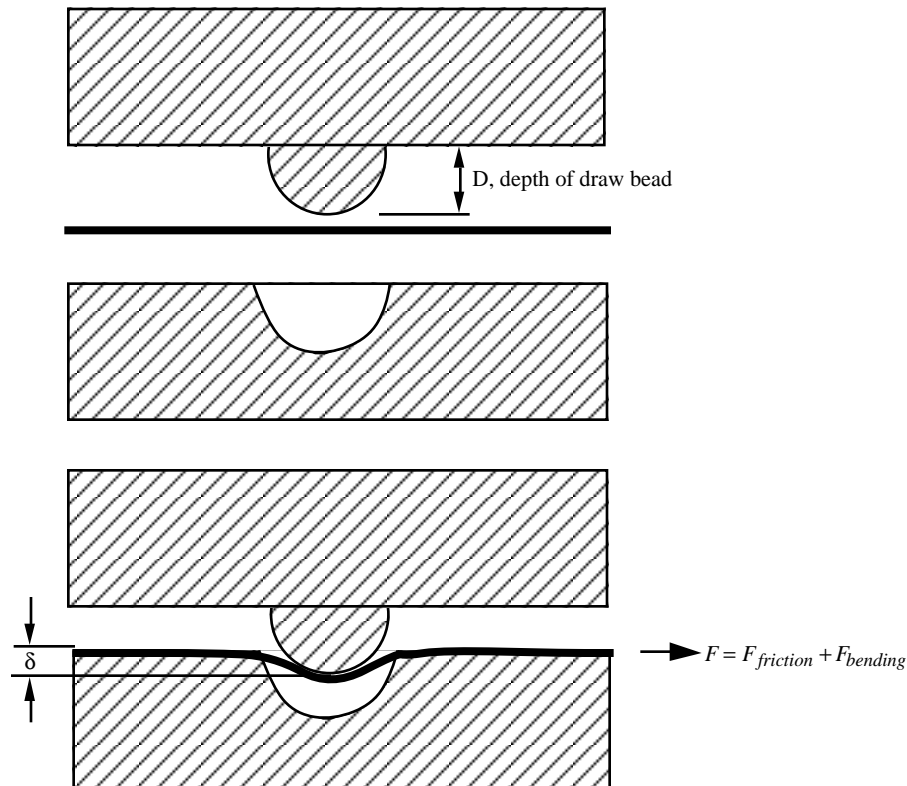
Variable	LCIDRF	LCIDNF	DBDTH	DFSCL	NUMINT			
Type	I	I	F	F	I			
Default	required	none	0.0	1.0	0			

VARIABLE	DESCRIPTION
LCIDRF	<p>If LCIDRF is positive then it defines the load curve ID giving the bending component of the restraining force, <math>F_{bending}</math>, per unit draw bead length as a function of displacement, <math>\delta</math>, see Figure 6.2. This force is due to the bending and unbending of the blank as it moves through the drawbead. The total restraining force is the sum of the bending and friction components.</p> <p>If LCIDRF is negative then the absolute value gives the load curve ID defining max bead force versus normalized drawbead length. The abscissa values is between zero and 1 and is the normalized drawbead length. The ordinate gives the maximum allowed drawbead retaining force when the bead is in the fully closed position. If the drawbead is not fully closed linear interpolation is used to compute the drawbead force.</p>
LCIDNF	<p>Load curve ID giving the normal force per unit draw bead length as a function of displacement, <math>\delta</math>, see Figure 6.2. This force is due to the bending of the blank into the draw bead as the binder closes on the die and represents a limiting value. The normal force begins to develop when the distance between the die and binder is less than the draw bead depth. <i>As the binder and die close on the blank this force should diminish or reach a plateau</i>, see the explanation below.</p>
DBDTH	<p>Draw bead depth, see Figure 6.2. Necessary to determine correct <math>\delta</math> displacement from contact displacements.</p>
DFSCL	<p>Scale factor for load curve. Default=1.0. This factor scales load curve ID, LCIDRF above.</p>

VARIABLE	DESCRIPTION
NUMINT	<p>Number of equally spaced integration points along the draw bead:            EQ.0: Internally calculated based on element size of elements that interact with draw bead.            This is necessary for the correct calculation of the restraining forces. More integration points may increase the accuracy since the force is applied more evenly along the bead.</p>

**Remarks:**

The draw bead is defined by a *consecutive* list of *slave* nodes that lie along the draw bead. For straight draw beads only two nodes need to be defined, i.e., one at each end, but for curved beads sufficient nodes are required to define the curvature of the bead geometry. The integration points along the bead are equally spaced and are independent of the nodal spacing used in the definition of the draw bead. By using the capability of tying extra nodes to rigid bodies (see \*CONSTRAINED\_EXTRA\_NODES\_OPTION) the draw bead nodal points do not need to belong to the element connectivities of the die and binder. The blank makes up the master surface. IT IS HIGHLY RECOMMENDED TO DEFINE A BOXID AROUND THE DRAWBEAD TO LIMIT THE SIZE OF THE MASTER SURFACE CONSIDERED FOR THE DRAW BEAD. THIS WILL SUBSTANTIALLY REDUCE COST AND MEMORY REQUIREMENTS.



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

# \*CONTACT

---

This Card 4 is mandatory for:

\*CONTACT\_ ERODING\_NODES\_TO\_SURFACE

\*CONTACT\_ ERODING\_SINGLE\_SURFACE

\*CONTACT\_ ERODING\_SURFACE\_TO\_SURFACE

Card 4            1            2            3            4            5            6            7            8

Variable	ISYM	EROSOP	IADJ						
Type	I	I	I						
Default	0	1	0						

VARIABLE	DESCRIPTION
ISYM	<p>Symmetry plane option:            EQ.0: off,            EQ.1: do not include faces with normal boundary constraints (e.g., segments of brick elements on a symmetry plane).            This option is important to retain the correct boundary conditions in the model with symmetry.</p>
EROSOP	<p>Erosion/Interior node option:            EQ.0: only exterior boundary information is saved,            EQ.1: storage is allocated so that eroding contact can occur. Otherwise, no contact is assumed after erosion of the corresponding element.</p>
IADJ	<p>Adjacent material treatment for solid elements:            EQ.0: solid element faces are included only for free boundaries,            EQ.1: solid element faces are included if they are on the boundary of the material subset. This option also allows the erosion within a body and the consequent treatment of contact.</p>

**This Card 4 is mandatory for:**

**\*CONTACT\_RIGID\_NODES\_TO\_RIGID\_BODY**

**\*CONTACT\_RIGID\_BODY\_ONE\_WAY\_TO\_RIGID\_BODY**

**\*CONTACT\_RIGID\_BODY\_TWO\_WAY\_TO\_RIGID\_BODY**

Card 4            1            2            3            4            5            6            7            8

Variable	LCID	FCM	US					
Type	I	I	F					
Default	required	required	from LCID					

**VARIABLE**

**DESCRIPTION**

LCID	Load curve ID giving force versus penetration behavior for RIGID_ contact. See also the definition of FCM below.
FCM	Force calculation method for RIGID_contact: EQ.1: Load curve gives total normal force on surface versus maximum penetration of any node (RIGID_BODY_ONE_WAY only). EQ.2: Load curve gives normal force on each node versus penetration of node through the surface (all RIGID_contact types). EQ.3: Load curve gives normal pressure versus penetration of node through the surface (RIGID_BODY_TWO_WAY and RIGID_BODY_ONE_WAY only). EQ.4: Load curve gives total normal force versus maximum soft penetration. In this case the force will be followed based on the original penetration point. (RIGID_BODY_ONE_WAY only).
US	Unloading stiffness for RIGID_contact. The default is to unload along the loading curve. This should not be larger than the maximum value used in the loading curve.

# \*CONTACT

---

This Card 4 is mandatory for:

## \*CONTACT\_ TIEBREAK\_NODES\_TO\_SURFACE

Card 4            1            2            3            4            5            6            7            8

Variable	NFLF	SFLF	NEN	MES				
Type	F	F	F	F				
Default	required	required	2.	2.				

<b>VARIABLE</b>	<b>DESCRIPTION</b>
-----------------	--------------------

- |      |   |
|------|---|
| NFLF | Normal failure force. Only tensile failure, i.e., tensile normal forces, will be considered in the failure criterion. |
| SFLF | Shear failure force   |
| NEN  | Exponent for normal force   |
| MES  | Exponent for shear force. Failure criterion:  |

$$\left( \frac{|f_n|}{NFLF} \right)^{NEN} + \left( \frac{|f_s|}{SFLF} \right)^{MES} \geq 1.$$

Failure is assumed if the left side is larger than 1.  $f_n$  and  $f_s$  are the normal and shear interface force.

**Remarks:**

These attributes can be overridden node by node on the \*SET\_NODE\_option cards.

**This Card 4 is mandatory for:**

**\*CONTACT\_ TIEBREAK\_SURFACE\_TO\_SURFACE**

Card 4            1            2            3            4            5            6            7            8

Variable	NFLS	SFLS	TBLCID					
Type	F	F	I					
Default	required	required	0					

**VARIABLE**

**DESCRIPTION**

NFLS            Normal failure stress. See remark below.

SFLS            Shear failure stress. Failure criterion:

$$\left( \frac{|\sigma_n|}{NFLS} \right)^2 + \left( \frac{|\sigma_s|}{SFLS} \right)^2 \geq 1.$$

TBLCID            Optional load curve number defining the resisting stress versus gap opening for the post failure response. This can be used to model the failure of adhesives.

**Remarks:**

The failure attributes can be overridden segment by segment on the \*SET\_SEGMENT or \*SET\_SHELL\_option cards for the **slave surface only** as A1 and A2. These variables do not apply to the master surface.

After failure, this contact option behaves as a surface-to-surface contact with no thickness offsets. After failure, no interface tension is possible.

# \*CONTACT

---

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

**Reminder:** If Card 4 is required, then it must go before this optional card. (Card 4 is required for certain contact types - see earlier in this section for the list, later in this section for details of Card 4.)

\*CONTACT\_ ...\_THERMAL\_.....

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

VARIABLE	DESCRIPTION
CF	<p>Thermal conductivity (<math>k</math>) of fluid between the slide surfaces. If a gap with a thickness <math>l_{gap}</math> exists between the slide surfaces, then the conductance due to thermal conductivity between the slide surfaces is</p> $h_{cond} = \frac{k}{l_{gap}}$ <p>Note that LS- DYNA calculates <math>l_{gap}</math> based on deformation.</p>
RAD	<p>Radiation conductance (<math>h_{rad}</math>) between the slide surfaces. If a gap exists between the slide surfaces, then the contact conductance is calculated by</p> $h = h_{cond} + h_{rad}$
HTC	<p>Heat transfer conductance (<math>h_{cont}</math>) for closed gaps. Use this heat transfer conductance for gaps in the range</p> $0 \leq l_{gap} \leq l_{min}$ <p>where <math>l_{min}</math> is GCRIT defined below.</p>
GCRIT	<p>Critical gap (<math>l_{min}</math>), use the heat transfer conductance defined (HTC) for gap thicknesses less than this value.</p>



<u>VARIABLE</u>	<u>DESCRIPTION</u>
GMAX	No thermal contact if gap is greater than this value ( $l_{max}$ ).
CD_FACT	Is a multiplier used on the element characteristic distance for the search routine. The characteristic length is the largest interface surface element diagonal.

EQ:0. Default set to 1.0

## **Remarks:**

In summary:

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

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

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

# \*CONTACT

---

## Optional Card A

**Reminder:** If Card 4 is required, then it must go before this optional card. (Card 4 is required for certain contact types - see earlier in this section for the list, later in this section for details of Card 4.)

Optional Card A	1	2	3	4	5	6	7	8
Variable	SOFT	SOFSCCL	LCIDAB	MAXPAR	EDGE	DEPTH	BSORT	FRCFRQ
Type	I	F	I	F	F	I	I	I
Default	0	.1	0	1.025.	0.	2	10-100	1
Remarks			type a13			type 13		

### VARIABLE

### DESCRIPTION

SOFT	Soft constraint option: EQ.0: penalty formulation, EQ.1: soft constraint formulation, EQ.2: alternate penalty formulation with segment based contact. EQ.4: constraint approach for FORMING contact option. Soft constraint necessary if a surface in contact has wildly varying stiffnesses along surface. See the remark below for the alternate penalty formulation.
SOFSCCL	Scale factor for constraint forces of soft constraint option (default=.10). Values greater than .5 for single surface contact and 1.0 for a one way treatment are inadmissible.
LCIDAB	Load curve ID defining airbag thickness as a function of time for type a13 contact (*CONTACT_AIRBAG_SINGLE_SURFACE).
MAXPAR	Maximum parametric coordinate in segment search (values 1.025 and 1.20 recommended). Larger values can increase cost. If zero, the default is set to 1.025. This factor allows an increase in the size of the segments . May be useful at sharp corners.
EDGE	Edge-to-edge penetration check for alternate penalty formulation (SOFT=2). EQ.0: Check only surface penetrations (default). GT.0: Check both surface and edge-edge penetrations.

**Optional Card A (continued)**

<u>VARIABLE</u>	<u>DESCRIPTION</u>
DEPTH	Search depth in automatic contact. Value of 1 is sufficiently accurate for most crash applications and is much less expensive. LS-DYNA for improved accuracy sets this value to 2. If zero, the default is set to 2. LT.0:  DEPTH  is the load curve ID defining searching depth versus time.
BSORT	Number of cycles between bucket sorts. Values of 25 and 100 are recommended for contact types 4 and 13 (SINGLE_SURFACE), respectively. Values of 10-15 are okay for the surface to surface and node to surface contact. If zero, LS-DYNA determines the interval. LT.0:  BSORT  load curve ID defining bucket sorting frequency versus time.
FRCFRQ	Number of cycles between contact force updates for penalty contact formulations. This option can provide a significant speed-up of the contact treatment. If used, values exceeding 3 or 4 are dangerous. Considerable care must be exercised when using this option, as this option assumes that contact does not change FRCFRG cycles. EQ.0: FRCFRG is set to 1 and force calculations are performed each cycle-strongly recommended.

**Remark:**

Setting SOFT=2 on optional contact card A activates an alternate penalty formulation contact algorithm for general shell and solid element contact. This option is available for all SURFACE\_TO\_SURFACE, ONE\_WAY\_SURFACE\_TO\_SURFACE, and SINGLE\_SURFACE options except AIRBAG\_SINGLE\_SURFACE. When the AUTOMATIC option is used, orientation of shell segment normals is automatic. When the AUTOMATIC option is not used, the segment or element orientations are used as input. The alternate penalty formulation contact algorithm checks for segments vs. segment penetration rather than node vs. segment. After penetrating segments are found, an automatic judgment is made as to which is the master segment, and penalty forces are applied normal to that segment. The user may override this automatic judgment by using the ONE\_WAY options in which case the master segment normals are used as input by the user. The EDGE parameter on optional card A is used to enable a segment edge to segment edge penetration check. Setting EDGE=0 disables this check and is recommended when edge penetrations are not likely to occur. Setting EDGE>0 enables the edge-edge penetration judgment and EDGE=1 is recommended. Smaller values may be tried if problems occur when the EDGE option is active. In this version, all parameters on the first three cards are active except for VC, and VSF. Only the SOFT and EDGE parameters on optional card A are active. Only the ISYM parameter on optional card B is active.

# \*CONTACT

---

## Optional Card B

**Reminder:** If Optional Card B is used, then Optional Card A must be defined. (Optional Card A may be a blank line).

Optional Card B	1	2	3	4	5	6	7	8
Variable	PENMAX	THKOPT	SHLTHK	SNLOG	ISYM	I2D3D		
Type	F	I	I	I	I	I		
Default	0	0	0	0	0	0		
Remarks		Old types 3, 5, 10	Old types 3, 5, 10					

### VARIABLE

### DESCRIPTION

**PENMAX**

Maximum penetration distance for old type 3, 5, 8, 9, and 10 contact or the segment thickness multiplied by PENMAX defines the maximum penetration allowed (as a multiple of the segment thickness) for contact types a 3, a 5, a10, 13, 15, and 26. (see discussion at end of section, including Table 6.1):

EQ.0.0 for old type contacts 3, 5, and 10: Use small penetration search and value calculated from thickness and XPENE, see \*CONTROL\_CONTACT.

EQ.0.0 for contact types a 3, a 5, a10, 13, and 15: Default is 0.4, or 40 percent of the segment thickness

EQ.0.0 for contact type26: Default is 200.0 times the segment thickness

**THKOPT**

Thickness option for contact types 3, 5, and 10:

EQ.0: default is taken from control card, \*CONTROL\_CONTACT,

EQ.1: thickness offsets are included,

EQ.2: thickness offsets are not included (old way).

**SHLTHK**

Define if and only if THKOPT above equals 1. Shell thickness considered in type surface to surface and node to surface type contact options, where options 1 and 2 below activate the new contact algorithms. The thickness offsets are always included in single surface and constraint method contact types:

EQ.0: thickness is not considered,

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

EQ.2: thickness is considered including rigid bodies.

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
SNLOG	Disable shooting node logic in thickness offset contact. With the shooting node logic enabled, the first cycle that a slave node penetrates a master segment, that node is moved back to the master surface without applying any contact force. EQ.0: logic is enabled (default), EQ.1: logic is skipped (sometimes recommended for metalforming calculations).
ISYM	Symmetry plane option: EQ.0: off, EQ.1: do not include faces with normal boundary constraints (e.g., segments of brick elements on a symmetry plane). This option is important to retain the correct boundary conditions in the model with symmetry. For the <u>_ERODING_</u> contacts this option may also be defined on card 4.
I2D3D	Segment searching option: EQ.0: search 2D elements (shells) before 3D elements (solids, thick shells) when locating segments. EQ.1: search 3D (solids, thick shells) elements before 2D elements (shells) when locating segments.

## **Remarks:**

### 1. TIED\_NODES\_TO\_SURFACE

TIED\_SHELL\_EDGE\_TO\_SURFACE

TIED\_SURFACE\_TO\_SURFACE

These contact definitions are based on constraint equations and will not work with rigid bodies. However, tied interfaces with the offset option can be used with rigid bodies, i.e.,

TIED\_NODES\_TO\_SURFACE\_OFFSET

TIED\_SHELL\_EDGE\_TO\_SURFACE\_OFFSET

TIED\_SURFACE\_TO\_SURFACE\_OFFSET

Also, it may sometimes be advantageous to use the CONSTRAINED\_EXTRA\_NODE\_OPTION instead for tying deformable nodes to rigid bodies since in this latter case the tied nodes may be an arbitrary distance away from the rigid body.

Tying will only work if the surfaces are near each other. The criteria used to determine whether a slave node is tied down is that it must be "close". For shell elements "close" is defined as a distance,  $\delta$ , less than:

# \*CONTACT

---

$$\delta_1 = 0.60 * (\textit{thickness\_slave\_node} + \textit{thickness\_master\_segment})$$

$$\delta_2 = 0.05 * \min(\textit{master\_segment\_diagonals})$$

$$\delta = \max(\delta_1, \delta_2)$$

If a node is further away it will not be tied and a warning message will be printed.

## 2. CONSTRAINT\_NODES\_TO\_SURFACE

### CONSTRAINT\_SURFACE\_TO\_SURFACE

These contact definitions must be used with care. The surface and the nodes which are constrained to a surface are not allowed to be used in any other CONSTRAINT\_... contact definition. If, however, contact has to be defined from both sides as in sheetmetalforming, one of these contact definitions can be a CONSTRAINT\_ type; the other one could be a standard penalty type such as SURFACE\_TO\_SURFACE or NODES\_TO\_SURFACE.

## 3. AIRBAG\_SINGLE\_SURFACE

### AUTOMATIC\_GENERAL

### AUTOMATIC\_NODES\_TO\_SURFACE

### AUTOMATIC\_ONE\_WAY\_SURFACE\_TO\_SURFACE

### AUTOMATIC\_SINGLE\_SURFACE

### AUTOMATIC\_SURFACE\_TO\_SURFACE

### SINGLE\_SURFACE

These contact definitions require thickness to be taken into account for rigid bodies modeled with shell elements. Therefore, care should be taken to ensure that realistic thicknesses are specified for the rigid body shells. A thickness that is too small may result in loss of contact and an unrealistically large thickness may result in a degradation in speed during the bucket sorts as well as nonphysical behavior. The SHLTHK option on the \*CONTROL\_CONTACT card is ignored for these contact types.

## 4. Two methods are used in LS-DYNA for projecting the contact surface to account for shell thicknesses. The choice of methods can influence the accuracy and cost of the calculation. Segment based projection is used in contact types:

### AIRBAG\_SINGLE\_SURFACE

### AUTOMATIC\_GENERAL

### AUTOMATIC\_NODES\_TO\_SURFACE

### AUTOMATIC\_ONE\_WAY\_SURFACE\_TO\_SURFACE

### AUTOMATIC\_SINGLE\_SURFACE

AUTOMATIC\_SURFACE\_TO\_SURFACE

FORMING\_NODES\_TO\_SURFACE

FORMING\_ONE\_WAY\_SURFACE\_TO\_SURFACE

FORMING\_SURFACE\_TO\_SURFACE

The remaining contact types use nodal normal projections if projections are used. The main advantage of nodal projections is that a continuous contact surface is obtained which is much more accurate in applications such as metal forming. The disadvantages of nodal projections are the higher costs due to the nodal normal calculations, difficulties in treating T-intersections and other geometric complications, and the need for consistent orientation of contact surface segments. The contact type:

SINGLE\_SURFACE

uses nodal normal projections and consequently is slower than the alternatives.

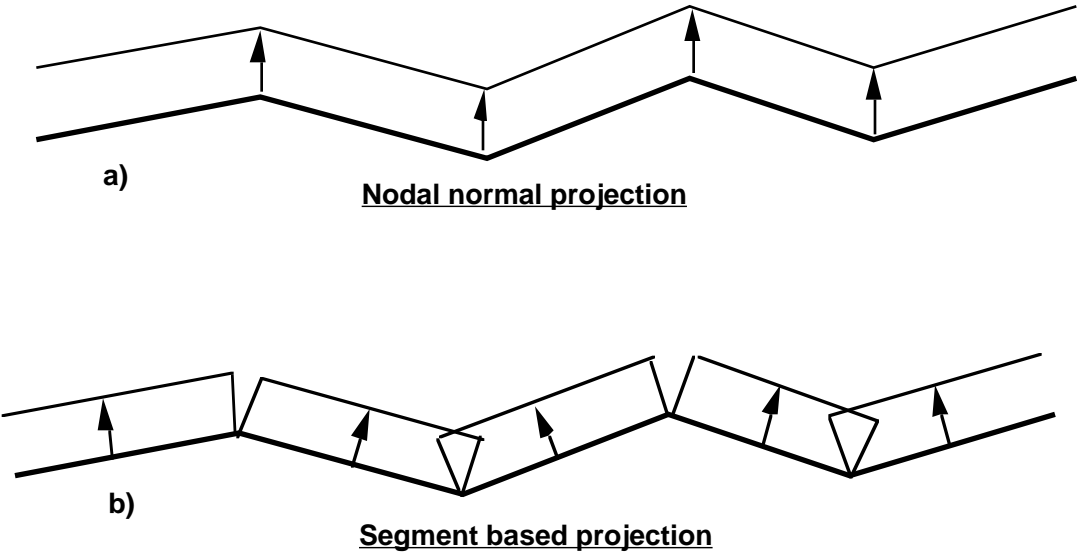
5. FORCE\_TRANSDUCER\_PENALTY  
FORCE\_TRANSDUCER\_CONSTRAINT

This contact allows the total contact forces applied by all contacts to be picked up. This contact does not apply any force to the model. Only the slave set and slave set type need be defined for this contact type. Generally, only the first three cards are defined. The force transducer option, `_PENALTY`, works with penalty type contact algorithms only, i.e., it does not work with the `CONSTRAINT` or `TIED` options. For these latter options, use the `_CONSTRAINT` option.

6. FORMING\_... These contacts are mainly used for metal forming applications. A connected mesh is not required for the master (tooling) side but the orientation of the mesh **must** be in the same direction. These contact types are based on the `AUTOMATIC` type contacts and consequently the performance is better than the original two surface contacts.

# \*CONTACT

---



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



INTERFACE TYPE ID	PENCHK	ELEMENT TYPE	FORMULA FOR RELEASE OF PENETRATING NODAL POINT
1, 2, 6, 7	———	———	-----
3, 5, 8, 9, 10 (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 6.1.** Criterion for node release for nodal points which have penetrated too far. Larger penalty stiffnesses are recommended for the contact interface which allows nodes to be released. For node-to-surface type contacts (5, 5a) the element thicknesses which contain the node determines the nodal thickness. The parameter is defined on the \*CONTROL\_CONTACT input.

# \*CONTACT

---

The keyword options for the contact type and the corresponding Version 92X, 93X, 94X type numbers are:

STRUCTURED INPUT TYPE ID	KEYWORD NAME
a13	AIRBAG_SINGLE_SURFACE
26	AUTOMATIC_GENERAL
a 5	AUTOMATIC_NODES_TO_SURFACE
a10	AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE
13	AUTOMATIC_SINGLE_SURFACE
a 3	AUTOMATIC_SURFACE_TO_SURFACE
18	CONSTRAINT_NODES_TO_SURFACE
17	CONSTRAINT_SURFACE_TO_SURFACE
23	DRAWBEAD
16	ERODING_NODES_TO_SURFACE
14	ERODING_SURFACE_TO_SURFACE
15	ERODING_SINGLE_SURFACE
27	FORCE_TRANSDUCER_CONSTRAINT
25	FORCE_TRANSDUCER_PENALTY
m 5	FORMING_NODES_TO_SURFACE
m10	FORMING_ONE_WAY_SURFACE_TO_SURFACE
m 3	FORMING_SURFACE_TO_SURFACE
5	NODES_TO_SURFACE
10	ONE_WAY_SURFACE_TO_SURFACE
20	RIGID_NODES_TO_RIGID_BODY
21	RIGID_BODY_ONE_WAY_TO_RIGID_BODY
19	RIGID_BODY_TWO_WAY_TO_RIGID_BODY
22	SINGLE_EDGE
4	SINGLE_SURFACE
1	SLIDING_ONLY
p 1	SLIDING_ONLY_PENALTY
3	SURFACE_TO_SURFACE
8	TIEBREAK_NODES_TO_SURFACE
9	TIEBREAK_SURFACE_TO_SURFACE

<u>STRUCTURED INPUT TYPE ID</u>	<u>KEYWORD NAME</u>
6	TIED_NODES_TO_SURFACE
o 6	TIED_NODES_TO_SURFACE_OFFSET
7	TIED_SHELL_EDGE_TO_SURFACE
o 7	TIED_SHELL_EDGE_TO_SURFACE_OFFSET
2	TIED_SURFACE_TO_SURFACE
o 2	TIED_SURFACE_TO_SURFACE_OFFSET

# \*CONTACT

---

## CONTACT EXAMPLES

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$ *CONTACT_NODES_TO_SURFACE
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$ Make a simple contact that prevents the nodes in part 2 from
$ penetrating the segments in part 3.
$
*CONTACT_NODES_TO_SURFACE
$
$.>...1.>...2.>...3.>...4.>...5.>...6.>...7.>...8
$ ssid msid sstyp mstyp sboxid mboxid spr mpr
$ 2 3 3 3 vdc penchk bt dt
$
$ fs fd dc vc vdc penchk bt dt
$
$ sfs sfm sst mst sfst sfmt fsf vsf
$
$ sstype, mstype = 3 id's specified in ssid and msid are parts
$ ssid = 2 use slave nodes in part 2
$ msid = 3 use master segments in part 3
$
$ Use defaults for all parameters.
$
$$$$ Optional Cards A and B not specified (default values will be used).
$
$
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```



# \*CONTACT

---

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$ *CONTACT_DRAWBEAD
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$ Define a draw bead contact:
$ - the draw bead is to be made from the nodes specified in node set 2
$ - the master segments are to be those found in the box defined by box 2
$   that are in part 18
$ - include slave and master forces in interface file (spr, mpr = 1)
$
*CONTACT_DRAWBEAD
$
$.>...1.>...2.>...3.>...4.>...5.>...6.>...7.>...8
$  ssid      msid      sstyp      mstyp      sboxid      mboxid      spr      mpr
$          2         18         4         3         2         1         1
$
$          fs         fd         dc         vc         vdc        penchk      bt         dt
$        0.10
$
$          sfs        sfm        sst        mst        sfst       sfmt        fsf        vsf
$
$
$$$$ Card 4 required because it's a drawbead contact
$
$  lcdidrf   lcdidnf   dbdth   dfsc1   numint
$          3          0.17436  2.0
$
$  lcdidrf = 3 load curve 3 specifies the bending component of the
$              restraining force per unit draw bead length
$  dbdth = 0.17436 drwa bead depth
$  dfsc1 = 2.0 scale load curve 3 (lcdidrf) by 2
$
$$$$ Optional Cards A and B not specified (default values will be used).
$
$
*DEFINE_BOX
$  boxid      xmm      xmx      ymn      ymx      zmn      zmx
$          2  0.000E+00  6.000E+00  6.000E+00  1.000E+02 -1.000E+03  1.000E+03
$
*SET_NODE_LIST
$  sid      da1      da2      da3      da4
$          2
$  nid1     nid2     nid3     nid4     nid5     nid6     nid7     nid8
$  2580     2581     2582     2583     2584     2585     2586     2587
$  2588     2589     2590
$
*DEFINE_CURVE
$  lcd      sidr      scla      sclo      offa      offo
$          3
$
$          a          o
$          DEPTH      FORC/LGTH
$          0.000E+00      0.000E+00
$          1.200E-01      1.300E+02
$          1.500E-01      2.000E+02
$          1.800E-01      5.000E+02
```

## \*CONTACT\_ENTITY

Purpose: Define a contact entity. Geometric contact entities treat the impact between a deformable body defined as a set of slave nodes or nodes in a shell part set and a rigid body. The shape of the rigid body is determined by attaching geometric entities. Contact is treated between these geometric entities and the slave nodes using a penalty formulation. The penalty stiffness is optionally maximized within the constraint of the Courant criterion. As an alternative, a finite element mesh made with shells can be used as geometric entity. Also, axisymmetric entities with arbitrary shape made with multilinear polygons are possible. The latter is particularly useful for metalforming simulations.

WARNING: If the problem being simulated involves dynamic motion of the entity, care should be taken to insure that the inertial properties of the entity are correct. It may be necessary to use the \*PART\_INERTIA option to specify these properties.

### Define 5 cards for the contact entity definition below.

#### Card 1 Format

Card 1            1            2            3            4            5            6            7            8

Variable	PID	GEOTYP	SSID	SSTYP	SF	DF	CF	INTORD
Type	I	I	I	I	F	F	F	I
Default	required	required	required	0	1.	0.	0.	0

#### VARIABLE

#### DESCRIPTION

PID

Part ID of the rigid body to which the geometric entity is attached, see \*PART.

GEOTYP

Type of geometric entity:

EQ.1: plane,

EQ.2: sphere,

EQ.3: cylinder,

EQ.4: ellipsoid,

EQ.5: torus,

EQ.6: CAL3D/MADYMO Plane, see Appendix F,

EQ.7: CAL3D/MADYMO Ellipsoid, see Appendix F,

EQ.8: VDA surface, see Appendix I,

# \*CONTACT

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
	EQ.9: rigid body finite element mesh (shells only), EQ.10: finite plane, EQ.11: load curve defining line as surface profile of axisymmetric rigid bodies.
SSID	Slave set ID, see *SET_NODE_OPTION, *PART, or *SET_PART.
SSTYP	Slave set type: EQ.0: node set, EQ.1: part ID, EQ.2: part set ID.
SF	Penalty scale factor. Useful to scale maximized penalty.
DF	Damping option, see description for *CONTACT_OPTION: EQ..0: no damping, GT..0: viscous damping in percent of critical, e.g., 20 for 20% damping, EQ.-n:  n  is the load curve ID giving the damping force versus relative normal velocity (see remark 1 below).
CF	Coulomb friction coefficient. Assumed to be constant.
INTORD	Integration order (slaved materials only). This option is not available with entity types 8 and 9 where only nodes are checked: EQ.0: check nodes only, EQ.1: 1 point integration over segments, EQ.2: 2×2 integration, EQ.3: 3×3 integration, EQ.4: 4×4 integration, EQ.5: 5×5 integration. This option allows a check of the penetration of the rigid body into the deformable (slaved) material. Then virtual nodes at the location of the integration points are checked.

## Remark:

1. The optional load curves that are defined for damping versus relative normal velocity and for force versus normal penetration should be defined in the positive quadrant. The sign for the damping force depends on the direction of the relative velocity and the treatment is symmetric if the damping curve is in the positive quadrant. If the damping force is defined in the negative and positive quadrants, the sign of the relative velocity is used in the table look-up.



**Card 2 Format**

Card 2            1            2            3            4            5            6            7            8

Variable	BT	DT	SO	GO				
Type	F	F	I	I				
Default	0.	1.E+20	0	0				

---

**VARIABLE**

---

**DESCRIPTION**

BT	Birth time
DT	Death time
SO	Flag to use penalty stiffness as in surface to surface contact: EQ.0: contact entity stiffness formulation, EQ.1: surface to surface contact method, EQ.-n:  n  is the load curve ID giving the force versus the normal penetration.
GO	Flag for mesh generation of the contact entity for entity types 1-5 and 10-11. This is used for visualization in post-processing only. EQ.0: mesh is not generated, EQ.1: mesh is generated.

# \*CONTACT

---

## Cards 3 and 4 Format

Card 3            1            2            3            4            5            6            7            8

Variable	XC	YC	ZC	AX	AY	AZ		
Type	F	F	F	F	F	F		
Default	0.	0.	0.	0.	0.	0		

Card 4            1            2            3            4            5            6            7            8

Variable	BX	BY	BZ					
Type	F	F	F					
Default	0.	0.	0.					

**VARIABLE**

**DESCRIPTION**

- XC            x-center,  $x_c$ , see remarks below.
- YC            y-center,  $y_c$ , see remarks below.
- ZC            z-center,  $z_c$ . See remarks below.
- AX            x-direction for local axis A,  $A_x$ , see remarks below.
- AY            y-direction for local axis A,  $A_y$ , see remarks below.
- AZ            z-direction for local axis A,  $A_z$ , see remarks below.
- BX            x-direction for local axis B,  $B_x$ , see remarks below.
- BX            x-direction for local axis B,  $B_x$ , see remarks below.
- BY            y-direction for local axis B,  $B_y$ , see remarks below.
- BZ            z-direction for local axis B,  $B_z$ , see remarks below.

**Remarks:**

1. The coordinates,  $(x_c, y_c, z_c)$ , are the positions of the local origin of the geometric entity in global coordinates. The entity's local A-axis is determined by the vector  $(A_x, A_y, A_z)$  and the local B-axis by the vector  $(B_x, B_y, B_z)$ .
2. Cards 3 and 4 define a local to global transformation. The geometric contact entities are defined in a local system and transformed into the global system. For the ellipsoid, this is necessary because it has a restricted definition for the local position. For the plane, sphere, and cylinder, the entities can be defined in the global system and the transformation becomes  $(x_c, y_c, z_c)=(0,0,0)$ ,  $(A_x, A_y, A_z)=(1,0,0)$ , and  $(B_x, B_y, B_z)=(0,1,0)$ .

# \*CONTACT

---

## Card 5 Format

Card 5            1            2            3            4            5            6            7            8

Variable	INOUT	G1	G2	G3	G4	G5	G6	G7
Type	I	F	F	F	F	F	F	F
Default	0	0.	0.	0.	0.	0.	0.	0.

---

### VARIABLE

### DESCRIPTION

---

INOUT	In-out flag. Allows contact from the inside or the outside (default) of the entity: EQ.0: slave nodes exist outside of the entity, EQ.1: slave nodes exist inside the entity.
G1	Entity coefficient $g_1$ (CAL3D/MADYMO plane or ellipse number) for coupled analysis (see Appendix F).
G2	Entity coefficient $g_2$ , see remarks below.
G3	Entity coefficient $g_3$ , see remarks below.
G4	Entity coefficient $g_4$ , see remarks below.
G5	Entity coefficient $g_5$ , see remarks below.
G6	Entity coefficient $g_6$ , see remarks below.
G7	Entity coefficient $g_7$ , see remarks below.

### Remarks:

Figures 6.4a and 6.4b show the definitions of the geometric contact entities. The relationships between the entity coefficients and the Figure 6.4a and 6.4b variables are as follows (please note that  $(P_x, P_y, P_z)$  is a position vector and that  $(Q_x, Q_y, Q_z)$  is a direction vector):

$$\begin{aligned}
 \text{GEOTYP} = 1: \quad & g_1 = P_x & g_4 = Q_x \\
 & g_2 = P_y & g_5 = Q_y \\
 & g_3 = P_z & g_6 = Q_z \\
 & & g_7 = L
 \end{aligned}$$

If automatic generation is used, a square plane of length L on each edge is generated which represents the infinite plane. If generation is inactive, then  $g_7$  may be ignored.

GEOTYP = 2: g1 = Px g4 = r

g2 = Py

g3 = Pz

GEOTYP = 3: g1 = Px g4 = Qx

g2 = Py g5 = Qy

g3 = Pz g6 = Qz

g7 = r

If automatic generation is used, a cylinder of length  $\sqrt{Qx^2 + Qy^2 + Qz^2}$  and radius r is generated which represents the infinite cylinder.

GEOTYP = 4: g1 = Px g4 = a

g2 = Py g5 = b

g3 = Pz g6 = c

g7 = n (order of the ellipsoid)

GEOTYP = 5: g1 = Radius of torus

g2 = r

g3 = number of elements along minor circumference

g4 = number of elements along major circumference

GEOTYP = 8: g1 = Blank thickness (option to override true thickness)

g2 = Scale factor for true thickness (optional)

g3 = Load curve ID defining thickness versus time. (optional)

GEOTYP = 9: g1 = Shell thickness (option to override true thickness).

**NOTE: The shell thickness specification is necessary if the slave surface is generated from solid elements.**

g2 = Scale factor for true thickness (optional)

g3 = Load curve ID defining thickness versus time. (optional)

GEOTYP =10: g1 = Length of edge along X' axis

g2 = Length of edge along Y' axis

# \*CONTACT

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

g2 = Number of elements along circumference

EQ.0: default set to 10

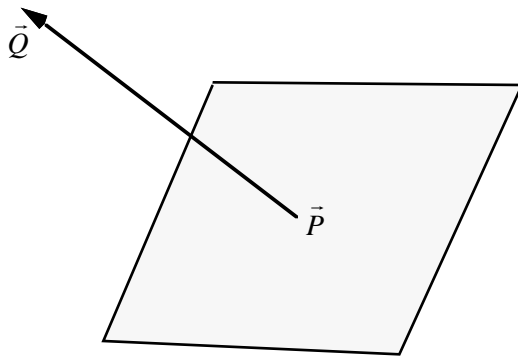
g3 = Number of elements along axis

EQ.0: default set to 20

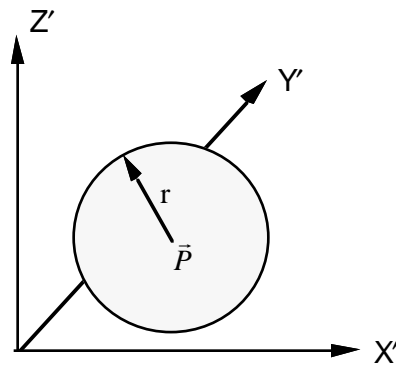
EQ.-1: the elements are generated from the points on the load curve

g4 = Number of sub divisions on load curve used to calculate contact

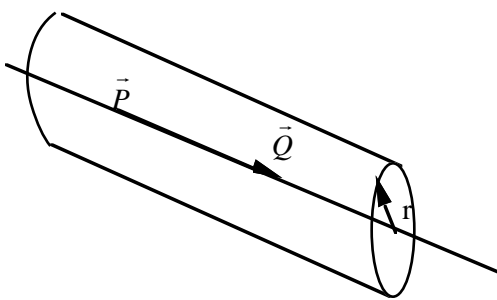
EQ:0 default set to 1000



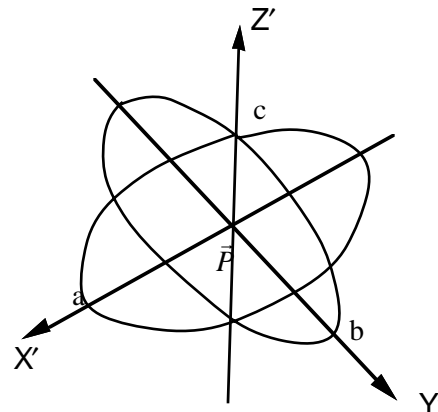
IGTYPE= 1: Infinite Plane



IGTYPE= 2: Sphere



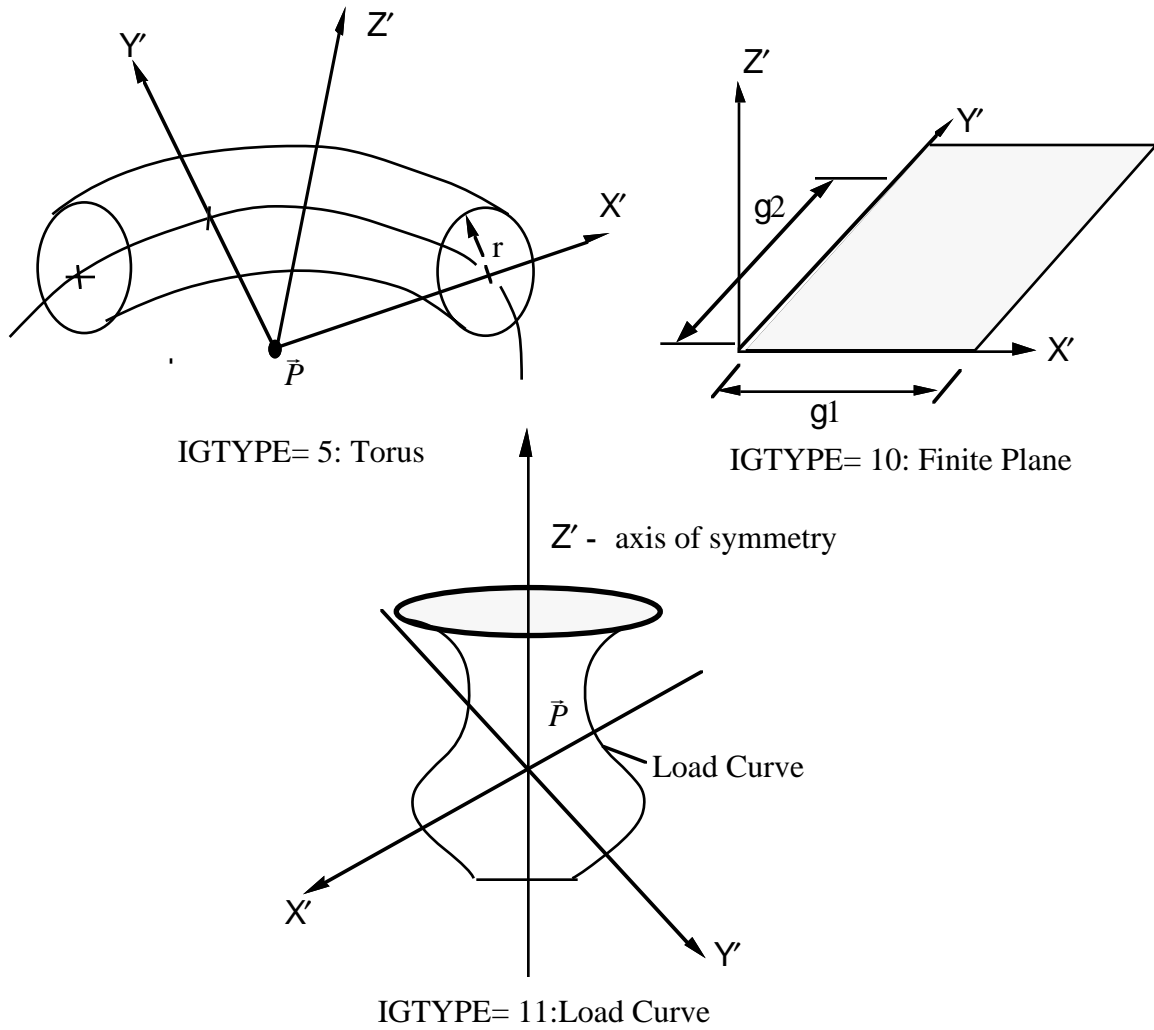
IGTYPE= 3: Infinite Cylinder



$$\left(\frac{X'}{a}\right)^n + \left(\frac{Y'}{b}\right)^n + \left(\frac{Z'}{c}\right)^n = 1$$

IGTYPE= 4: Hyperellipsoid

**Figure 6.4a.** Contact Entities.



**Figure 6.4b.** Contact Entities.

# **\*CONTACT**

---

## **\*CONTACT\_GEBOD\_OPTION**

Purpose: Define contact interaction between the segment of a GEBOD dummy and parts or nodes of the finite element model. This implementation follows that of the contact entity, however, it is specialized for the dummies. Forces may be output using the \*DATABASE\_GCEOUT command. See \*COMPONENT\_GEBOD and Appendix K for further details.

Conventional \*CONTACT\_OPTION treatment (surface-to-surface, nodes-to-surface, etc.) can also be applied to the segments of a dummy. To use this approach it is first necessary to determine part ID assignments by running the model through LSDYNA's initialization phase.

The following options are available and refer to the ellipsoids which comprise the dummy. Options involving **HAND** are not applicable for the child dummy since its lower arm and hand share a common ellipsoid.

**LOWER\_TORSO**  
**MIDDLE\_TORSO**  
**UPPER\_TORSO**  
**NECK**  
**HEAD**  
**LEFT\_SHOULDER**  
**RIGHT\_SHOULDER**  
**LEFT\_UPPER\_ARM**  
**RIGHT\_UPPER\_ARM**  
**LEFT\_LOWER\_ARM**  
**RIGHT\_LOWER\_ARM**  
**LEFT\_HAND**  
**RIGHT\_HAND**  
**LEFT\_UPPER\_LEG**  
**RIGHT\_UPPER\_LEG**  
**LEFT\_LOWER\_LEG**  
**RIGHT\_LOWER\_LEG**  
**LEFT\_FOOT**  
**RIGHT\_FOOT**



**Card 1 Format**

Card 1            1            2            3            4            5            6            7            8

Variable	DID	SSID	SSTYP	SF	DF	CF	INTORD	
Type	I	I	I	F	F	F	I	
Default	required	required	required	1.	20.	0.5	0	

**VARIABLE****DESCRIPTION**

DID	Dummy ID, see *COMPONENT_GEBOD_OPTION.
SSID	Slave set ID, see *SET_NODE_OPTION, *PART, or *SET_PART.
SSTYP	Slave set type: EQ.0: node set, EQ.1: part ID, EQ.2: part set ID.
SF	Penalty scale factor. Useful to scale maximized penalty.
DF	Damping option, see description for *CONTACT_OPTION: EQ..0: no damping, GT..0: viscous damping in percent of critical, e.g., 20 for 20% damping, EQ.-n:  n  is the load curve ID giving the damping force versus relative normal velocity (see remark 1 below).
CF	Coulomb friction coefficient (see remark 2 below). Assumed to be constant.
INTORD	Integration order (slaved materials only). EQ.0: check nodes only, EQ.1: 1 point integration over segments, EQ.2: 2×2 integration, EQ.3: 3×3 integration, EQ.4: 4×4 integration, EQ.5: 5×5 integration. This option allows a check of the penetration of the dummy segment into the deformable (slaved) material. Then virtual nodes at the location of the integration points are checked.

# \*CONTACT

---

## Card 2 Format

Card 2            1            2            3            4            5            6            7            8

Variable	BT	DT	SO					
Type	F	F	I					
Default	0.	1.E+20	0					

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
BT	Birth time
DT	Death time
SO	Flag to use penalty stiffness as in surface to surface contact: EQ.0: contact entity stiffness formulation, EQ.1: surface to surface contact method, EQ.-n:  n  is the load curve ID giving the force versus the normal penetration.

### Remarks:

1. The optional load curves that are defined for damping versus relative normal velocity and for force versus normal penetration should be defined in the positive quadrant. The sign for the damping force depends on the direction of the relative velocity and the treatment is symmetric if the damping curve is in the positive quadrant. If the damping force is defined in the negative and positive quadrants, the sign of the relative velocity is used in the table look-up.
2. Insofar as these ellipsoidal contact surfaces are continuous and smooth it may be necessary to specify Coulomb friction values larger than those typically used with faceted contact surfaces.

**\*CONTACT\_INTERIOR**

Purpose: Define interior contact for foam brick elements. Frequently, when foam materials are compressed under high pressure, the solid elements used to discretize these materials may invert leading to negative volumes and error terminations. In order to keep these elements from inverting, it is possible to consider interior contacts within the foam between layers of interior surfaces made up of the faces of the solid elements. Since these interior surfaces are generated automatically, the part (material) ID's for the materials of interest are defined here, prior to the interface definitions. ONLY ONE PART SET ID CAN BE DEFINED.

**Card Format**

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

---

**VARIABLE****DESCRIPTION**

---

PSID

Part set ID including all parts for which interior contact is desired.

Three attributes should be defined for the part set:

Attribute 1: PSF, penalty scale factor (Default=1.00).

Attribute 2: Activation factor,  $F_a$  (Default=0.10). When the crushing of the element reaches  $F_a$  times the initial thickness the contact algorithm begins to act.

Attribute 3: ED, Optional modulus for interior contact stiffness.

**Remarks:**

The interior penalty is determined by the formula:

$$K = \frac{SLSFAC \cdot PSF \cdot Volume^{\frac{2}{3}} \cdot E}{Min. Thickness}$$

## \*CONTACT

---

where SLSFAC is the value specified on the \*CONTROL\_CONTACT card , volume is the volume of the brick element, E is a constitutive modulus, and min. thickness is approximately the thickness of the solid element through its thinnest dimension. If ED, is defined above the interior penalty is then given instead by:

$$K = \frac{Volume^{\frac{2}{3}} \cdot ED}{Min. Thickness}$$

where the scaling factors are ignored. Generally, ED should be taken as the locking modulus specified for the foam constitutive model.

Caution should be observed when using this option since if the time step size is too large an instability may result. The time step size is not affected by the use of interior contact.

**\*CONTACT\_1D**

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

**Card Format**

Card 1	1	2	3	4	5	6	7	8
Variable	NSIDS	NSIDM	ERR	SIGC	GB	SMAX	EXP	
Type	I	I	F	F	F	F	F	
Default	none	none	0.	0.	0.	0.	0.	

**VARIABLE****DESCRIPTION**

NSIDS	Nodal set ID for the slave nodes, see *SET_NODE.
NSIDM	Nodal set ID for the master nodes, see *SET_NODE.
ERR	External radius of rebar
SIGC	Compressive strength of concrete
GB	Bond shear modulus
SMAX	Maximum shear strain displacement
EXP	Exponent in damage curve

**Remarks:**

With this option the concrete is defined with solid elements and the rebar with truss elements, each with their own unique set of nodal points. A string of consecutive nodes, called slave nodes, related to the truss elements may slide along a string of consecutive nodes, called master nodes, related to the solid elements. The sliding commences after the rebar debonds.

The bond between the rebar and concrete is assumed to be elastic perfectly plastic. The maximum allowable slip strain is given as:

$$u_{max} = SMAX \cdot e^{-EXP \cdot D}$$

where D is the damage parameter  $D_{n+1} = D_n + \Delta u$ . The shear force at time  $n+1$  is given as:

$$f_{n+1} = \min(f_n - GB \cdot \pi \cdot ERR^2 \cdot \Delta u, G \cdot \pi \cdot ERR^2 \cdot u_{max})$$

# \*CONTACT

---

## \*CONTACT\_2D\_OPTION1\_{OPTION2}

Purpose : Define a 2-dimensional contact or slide line. This option is to be used with 2D solid and shell elements using the plane\_stress, plane\_strain or axisymmetric formulations, see \*SECTION\_SHELL,

*OPTION1* specifies the contact type. The following options should be used with deformable materials only (i.e., not rigid):

**SLIDING\_ONLY**  
**TIED\_SLIDING**  
**SLIDING\_VOIDS**

since these methods are based on the imposition of constraints. The constraint methods may be used with rigid bodies if the rigid body is the master surface and all rigid body motions are prescribed. The following options may be used with rigid materials as well:

**PENALTY\_FRICTION**  
**PENALTY**  
**AUTOMATIC\_SINGLE\_SURFACE**  
**AUTOMATIC\_SURFACE\_TO\_SURFACE**  
**AUTOMATIC\_NODE\_TO\_SURFACE**  
**AUTOMATIC\_SURFACE\_IN\_CONTINUUM**

*OPTION2* specifies that the first card to read defines the title and ID number of contact interface and takes the single option:

**TITLE**

**Note:** *OPTION1* and *OPTION2* may appear in any order.

At present, the contact ID number and title are ignored by LS-DYNA but are included for extension in the near future. The title card is picked up by some of the peripheral LS-DYNA codes to aid in post-processing.

Single surface contact in two dimensions is accomplished by the AUTOMATIC\_SURFACE\_TO\_SURFACE option when the master surface part set is set to zero. The SINGLE\_SURFACE option in version 940 has been removed.

# \*CONTACT

---

Read the following card here if and only if the option **TITLE** is specified:

Optional	1	2
Variable	CID	NAME
Type	I	A70

**For all options except the AUTOMATIC options, define the following two cards.**

**Card 1 Format**

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

**Card 2 Format**

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

# \*CONTACT

---

For the PENALTY\_FRICTION option define the following additional card

Card 3	1	2	3	4	5	6	7	8
Variable	FRIC	FRIC_L	FRIC_H	FRIC_S				
Type	F	F	F	F				
Default	none	none	none	none				

<u>VARIABLE</u>	<u>DESCRIPTION</u>
SSID	Nodal set ID for the slave nodes, see *SET_NODE. <b>The slave surface must be to the left of the master surface.</b>
MSID	Nodal set ID for the master nodes, see *SET_NODE.
EXT_PAS	Slideline extension bypass option. EQ:0 extensions are use EQ:1 extensions are not used
THETA1	Angle in degrees of slideline extension at first master node. EQ:0 extension remains tangent to first master segment.
THETA2	Angle in degrees of slideline extension at last master node. EQ:0 extension remains tangent to first master segment.
TOL_IG	Tolerance for determing initial gaps. EQ:0.0 default set to 0.001
PEN	Scale factor or penalty. EQ:0.0 default set to 0.10
FRIC	Coefficient of friction
FRIC_L	Coefficient of friction at low velocity.
FRIC_H	Coefficient of friction at high velocity.
FRIC_S	Friction factor for shear.



**For the AUTOMATIC options define the following two cards:**

Card 1	1	2	3	4	5	6	7	8
Variable	PSIDS	PSIDM	SFACT	FREQ	FS	FD	DC	MEMBS
Type	I	I	F	I	F	F	F	I
Default	none	none	1.0	50	0.	0.	0.	6
Remarks	1,2	1,2						

Card 2	1	2	3	4	5	6	7	8
Variable	TBIRTH	TDEATH	SOS	SOM	NDS	NDM	IPFLG	CPFLG
Type	F	F	F	F	I	I	I	I
Default	0.	1.e20	1.0	1.0	0	0	0	0
Remarks			3	3	4	5		

<u>VARIABLE</u>	<u>DESCRIPTION</u>
PSIDS	Part set ID to define the slave surface, see *SET_PART.
PSIDM	Part set ID to define the master surface, see *SET_PART. Do not define if single surface contact is desired.
SFACT	Scale factor for the penalty force stiffness.
FREQ	Search frequency. The number of timesteps between bucket sorts. EQ.0: default set to 50.
FS	Static coefficient of friction. The frictional coefficient is assumed to be dependent on the relative velocity $v_{rel}$ of the surfaces in contact according to the relationship given by: $\mu_c = FD + (FS - FD)e^{-DC \cdot 1v_{rel}^1}$ .

# \*CONTACT

---

VARIABLE	DESCRIPTION
FD	Dynamic coefficient of friction. The functional coefficient is assumed to be dependent on the relative velocity $v_{rel}$ of the surfaces in contact $\mu_c = FD + (FS - FD)e^{-DC v_{rel} }$
DC	Exponential decay coefficient. The functional coefficient is assumed to be dependent on the relative velocity $v_{rel}$ of the surfaces in contact $\mu_c = FD + (FS - FD)e^{-DC v_{rel} }$
MEMBS	Parameter to allocate memory for bucket sort pair information.
TBIRTH	Birth time for contact.
TDEATH	Death time for contact.
SOS	Surface offset from midline for 2D shells of slave surface EQ.0.0: default to 1. GT.0.0: scale factor applied to actual thickness LT.0.0: absolute value is used as the offset
SOM	Surface offset from midline for 2D shells of master surface EQ.0: default to 1. GT.0: scale factor applied to actual thickness LT.0: absolute value is used as the offset
NDS	Normal direction flag for 2D shells of slave surface EQ.0: Normal direction is determined automatically EQ.1: Normal direction is in the positive direction EQ.-1: Normal direction is in the negative direction
NDM	Normal direction flag for 2D shells of master surface EQ.0: Normal direction is determined automatically EQ.1: Normal direction is in the positive direction EQ.-1: Normal direction is in the negative direction
IPFLG	Initial penetration flag for explicit analysis EQ.0: Allow initial penetrations to remain EQ.1: Push apart initially penetrated surfaces
COFLG	Closing/Opening flag for implicit analysis EQ.0: Recommended for most problem where gaps are only closing. EQ.1: Recommended when gaps are opening to avoid sticking.

## **Remarks:**

Remarks 1 through 7 pertain to 2D\_AUTOMATIC contact.

1. For AUTOMATIC\_SURFACE\_TO\_SURFACE, AUTOMATIC\_SINGLE\_SURFACE contact and AUTOMATIC\_NODE\_TO\_SURFACE contact, penetration of 2D shell elements and external faces of 2D continuum elements is prevented by penalty forces. Parts in the slave part set are checked for contact with parts in the master part set. Self contact is checked for any part in both sets. If the slave part set is omitted, all parts are checked for contact. If the master part set is omitted, it is assumed to be identical to the slave part set.
2. For AUTOMATIC\_SURFACE\_IN\_CONTINUUM contact, penalty forces prevent the flow of slave element material (the continuum) through the master surfaces. Flow of the continuum tangent to the surface is permitted. Only 2D solid parts are permitted in the slave part set. Both 2D solid and 2D shell parts are permitted in the master part set. Neither the slave part set ID nor the master part set ID may be omitted.
3. By default, the true thickness of 2D shell elements is taken into account for AUTOMATIC\_SURFACE\_TO\_SURFACE and AUTOMATIC\_NODE\_TO\_SURFACE contact. The user can override the true thickness by using SOS and SOM. If the surface offset is reduced to a small value, the automatic normal direction algorithm may fail, so it is best to specify the normal direction using NDS or NDM. Thickness of 2D shell elements is not considered for AUTOMATIC\_SURFACE\_IN\_CONTINUUM contact.
4. By default, the normal direction of 2D shell elements is evaluated automatically for AUTOMATIC\_SURFACE\_TO\_SURFACE and AUTOMATIC\_NODE\_TO\_SURFACE contact. The user can override the automatic algorithm using NDS or NDM and contact will occur with the positive or negative face of the element.
5. For SURFACE\_IN\_CONTINUUM contact, flow through 2D shell elements is prevented in both directions by default. If NDM is set to  $\pm 1$ , flow in the direction of the normal is permitted.
6. When using AUTOMATIC\_SURFACE\_IN\_CONTINUUM contact, there is no need to mesh the continuum around the structure because contact is not with continuum nodes but with material in the interior of the continuum elements. The algorithm works well for Eulerian or ALE elements since the structure does not interfere with remeshing. However, a structure will usually not penetrate the surface of an ALE continuum since the nodes are Lagrangian normal to the surface. Therefore, if using an ALE fluid, the structure should be initially immersed in the fluid and remain immersed throughout the calculation. Penetrating the surface of an Eulerian continuum is not a problem.
7. For all types of 2D\_AUTOMATIC contact, eroding materials are treated by default. At present, subcycling is not possible.

The remaining discussion applies to the SLIDING\_ONLY, TIED\_SLIDING, SLIDING\_VOIDS, PENALTY\_FRICTION, and PENALTY options. These options were adopted from LS-DYNA2D and originated in the public domain version of DYNA2D from the Lawrence Livermore National Laboratory. The AUTOMATIC contact options are generally recommended excepted for the TIED option.

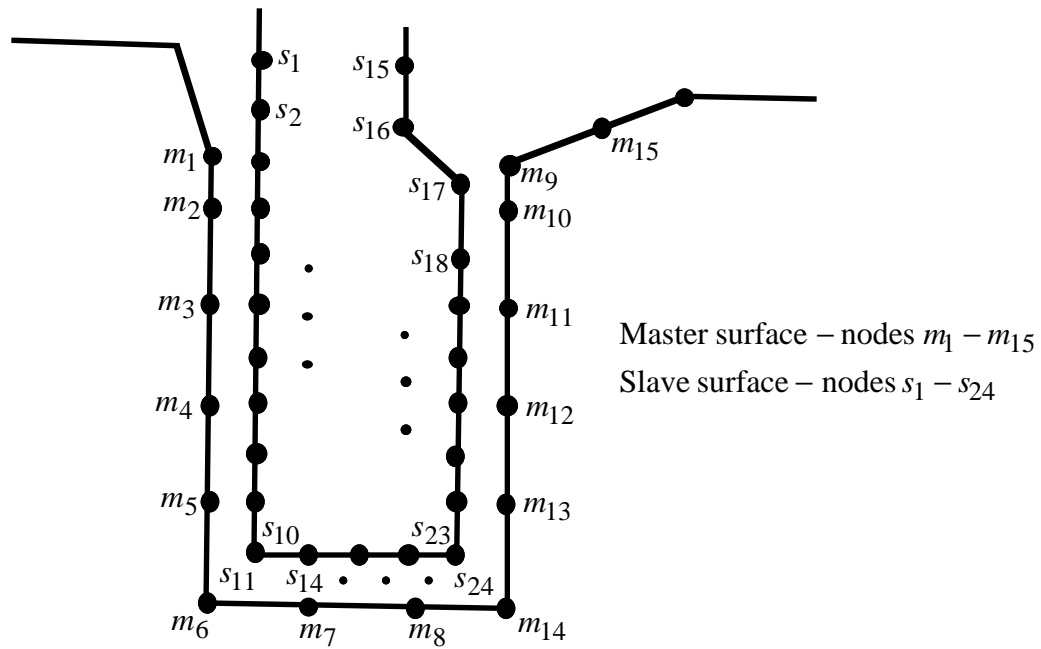
# \*CONTACT

---

Consider two slideline surfaces in contact. It is necessary to designate one as a slave surface and the other as a master surface. Nodal points defining the slave surface are called slave nodes, and similarly, nodes defining the master surface are called master nodes. Each slave-master surface combination is referred to as a slideline.

Many potential problems with the algorithm can be avoided by observing the following precautions:

- Metallic materials should contain the master surface along high explosive-metal interfaces.
- Sliding only type slidelines are appropriate along high explosive-metal interfaces. The penalty formulation is not recommended along such interfaces.
- If one surface is more finely zoned, it should be used as the slave surface. If penalty slidelines are used, PENALTY and PENALTY\_FRICTION, the slave-master distinction is irrelevant.
- A slave node may have more than one master segment, and may be included as a member of a master segment if a slideline intersection is defined.
- Angles in the master side of a slideline that approach 90° must be avoided. Whenever such angles exist in a master surface, two or more slidelines should be defined. This procedure is illustrated in Figure 6.5. An exception for the foregoing rule arises if the surfaces are tied. In this case, only one slideline is needed.
- Whenever two surfaces are in contact, the smaller of the two surfaces should be used as the slave surface. For example, in modeling a missile impacting a wall, the contact surface on the missile should be used as the slave surface.
- Care should be used when defining a master surface to prevent the extension from interfering with the solution. In Figures 6.6 and 6.7, slideline extensions are shown.

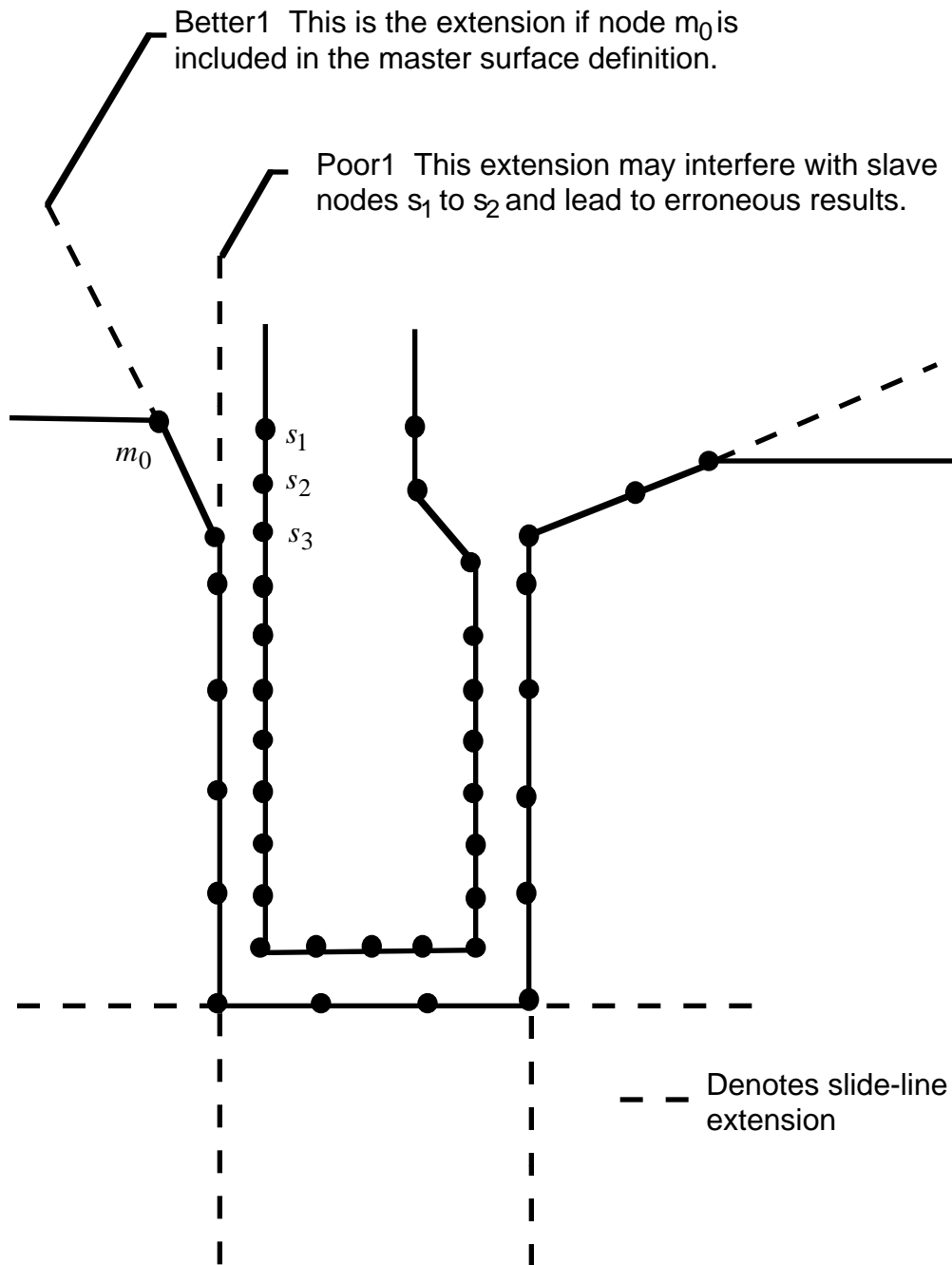


1		2		3	
Slaves	Masters	Slaves	Masters	Slaves	Masters
$s_1$	$m_1$	$s_{11}$	$m_6$	$s_{24}$	$m_{14}$
$s_2$	$m_2$	$s_{12}$	$m_7$	$s_{23}$	$m_{13}$
.	.	.	$m_8$	.	.
.	.	.	$m_{14}$	.	.
.	.	.	.	.	.
		$s_{14}$			$m_9$
$s_{11}$	$m_6$	$s_{24}$		$s_{15}$	$m_{15}$

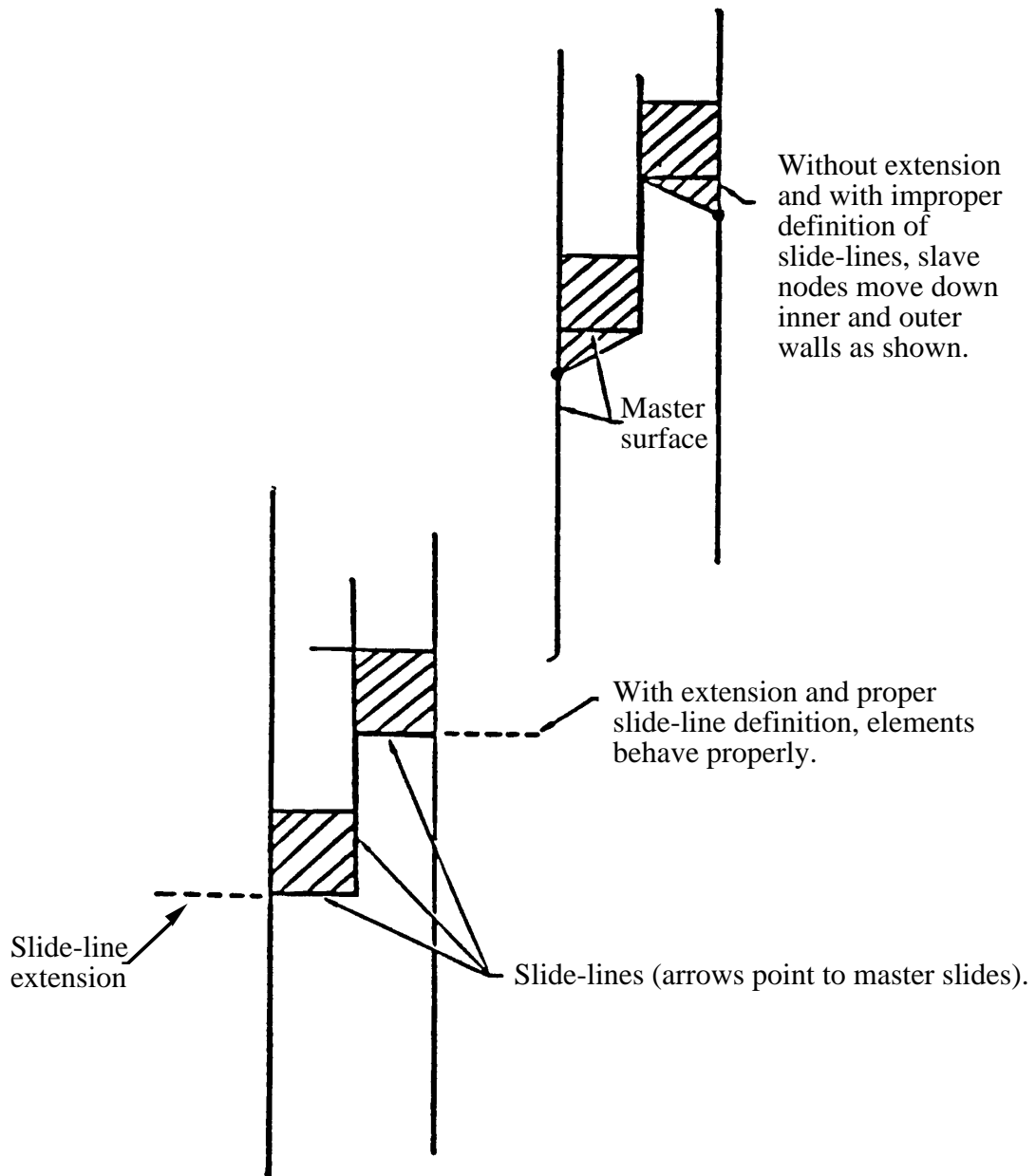
**Figure 6.5.** Proper definition of illustrated slave-master surface requires three slidelines (note that slave surface is to the left of the master surface as one moves along master nodes in order of definition).

# \*CONTACT

---



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



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

# \*CONTACT

---



# **\*CONTROL**

The keyword control cards are optional and can be used to change defaults, activate solution options such as mass scaling or adaptive remeshing; however, it is advisable to define the \*CONTROL\_TERMINATION card. **The ordering of the control cards in the input file is arbitrary. To avoid ambiguities, define no more than one control card of each type.** The following control cards are organized in an alphabetical order:

- \*CONTROL\_ACCURACY**
- \*CONTROL\_ADAPSTEP**
- \*CONTROL\_ADAPTIVE**
- \*CONTROL\_ALE**
- \*CONTROL\_BULK\_VISCOSITY**
- \*CONTROL\_CONTACT**
- \*CONTROL\_COUPLING**
- \*CONTROL\_CPU**
- \*CONTROL\_DYNAMIC\_RELAXATION**
- \*CONTROL\_ENERGY**
- \*CONTROL\_EXPLOSIVE\_SHADOW**
- \*CONTROL\_HOURLASS\_{OPTION}**
- \*CONTROL\_IMPLICIT\_AUTO**
- \*CONTROL\_IMPLICIT\_DYNAMICS**
- \*CONTROL\_IMPLICIT\_GENERAL**
- \*CONTROL\_IMPLICIT\_LINEAR**
- \*CONTROL\_IMPLICIT\_NONLINEAR**
- \*CONTROL\_IMPLICIT\_STABILIZATION**
- \*CONTROL\_OUTPUT**
- \*CONTROL\_PARALLEL**
- \*CONTROL\_RIGID**
- \*CONTROL\_SHELL**
- \*CONTROL\_SOLUTION**
- \*CONTROL\_STRUCTURED\_{OPTION}**
- \*CONTROL\_SUBCYCLE**
- \*CONTROL\_TERMINATION**

# **\*CONTROL**

---

**\*CONTROL\_THERMAL\_NONLINEAR**

**\*CONTROL\_THERMAL\_SOLVER**

**\*CONTROL\_THERMAL\_TIMESTEP**

**\*CONTROL\_TIMESTEP**

**\*CONTROL\_ACCURACY**

Purpose: Define control parameters that can improve the accuracy of the calculation.

**Card Format**

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

<u>VARIABLE</u>	<u>DESCRIPTION</u>
OSU	Objective stress update (See Remark 1 below). EQ.0: Off (default) EQ.1: On
INN	Invariant node numbering for shell element (See Remark 2 below). EQ.1: Off (default) EQ.2: On

**Remarks:**

1. Objective stress updates are occasionally necessary. Some examples include spinning bodies such as turbine blades in a jet engine, high velocity impacts generating large strains in a few time steps, and large time step sizes due to mass scaling in metal forming. There is a significant added cost which is due in part to the added cost of the second order terms in the stress update when the Jaumann rate is used and the need to compute the strain-displacement matrix at the mid-point geometry. This option is available for one point brick elements, the selective-reduced integrated brick element which uses eight integration points, the fully integrated plane strain and axisymmetric volume weighted (type 15) 2D solid elements, and the following shell elements: Belytschko-Tsay, Belyschko-Tsay with warping stiffness, Belyschko-Chiang-Wong, S/R Hughes-Liu, and the type 16 fully integrated shell element.
2. Invariant node numbering for shell elements affects the choice of the local element shell coordinate system. The orientation of the default local coordinate system is based on the shell normal vector and the direction of the 1-2 side of the element. If the element numbering is permuted, the results will change in irregularly shaped elements. With invariant node numbering, permuting the nodes shifts the local system by an exact multiple of 90 degrees. In spite of its higher costs [ $<5\%$ ], the invariant local system is recommended for several reasons. First, element forces are nearly independent of node sequencing; secondly, the hourglass modes will not substantially affect the material directions; and, finally, stable calculations over long time periods are achievable.

# \*CONTROL

---

## \*CONTROL\_ADAPSTEP

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

### Card Format

Card 1            1            2            3            4            5            6            7            8

Variable	FACTIN	DFACTR						
Type	F	F						
Default	1.0	0.01						

---

#### VARIABLE

#### DESCRIPTION

---

FACTIN	Initial relaxation factor for contact force during each adaptive remesh. To turn this option off set FACTIN=1.0. Unless stability problems occur in the contact, FACTIN=1.0 is recommended since this option can create some numerical noise in the resultant tooling forces. A typical value for this parameter is 0.10.
DFACTR	Incremental increase of FACTIN during each time step after the adaptive step. FACTIN is not allowed to exceed unity. A typical value might be 0.01.

### **Remarks:**

1. This command applies to contact with thickness offsets including contact types \*CONTACT\_FORMING\_..., \*CONTACT\_NODES\_TO\_SURFACE, \*CONTACT\_SURFACE\_TO\_SURFACE, and \*CONTACT\_ONE\_WAY\_SURFACE\_TO\_SURFACE.

**\*CONTROL\_ADAPTIVE**

Purpose: Activate adaptive meshing. The parts which are adaptively meshed are defined by \*PART.  
See remarks below.

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	ADPFREQ	ADPTOL	ADPOPT	MAXLVL	TBIRTH	TDEATH	LCADP	IOFLAG
Type	F	F	I	I	F	F	I	I
Default	none	$10^{20}$	1	3	0.0	$10^{20}$	0	0

**Card Format (This card is optional).**

Card 2            1            2            3            4            5            6            7            8

Variable	ADPSIZE	ADPASS	IREFLG	ADPENE	ADPTH	MEMORY	ORIENT	MAXEL
Type	F	I	I	F	F	I	I	I
Default	$10^{20}$	0	0	0.0	inactive	inactive	0	inactive

**VARIABLE****DESCRIPTION**

ADPFREQ	Time interval between adaptive refinements, see Figure 7.1.
ADPTOL	Adaptive error tolerance in degrees for ADPOPT set to 1 or 2 below. If ADPOPT is set to 8, ADPTOL is the characteristic element size.
ADPOPT	Adaptive options:  EQ.1: angle change in degrees per adaptive refinement relative to the surrounding elements for each element to be refined.

# \*CONTROL

---

VARIABLE	DESCRIPTION
	<p>EQ.2: total angle change in degrees relative to the surrounding element for each element to be refined. For example, if the <i>adptol</i>=5 degrees, the element will be refined to the second level when the total angle change reaches 5 degrees. When the angle change is 10 degrees the element will be refined to the third level.</p> <p>EQ.8: 2D r-adaptive remeshing for axisymmetric and plane strain solid elements. A completely new mesh is generated which is initialized from the old mesh using a least squares approximation. The mesh size is currently based on the value, <i>ADPTOL</i>, which gives the characteristic element size. This option is based on earlier work by Dick and Harris [1992].</p>
MAXLVL	Maximum number of refinement levels. Values of 1, 2, 3, 4,... allow a maximum of 1, 4, 16, 64, ... elements, respectively, to be created for each original element.
TBIRTH	Birth time at which the adaptive remeshing begins, see Figure 7.1.
TDEATH	Death time at which the adaptive remeshing ends, see Figure 7.1.
LCADP	Adaptive interval is changed as a function of time given by load curve ID, <i>LCADP</i> . If this option is nonzero, the <i>ADPFREQ</i> will be replaced by <i>LCADP</i> . The x-axis is time and the y-axis is the varied adaptive time interval.
IOFLAG	Flag to generate adaptive mesh at exit including <i>*NODE</i> , <i>*ELEMENT</i> , <i>*SHELL</i> , and <i>*BOUNDARY_</i> , <i>*CONTACT_NODE_</i> , and <i>*CONSTRAINED_ADAPTIVITY</i> , to be saved in the file, <i>adapt.msh</i> . EQ.1: generate adaptive mesh.
ADPSIZE	Minimum element size to be adapted based on element edge length. If undefined the edge length limit is ignored.
ADPASS	One or two pass adaptivity flag:  EQ.0: two pass adaptivity as shown in Figure 7.1a, EQ.1: one pass adaptivity as shown in Figure 7.1b.
IREFLG	Uniform refinement level. A values of 1, 2, 3, ... allow 4, 16, 64, .... elements, respectively, to be created uniformly for each original element.
ADPENE	Adapt the mesh when the contact surfaces approach or penetrate the tooling surface depending on whether the value of <i>ADPENE</i> is positive ( <i>approach</i> ) or negative ( <i>penetrates</i> ), respectively. The tooling adaptive refinement is based on the curvature of the tooling. If <i>ADPENE</i> is positive the refinement generally occurs before contact takes place; consequently, it is possible that the parameter <i>ADPASS</i> can be set to 1 in invoke the one pass adaptivity.

---

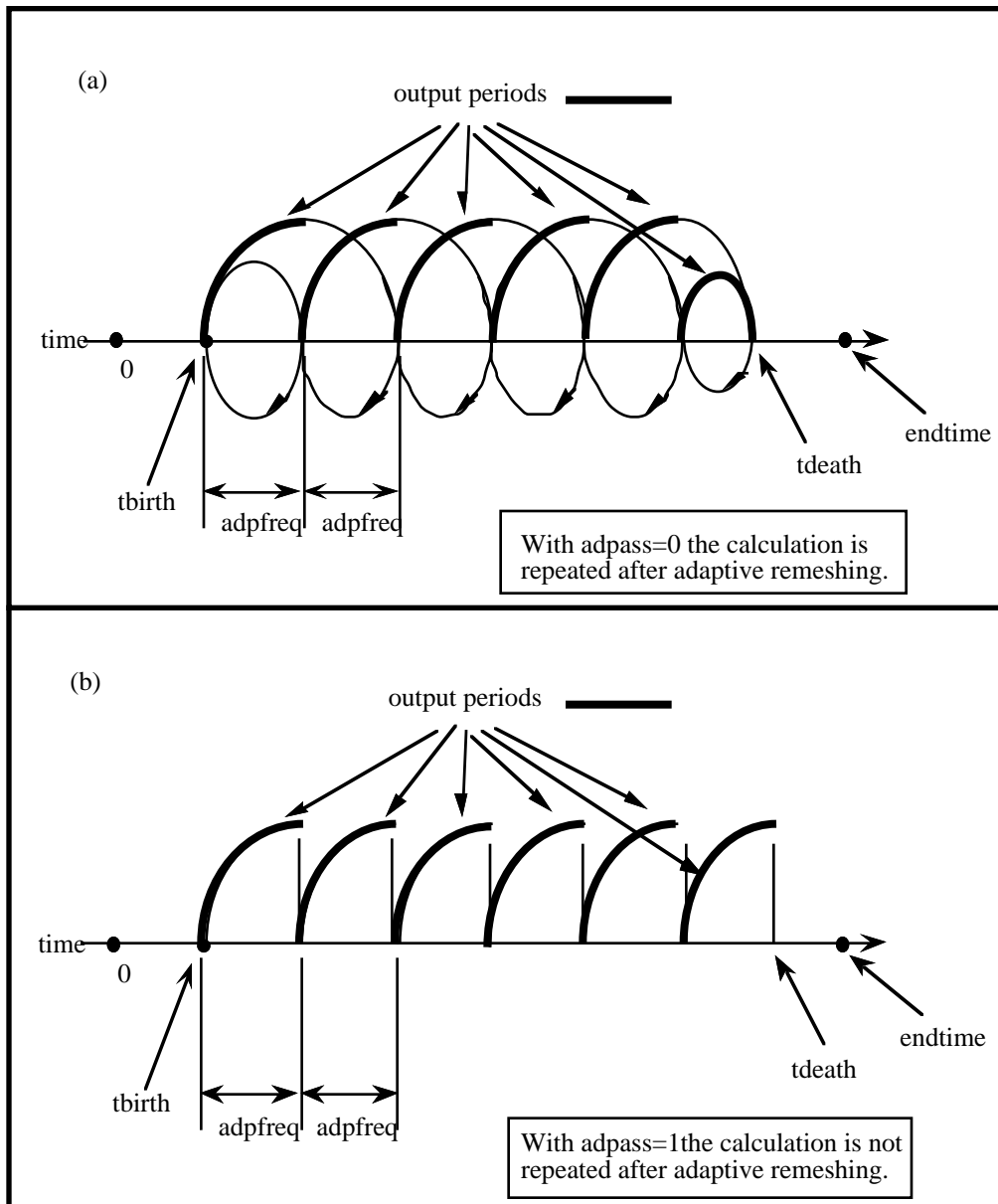
<u>VARIABLE</u>	<u>DESCRIPTION</u>
ADPTH	Absolute shell thickness level below which adaptive remeshing should began. If zero, this parameter is ignored. <b>This option works only if ADPTOL is nonzero. If thickness based adaptive remeshing is desired without angle changes, then, set ADPTOL to a large angle.</b>
MEMORY	<p>This flag can have two meanings depending on whether the memory environmental variable is or is not set. The command "<i>setenv LSTC_MEMORY auto</i>" sets the memory environmental variable which causes LS-DYNA to expand memory automatically. Note that automatic memory expansion is not always 100% reliable depending on the machine and operating system level; consequently, it is not yet the default. To see if this is set on a particular machine type the command "<i>env</i>".</p> <p>If the environmental variable <u>is not set</u> then when memory usage reaches this percentage, MEMORY, further adaptivity is prevented to avoid exceeding the memory specified at execution time. Caution is necessary since memory usage is checked after each adaptive step, and, if the memory usage increases by more than the residual percentage, 100-PERCENT, the calculation will terminate.</p> <p>If the memory environmental variable <u>is set</u> then when the number of words of memory allocated reaches or exceeds this value, MEMORY, further adaptivity is stopped.</p>
ORIENT	This option applies to the FORMING contact option only. If this flag is set to one (1), the user orientation for the contact interface is used. If this flag is set to zero (0), LS-DYNA sets the global orientation of the contact surface the first time a potential contact is observed after the birth time. If slave nodes are found on both sides of the contact surface, the orientation is set based on the principle of "majority rules". Experience has shown that this principle is not always reliable.
MAXEL	Adaptivity is stopped if this number of elements is exceeded.

**Remarks:**

1. D3DUMP and RUNRSF files contain all information necessary to restart an adaptive run. This did not work in version 936 of LS-DYNA.
2. Card 2 input is optional and is not required.
3. In order for this control card to work, the flag IADPFLG=1 must be set in the \*PART definition. Otherwise, adaptivity will not function.
4. In order for adaptivity to work optimally, the parameter SNLOG=1, must be set on Optional Control Card B in the \*CONTACT Section. On disjoint tooling meshes the contact option \*CONTACT\_FORMING\_..... is recommended.

## \*CONTROL

5. A file *adapt.rid* is left on disk after the adaptive run is completed. This file contains the root ID of all elements that are created during the calculation, and it does not need to be kept if it is not used in post-processing.



**Figure 7.1.** At time= $t_{birth}$  the adaptive calculation begins. After computing for a time interval  $adpfreq$  error norms are computed. If  $ADPASS=0$ , then the mesh that existed at time= $t_{birth}$  is refined based on the computed error norms. With the new mesh, the calculation continues to time= $t_{birth}+2 \times adpfreq$  where the error norms are again computed. The mesh that existed at time= $t_{birth}+adpfreq$  is refined and the calculation continues to time= $t_{birth}+3 \times adpfreq$ , and so on. However, if  $ADPASS=1$ , then the mesh that exist at time= $t_{birth}+adpfreq$  is refined and the calculation continues. Errors that develop between adaptive remeshing are preserved. Generally,  $ADPASS=0$  is recommended but this option is considerably more expensive.



**\*CONTROL\_ALE**

Purpose: Set default control parameters for the Arbitrary Lagrange-Eulerian and Eulerian calculations. See also \*ALE\_MULTI-MATERIAL\_GROUP, \*ALE\_SMOOTHING, \*INITIAL\_VOID\_OPTION., and \*SECTION\_SOLID\_ALE.

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	DCT	NADV	METH	AFAC	BFAC	CFAC	DFAC	EFAC
Type	I	I	I	F	F	F	F	F
Default	1	0	3	0	0	0	0	0

Card 2            1            2            3            4            5            6            7            8

Variable	START	END	AAFAC	VFACT	VLIMIT	EBC		
Type	F	F	F	F	F	I		
Default	0	1. x 10 <sup>20</sup>	1	1.0E-04	0.0	0		

---

**VARIABLE**

---

**DESCRIPTION**

DCT	Default continuum treatment: EQ.1: Lagrangian (default), EQ.2: Eulerian, EQ.3: Arbitrary Lagrangian Eulerian, EQ.4: Eulerian Ambient.
NADV	Number of cycles between advections.
METH	Advection method: EQ.1: donor cell (first order accurate), EQ.2: Van Leer + half index shift, EQ.3: Van Leer (default). EQ.4: donor cell + half index shift.

# \*CONTROL

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
AFAC	ALE smoothing weight factor - Simple average: EQ.-1: turn smoothing off.
BFAC	ALE smoothing weight factor – Volume weighting
CFAC	ALE smoothing weight factor – Isoparametric
DFAC	ALE smoothing weight factor – Equipotential
EFAC	ALE smoothing weight factor – Equilibrium
START	Start time for ALE smoothing
END	End time for ALE smoothing
AAFAC	ALE advection factor (donor cell options, default=1.0)
VFACT	Void factor. 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”. EQ.0.0: set to default 1.0e-04.
VLIMIT	Velocity limit. The time step is scaled down if the velocity exceed this limit.
EBC	Automatic Euler boundary condition EQ.0. off EQ.1. On with stick condition EQ.2. On with slip condition

This option, used for ALE and EULER formulations, defines velocity boundary conditions for the user. Velocity boundary conditions are applied to all nodes on free surfaces of an ALE or Eulerian material. For problems where the normal velocity of the material at the boundary is zero such as injection molding problems, the automatic boundary condition parameter is set to 2. This will play the same role as the Nodal Single Point Constraint. For EBC=1, the material velocity of all free surface nodes of ALE and Euler material is set to zero.

**\*CONTROL\_BULK\_VISCOSITY**

Purpose: Reset the default values of the bulk viscosity coefficients globally. This may be advisable for shock wave propagation and some materials. Bulk viscosity is used to treat shock waves. A viscous term  $q$  is added to the pressure to smear the shock discontinuities into rapidly varying but continuous transition regions. With this method the solution is unperturbed away from a shock, the Hugoniot jump conditions remain valid across the shock transition, and shocks are treated automatically.

**Card Format**

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

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

**Remarks**

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

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

$$q = 0 \quad \text{if } \dot{\epsilon}_{kk} \geq 0$$

where  $Q_1$  and  $Q_2$  are dimensionless input constants which default to 1.5 and .06, respectively, and  $l$  is a characteristic length given as the square root of the area in two dimensions and as the cube root of the volume in three,  $a$  is the local sound speed,  $Q_1$  defaults to 1.5 and  $Q_2$  defaults to .06. See Chapter 18 in Theoretical Manual for more details

# \*CONTROL

---

## \*CONTROL\_CONTACT

Purpose: Change defaults for computation with contact surfaces.

### Card Format

Card 1      1            2            3            4            5            6            7            8

Variable	SLSFAC	RWPNAL	ISLCHK	SHLTHK	PENOPT	THKCHG	ORIEN	
Type	F	F	I	I	I	I	I	
Default	.1	none	2	0	1	1	1	

Card 2      1            2            3            4            5            6            7            8

Variable	USRSTR	USRFRC	NSBCS	INTERM	XPENE	SSTHK	ECDT	TIEDPRJ
Type	I	I	I	I	F	I	I	I
Default	0	0	10-100	0	4.0	0	0	0

**Card 3 is optional. The following parameters are the default values used by parts in automatic contacts. These frictional coefficients apply only to contact types: SINGLE\_SURFACE, AUTOMATIC\_GENERAL, AUTOMATIC\_SINGLE\_SURFACE, AUTOMATIC\_NODES\_TO\_..., AUTOMATIC\_SURFACE\_..., and AUTOMATIC\_ONE\_WAY\_..., and ERODING\_SINGLE\_SURFACE. Also, see \*CONTACT, and \*PART.**

Card 3      1            2            3            4            5            6            7            8

Variable	SFRIC	DFRIC	EDC	VFC	TH	TH_SF	PEN_SF	
Type	F	F	F	F	F	F	F	
Default	0.0	0.0	0.0	0.0	0.0	0.0	0.0	

---

<b>VARIABLE</b>	<b>DESCRIPTION</b>
SLSFAC	Scale factor for sliding interface penalties, SLSFAC: EQ.0: default = .1.
RWPNAL	Scale factor for rigid wall penalties for treating rigid bodies interacting with <u>fixed</u> 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 <u>fixed</u> rigid walls. A value of 1.0 is recommended. Seven (7) variables are stored for each slave node. This can increase memory requirements significantly if all nodes are slaved to the rigid walls.
ISLCHK	Initial penetration check in contact surfaces with indication of initial penetration in output file, ISLCHK (see remarks below): EQ.0: the default is set to 1, EQ.1: no checking, EQ.2: full check of initial penetration is performed.
SHLTHK	Shell thickness considered in type surface to surface and node to surface type contact options, where options 1 and 2 below activate the new contact algorithms. The thickness offsets are always included in single surface, constraint method, and automatic surface to surface and node to surface contact types (See remarks below.): EQ.0: thickness is not considered, EQ.1: thickness is considered but rigid bodies are excluded, EQ.2: thickness is considered including rigid bodies.
PENOPT	Penalty stiffness value option. For default calculation of the penalty value please refer to the Theoretical Manual. EQ.0: the default is set to 1, EQ.1: minimum of master segment and slave node (default for most contact types), EQ.2: use master segment stiffness (old way), EQ.3: use slave node value, EQ.4: use slave node value, area or mass weighted, EQ.5: same as 4 but inversely proportional to the shell thickness. This may require special scaling and is not generally recommended. Options 4 and 5 are recommended for metalforming calculations.
THKCHG	Shell thickness changes considered in single surface contact: EQ.0: no consideration (default), EQ.1: shell thickness changes are included.

# \*CONTROL

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
ORIEN	Optional automatic reorientation of contact interface segments during initialization: EQ.0: default is set to 1. EQ.1: active for automated (part) input only. Contact surfaces are given by *PART definitions. EQ.2: active for manual (segment) and automated (part) input. EQ.3: inactive.
USRSTR	Storage per contact interface for user supplied interface control subroutine, see Appendix D. If zero, no input data is read and no interface storage is permitted in the user subroutine. This storage should be large enough to accommodate input parameters and any history data. This input data is available in the user supplied subroutine.
USRFRC	Storage per contact interface for user supplied interface friction subroutine, see Appendix E. If zero, no input data is read and no interface storage is permitted in the user subroutine. This storage should be large enough to accommodate input parameters and any history data. This input data is available in the user supplied subroutine.
NSBCS	Number of cycles between contact searching using three dimensional bucket searches. Defaults recommended.
INTERM	Flag for intermittent searching in old surface to surface contact using the interval specified as NSBCS above: EQ.0: off, EQ.1: on.
XPENE	Contact surface maximum penetration check multiplier. If the small penetration checking option, PENCHK, on the contact surface control card is active, then nodes whose penetration then exceeds the product of XPENE and the element thickness are set free, see *CONTROL_OPTION_...: EQ.0: default is set to 4.0.
SSTHK	Flag for using actual shell thickness in single surface contact logic-types 4, 13, 15 and 26. See remarks 1 and 2 below. EQ.0: Actual shell thickness is not used in the contacts.(default), EQ.1: Actual shell thickness is used in the contacts. (sometimes recommended for metal forming calculations).
ECDT	Time step size override for eroding contact: EQ.0: contact time size may control Dt. EQ.1: contact is not considered in Dt determination.
TIEDPRJ	Bypass projection of slave nodes to master surface in types: *CONTACT_TIED_NODES_TO_SURFACE, *CONTACT_TIED_SHELL_EDGE_TO_SURFACE, and, *CONTACT_TIED_SURFACE_TO_SURFACE tied interface options: EQ.0: eliminate gaps by projection nodes, EQ.1: bypass projection. Gaps create rotational constraints which can substantially affect results.

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
SFRIC	Default static coefficient of friction (see *PART_CONTACT)
DFRIC	Default dynamic coefficient of friction (see *PART_CONTACT)
EDC	Default exponential decay coefficient (see *PART_CONTACT)
VFC	Default viscous friction coefficient (see *PART_CONTACT)
TH	Default contact thickness (see *PART_CONTACT)
TH_SF	Default thickness scale factor (see *PART_CONTACT)
PEN_SF	Default local penalty scale factor (see *PART_CONTACT)

## **Remarks:**

1. The shell thickness change option must be activated in CONTROL\_SHELL control input (see ISTUPD) and a nonzero flag specified for SHLTHK above before the shell thickness changes can be included in the surface to surface contact types. An additional flag must be set, see THKCHG above, if thickness changes are included in the single surface contact algorithms. The 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 slightly more expensive.
2. In the single surface contacts types SINGLE\_SURFACE, AUTOMATIC\_SINGLE\_SURFACE, and ERODING\_SINGLE\_SURFACE, the default contact thickness is taken as the smaller value of the shell thickness or the shell edge lengths between shell nodes 1-2, 2-3, and 4-1. This may create unexpected difficulties if it is the intent to include thickness effects when the in-plane shell element dimensions are less than the thickness. The default is based on years of experience where it has been observed that sometimes rather large nonphysical thicknesses are specified to achieve high stiffness values. Since the global searching algorithm includes the effects of shell thicknesses, it is possible to slow the searches down considerably by using such nonphysical thickness dimensions.
3. The initial penetration check option is always performed in v. 950 irregardless of the value of ISLCHK. If you do not want to remove initial penetrations then set the contact birth time (see \*CONTACT\_...) so that the contact is not active at time 0.
4. Automatic reorientation requires offsets between the master and slave surface segments. The reorientation is based on segment connectivity and, once all segments are oriented consistently based on connectivity, a check is made to see if the master and slave surfaces face each other based on the right hand rule. If not, all segments in a given surface are reoriented. This procedure works well for non disjoint surfaces. If the surfaces are disjoint, the AUTOMATIC contact options, which do not require orientation, are recommended. In the FORMING contact options automatic reorientation works for disjoint surfaces.

# \*CONTROL

---

## \*CONTROL\_COUPLING

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

### Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	UNLENG	UNTIME	UNFORC	TIMIDL	FLIPX	FLIPY	FLIPZ	SUBCYL
Type	F	F	F	F	I	I	I	I
Default	1.	1.	1.	0.	0	0	0	1

<u>VARIABLE</u>	<u>DESCRIPTION</u>
UNLENG	Unit conversion factor for length. MADYMO3D/GM-CAL3D lengths are multiplied by UNLENG to obtain LS-DYNA lengths.
UNTIME	Unit conversion factor for time, UNTIME. MADYMO3D/GM-CAL3D time is multiplied by UNTIME to obtain LS-DYNA time.
UNFORC	Unit conversion factor for force, UNFORC. MADYMO3D/GM-CAL3D force is multiplied by UNFORC to obtain LS-DYNA force.
TIMIDL	Idle time during which CAL3D or MADYMO is computing and LS-DYNA3D remains inactive. Important for saving computer time.
FLIPX	Flag for flipping X-coordinate of CAL3D/MADYMO3D relative to the LS-DYNA3D model: EQ.0: off, EQ.1: on.
FLIPY	Flag for flipping Y-coordinate of CAL3D/MADYMO3D relative to the LS-DYNA3D model: EQ.0: off, EQ.1: on.
FLIPZ	Flag for flipping Z-coordinate of CAL3D/MADYMO3D relative to the LS-DYNA3D model: EQ.0: off, EQ.1: on.



<u>VARIABLE</u>	<u>DESCRIPTION</u>
SUBCYL	CAL3D/MADYMO3D subcycling interval (# of cycles): EQ.0: Set to 1, EQ.n: number of LS-DYNA time steps between each CAL3D/ MADYMO3D step. Then the position of the contacting rigid bodies is assumed to be constant for n LS-DYNA time steps. This may result in some increase in the spikes in contact, thus this option should be used carefully. As the CAL3D/MADYMO3D programs usually work with a very small number of degrees of freedom, not much gain in efficiency can be achieved.

# \*CONTROL

---

## \*CONTROL\_CPU

Purpose: Control cpu time.

### Card Format

1            2            3            4            5            6            7            8

Variable	CPUTIM							
Type	F							

---

**VARIABLE**

---

**DESCRIPTION**

---

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

### **Remarks:**

The CPU time limit applies to the current phase of the analysis or restart. The limit is not checked until after the initialization stage of the calculation. Upon reaching the cpu limit, the code will output a restart dump file and terminate. The CPU limit can also be specified on the input control line to LS-DYNA. If a value is specified on both the control line and in the input deck, the minimum value will be used.

**\*CONTROL\_DYNAMIC\_RELAXATION**

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

**Card Format**

	1	2	3	4	5	6	7	8
Variable	NRCYCK	DRTOL	DRFCTR	DRTERM	TSSFDR	IRELAL	EDTTL	IDRFLG
Type	I	F	F	F	F	I	F	I
Default	250	0.001	0.995	infinity	TSSFAC	0	0.04	0
Remarks								1, 2

**VARIABLE****DESCRIPTION**

NRCYCK	Number of iterations between convergence checks, for dynamic relaxation option (default = 250).
DRTOL	Convergence tolerance for dynamic relaxation option (default = 0.001).
DRFCTR	Dynamic relaxation factor (default = .995).
DRTERM	Optional termination time for dynamic relaxation. Termination occurs at this time or when convergence is attained (default = infinity).
TSSFDR	Scale factor for computed time step during dynamic relaxation. If zero, the value is set to SCRT defined on *CONTROL_TIMESTEP. After converging, the scale factor is reset to SCRT.
IRELAL	Automatic control for dynamic relaxation option based on algorithm of Papadrakakis [Papadrakakis 1981]: EQ.0: not active, EQ.1: active.
EDTTL	Convergence tolerance on automatic control of dynamic relaxation.

# \*CONTROL

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
IDRFLG	Dynamic relaxation flag for stress initialization: EQ.-999: dynamic relaxation not activated even if specified on a load curve, see *DEFINE_CURVE, EQ.-1: dynamic relaxation is activated and time history output is produced during dynamic relaxation, <b>see Remark 2 below,</b> EQ.0: not active, EQ.1: dynamic relaxation is activated, EQ.2: initialization to a prescribed geometry, <b>see Remark 1 below</b>

## Remarks:

1. Stress initialization in LS-DYNA for small strains may be accomplished by linking to an implicit code (option 2). A displacement state is required that gives for each nodal point its label, xyz displacements, xyz rotations and temperature. This data is read from unit 7 (m=) with the format (i8,7e15.0). See also INTRODUCTION, Execution Syntax.
2. If IDRFLG is set to -1 the dynamic relaxation proceeds as normal but time history data is written to the D3THDT file in addition to the normal data being written to the D3DRLF file. At the end of dynamic relaxation, the problem time is reset to zero. However, information is written to the D3THDT file with an increment to the time value. The time increment used is reported at the end of dynamic relaxation.

**\*CONTROL\_ENERGY**

Purpose: Provide controls for energy dissipation options.

**Card Format**

	1	2	3	4	5	6	7	8
Variable	HGEN	RWEN	SLNTEN	RYLEN				
Type	I	I	I	I				
Default	1	2	1	1				

---

**VARIABLE**

---

**DESCRIPTION**

HGEN	Hourglass energy calculation option. This option requires significant additional storage and increases cost by ten percent: EQ.1: hourglass energy is not computed (default), EQ.2: hourglass energy is computed and included in the energy balance. The hourglass energies are reported in the ASCII files GLSTAT and MATSUM, see *DATABASE_OPTION.
RWEN	Stonewall energy dissipation option: EQ.1: energy dissipation is not computed, EQ.2: energy dissipation is computed and included in the energy balance (default). The stonewall energy dissipation is reported in the ASCII file GLSTAT, see *DATABASE_OPTION.
SLNTEN	Sliding interface energy dissipation option (This parameter is always set to 2 if contact is active. The option SLNTEN=1 is not available.): EQ.1: energy dissipation is not computed, EQ.2: energy dissipation is computed and included in the energy balance. The sliding interface energy is reported in ASCII files GLSTAT and SLEOUT, see *DATABASE_OPTION.
RYLEN	Rayleigh energy dissipation option (damping energy dissipation): EQ.1: energy dissipation is not computed (default), EQ.2: energy dissipation is computed and included in the energy balance. The damping energy is reported in ASCII file GLSTAT, see *DATABASE_OPTION.

## \*CONTROL

---

### \*CONTROL\_EXPLOSIVE\_SHADOW

Purpose: Compute detonation times in explosive elements for which there is no direct line of sight.. If this control card is missing, the lighting time for an explosive element is computed using the distance from the center of the element to the nearest detonation point,  $L_d$ ; the detonation velocity,  $D$ ; and the lighting time for the detonator,  $t_d$ :

$$t_L = t_d + \frac{L_d}{D}$$

The detonation velocity for this option is taken from the element whose lighting time is computed and does not account for the possibilities that the detonation wave may travel through other explosives with different detonation velocities or that the line of sight may pass outside of the explosive material.

If this control card is present, the lighting time is based on the shortest distance through the explosive material. If inert obstacles exist within the explosive material, the lighting time will account for the extra time required for the detonation wave to travel around the obstacles. The lighting times also automatically accounts for variations in the detonation velocity if different explosives are used. No additional input is required for this control option. This option works for two and three dimensional solid elements. Also, see \*INITIAL\_DETONATION and \*MAT\_HIGH\_EXPLOSIVE.

**\*CONTROL\_HOURLASS\_{OPTION}**

One option is available:

**936**

which switches the hourglass formulation so that it is identical to that used in version 936 of LS-DYNA. The modification in the hourglass control from version 936 was to ensure that all components of the hourglass force vector are orthogonal to rigid body rotations. However, problems that run under version 936 sometimes lead to different results in versions 940 and later. This difference in results is primarily due to the modifications in the hourglass force vector. Versions released after 936 should be more accurate.

Purpose: Set the default values of the hourglass control to override the default values.

**Card Format**

	1	2	3	4	5	6	7	8
Variable	IHQ	QH						
Type	I	F						
Default	1	0.1						
Remarks	1							

---

**VARIABLE**

---

**DESCRIPTION**

IHQ

Default hourglass viscosity type:

EQ.1: standard LS-DYNA,

EQ.2: Flanagan-Belytschko integration,

EQ.3: Flanagan-Belytschko with exact volume integration,

EQ.4: stiffness form of type 2 (Flanagan-Belytschko),

EQ.5: stiffness form of type 3 (Flanagan-Belytschko),

EQ.6: Belytschko-Bindeman [1993] assumed strain co-rotational

stiffness form for 2D and 3D solid elements only. This form is available for explicit and IMPLICIT solution methods. In fact, type 6 is mandatory for the implicit options.

# \*CONTROL

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
	In the shell elements, $IHQ < 4$ is the viscous form based on Belytschko-Tsay. If $IHQ = 4, 5$ or $6$ , the stiffness form is obtained. The stiffness forms, however, can stiffen the response, especially if the deformations are large, and therefore should be used with care. For high velocities the viscous forms are recommended and for low velocities the stiffness forms are recommended. For large deformations and nonregular solids, option 3 or 5 is recommended.
QH	Default hourglass coefficient, QH. Values of QH that exceed .15 may cause instabilities. The recommended default applies to all options.

**Remark:**

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



**\*CONTROL\_IMPLICIT\_AUTO**

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

**Card Format**

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

---

**VARIABLE**

---

**DESCRIPTION**

IAUTO	Automatic time step control flag EQ.0: constant time step size (DEFAULT for “standard” implicit analysis) EQ.1: automatically adjusted time step size (DEFAULT for “springback” implicit analysis)
ITEOPT	Optimum equilibrium iteration count per time step
ITEWIN	Allowable iteration window. If iteration count is within ITEWIN iterations of ITEOPT, step size will not be adjusted.
DTMIN	Minimum allowable time step size. Simulation stops with error termination if time step falls below DTMIN.
DTMAX	Maximum allowable time step size.

**Remarks:**

IAUTO	The default for IAUTO depends on the analysis type. For “springback” analysis, automatic time step control and artificial stabilization are activated by default.
ITEOPT	The time step size is adjusted so that equilibrium is reached in ITEOPT iterations, increasing after “easy” steps, and decreasing after “difficult” but successful steps. A value of ITEOPT=21 or more can be more efficient for highly nonlinear simulations by allowing more iterations in each step, hence fewer total steps.
ITEWIN	The step size is not adjusted if the iteration count falls within ITEWIN of ITEOPT. Large values of ITEWIN make the controller more tolerant of variations in iteration count.

# \*CONTROL

---

## \*CONTROL\_IMPLICIT\_DYNAMICS

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

### Card Format

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

<u>VARIABLE</u>	<u>DESCRIPTION</u>
IMASS	Implicit analysis type EQ.0: static analysis EQ.1: dynamic analysis
GAMMA	Newmark time integration constant (see remarks below.)
BETA	Newmark time integration constant.

### Remarks:

For the dynamic problem, the linearized equilibrium equations may be written in the form

$$M\ddot{u}^{n+1} + D\dot{u}^{n+1} + K_t(x^n)\Delta u = P(x^n)^{n+1} - F(x^n)$$

where

$$M = \text{lumped mass matrix}$$

$$D = \text{damping matrix}$$

$$u^{n+1} = x^{n+1} - x^0 = \text{nodal displacement vector}$$

$$\dot{u}^{n+1} = \text{nodal point velocities at time n+1}$$

$$\ddot{u}^{n+1} = \text{nodal point accelerations at time n+1.}$$

The time integration is by the unconditionally stable, one-step, Newmark- $\beta$  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,  $\Delta t$  is the time step size, and  $\beta$  and  $\gamma$  are the free parameters of integration. For  $\gamma = \frac{1}{2}$  and  $\beta = \frac{1}{4}$  the method reduces to the trapezoidal rule and is energy conserving. If

$$\gamma > \frac{1}{2}$$

$$\beta > \frac{1}{4} \left( \frac{1}{2} + \gamma \right)^2$$

numerical damping is induced into the solution leading to a loss of energy and momentum.

# \*CONTROL

---

## \*CONTROL\_IMPLICIT\_GENERAL

Purpose: Define control parameters for implicit analysis.

### Card Format

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

(see remarks below)

---

#### VARIABLE

#### DESCRIPTION

---

IMFLAG	Implicit/Explicit switching flag EQ.0: explicit analysis EQ.1: implicit analysis EQ.2: explicit followed by one implicit step (“springback” analysis)
DT0	Initial time step size for implicit analysis
IMFORM	Element formulation switching flag EQ.1: switch to fully integrated formulation for implicit springback EQ.2: retain original element formulation (DEFAULT)
NSBS	Number of steps in nonlinear springback
IGS	Geometric (initial stress) stiffness flag EQ.1: ignore EQ.2: include

## **Remarks:**

- IMFLAG      The default value 0 indicates a standard explicit analysis will be performed. Using value 1 causes an entirely implicit analysis to be performed. Value 2 is automatically activated when the keyword \*INTERFACE\_SPRINGBACK\_SEAMLESS is present, causing the analysis type to switch when the termination time is reached. After this switch, the termination time is extended by NSBS\*DT0, or reset to twice its original value if DT0=0.0. The implicit simulation then proceeds until the new termination time is reached.
- DT0          This parameter selects the initial time step size for the implicit phase of a simulation. In a springback simulation, the default initial time step size is the termination time from the explicit forming phase of the simulation. The step size may be adjusted during a multiple step simulation if the automatic time step size control feature is active.
- IMFORM      The default Belytschko-Tsay shell element works well for forming analysis, but can perform poorly for springback analysis. This element formulation switching flag causes the more stable S/R Hughes-Liu shell element to be used for the springback phase. Adaptive mesh must be activated when using element formulation switching.
- NSBS        The default springback analysis is nonlinear, single step. When the automatic time step control and artificial stabilization are activated, a multiple step solution will be performed automatically if the single step fails. The NSBS option allows a user to skip the single step attempt, and proceed directly to a multiple step solution. *Artificial stabilization must be used for all multiple step solutions.*
- IGS          The geometric stiffness adds the effect of initial stress to the global stiffness matrix. This effect is seen in a piano string whose natural frequency changes with tension. Geometric stiffness does not always improve nonlinear convergence, so its inclusion is optional.

# \*CONTROL

---

## \*CONTROL\_IMPLICIT\_LINEAR

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

### Card Format

	1	2	3	4	5	6	7	8
Variable	LSOLVR	LPRINT	NEGEV					
Type	I	I	I					
Default	1	2	2					

(see remarks below)

<u>VARIABLE</u>	<u>DESCRIPTION</u>
LSOLVR	Linear equation solver method EQ.1: direct, sparse, incore with automatic out-of-core (DEFAULT) EQ.3: direct, sparse, double precision EQ.10: iterative, best of currently available iterative methods EQ.11: iterative, Conjugate Gradient method EQ.12: iterative, CG with Jacobi preconditioner EQ.13: iterative, CG with Incomplete Choleski preconditioner EQ.14: iterative, Lanczos method EQ.15: iterative, Lanczos with Jacobi preconditioner EQ.16: iterative, Lanczos with Incomplete Choleski preconditioner
LPRINT	Linear solver print flag EQ.1: print timing, storage information to screen and output file EQ.2: print only to output file (DEFAULT) (NOTE: during execution, sense switch "lprint" can also be used to toggle this print flag on and off.)
NEGEV	Negative eigenvalue flag. Selects procedure when negative eigenvalues are detected during stiffness matrix inversion. EQ.1: stop or retry step EQ.2: print warning message, try to continue (DEFAULT)

## **Remarks:**

- LSOLVR      The default linear equation solver is recommended. Better solvers may be added in the future.
- LPRINT      Select printing of the timing and storage information if you are comparing performance of linear equation solvers, or if you are running out of memory for large models. Minimum memory requirements for in-core and out-of-core solution are printed. This flag can also be toggled using sense switch "<ctrl-c> lprint".
- NEGEV      Negative eigenvalues result from underconstrained models (rigid body modes), severely deformed elements, or non-physical material properties. This flag allows control to be passed directly to the automatic time step controller when negative eigenvalues are detected. Otherwise, significant numerical roundoff error is likely to occur during factorization, and equilibrium iterations may fail.

# \*CONTROL

---

## \*CONTROL\_IMPLICIT\_NONLINEAR

Purpose: Define control parameters for the implicit nonlinear equation solver.

### Card 1 Format

1            2            3            4            5            6            7            8

Variable	NSOLVR	ILIMIT	MAXREF	DCTOL	ECTOL	(blank)	LSTOL	
Type	I	I	I	F	F	F	F	
Default	2	11	15	0.001	0.01		0.90	

### Optional Card 2

1            2            3            4            5            6            7            8

Variable	DNORM	DIVERG	ISTIF	NLPRT				
Type	I	I	I	I				
Default	2	1	1	2				

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

1            2            3            4            5            6            7            8

Variable	ARCCTL	ARCDIR	ARCLEN	ARCMTH	ARCDMP			
Type	I	I	F	I	I			
Default	0	none	0	1	2			

(see remarks below)



---

<b>VARIABLE</b>	<b>DESCRIPTION</b>
NSOLVR	Nonlinear solution method for implicit analysis EQ.1: Linear EQ.2: Nonlinear with BFGS updates (DEFAULT) EQ.3: Nonlinear with Broyden updates EQ.4: Nonlinear with DFP updates EQ.5: Nonlinear with Davidon updates EQ.6: Nonlinear with BFGS updates + arclength EQ.7: Nonlinear with Broyden updates + arclength EQ.8: Nonlinear with DFP updates + arclength EQ.9: Nonlinear with Davidon updates + arclength
ILIMIT	Iteration limit between automatic stiffness reformations
MAXREF	Stiffness reformation limit per time step
DCTOL	Displacement convergence tolerance
ECTOL	Energy convergence tolerance
LSTOL	Line search convergence tolerance
DNORM	Displacement norm for convergence test EQ.1: Increment vs. displacement over current step EQ.2: Increment vs. total displacement (DEFAULT)
DIVERG	Divergence flag (force imbalance increase during equilibrium iterations) EQ.1: reform stiffness if divergence detected (DEFAULT) EQ.2: ignore divergence
ISTIF	Initial stiffness formation flag EQ.1: reform stiffness at start of each step (DEFAULT) EQ.n: reform stiffness at start of every "n"th step
NLPRT	Nonlinear solver print flag EQ.1: print iteration information to screen, messag, d3hsp files EQ.2: print information only to messag, d3hsp files (DEFAULT) (NOTE: during execution, sense switch "nlprt" can also be used to toggle this print flag on and off.)

# \*CONTROL

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
	<i>The following parameters are for use with arc length methods only (<math>6 \leq \text{NSOLVR} \leq 9</math>):</i>
ARCCTL	Arc length controlling node ID EQ.0: generalized arc length method
ARCDIR	Arc length controlling node direction (ignored if ARCCTL=0 above) EQ.1: global X-translation EQ.2: global Y-translation EQ.3: global Z-translation
ARCLEN	Arc length size LE.0.0: chosen automatically using initial step size
ARCMTH	Arc length method EQ.1: Crisfield (DEFAULT) EQ.2: Ramm
ARCDMP	Arc length damping option EQ.2: off (DEFAULT) EQ.1: on, oscillations in static solution are suppressed

## **Remarks:**

NSOLVR	<p>This flag may be used to select a linear springback analysis. This disables equilibrium checking and iterations. The default nonlinear BFGS method can be used as a Full Newton method by resetting the ILIMIT parameter below.</p> <p>In the neighborhood of limit points the Newton based iteration schemes often fail. The arc length method of Riks and Wempner (combined here with the BFGS method) adds a constraint equation to limit the load step to a constant "arc Length" in load-displacement space. This latter method is frequently used to solve snap through buckling problems. <u>When applying the arc-length method, the load curves that define the loading should contain two points and start at the origin (0,0).</u> If the arc-length method is flagged and if two points characterize the load curve, LS-DYNA will extrapolate, if necessary, to determine the load. Time and load magnitude are related by a constant when the arc length method is used and it is possible that time can be negative. The arc length apply cannot be used with a dynamic analysis.</p>
ILIMIT	<p>In the default BFGS method, the global stiffness matrix is only reformed every ILIMIT iterations. Otherwise, an inexpensive stiffness update is applied. By setting ILIMIT=1, a stiffness reformation is performed every iteration. This is equivalent to the Full Newton method (with line search). A higher value of ILIMIT (20-25) can reduce the number of stiffness matrix reformations and factorizations which may lead to a significant reduction in cost.</p>
MAXREF	<p>The nonlinear equilibrium search will continue until the stiffness matrix has been reformed MAXREF times, with ILIMIT iterations between each reformation. If equilibrium has not been found, control will be passed to the automatic time step controller if it is activated. Otherwise, error termination will result. When the auto time step controller is active, it is often efficient to choose MAXREF=5 and try another stepsize quickly, rather than wasting too many iterations on a difficult step.</p>

DCTOL	When the displacement ratio (shown for each iteration) is reduced below DCTOL, this condition is satisfied. Smaller numbers lead to more strict determination of equilibrium and, on the negative side, result in more iterations and higher costs.
ECTOL	When the energy ratio shown for each iteration is reduced below ECTOL, this condition is satisfied. Smaller numbers lead to more strict determination of equilibrium and, on the negative side, result in more iterations and higher costs.
LSTOL	A line search is sometimes performed to this tolerance to guard against divergence. The search is done in the event that the system is stiffening which can often lead to a failure to converge during the equilibrium iterations.
DNORM	When computing the displacement ratio, the norm of the incremental displacement vector is divided by the norm of "total" displacement. This "total" displacement may be either the total over the current step, or the total over the entire simulation. The latter tends to be more lax, and can be poor at the end of simulations where large motions develop. For these problems, an effective combination is DNORM=1, and DCTOL=0.01 or larger.
DIVERG	By default, a new stiffness matrix is formed whenever divergence (growing out-of-balance force) is detected. This flag can be used to suppress this stiffness reformation.
ISTIF	By default, a new stiffness matrix is formed at the start of every time step. Suppressing this stiffness reformation can decrease the cost of simulations which have many tiny steps that are mostly linear.
NLPRT	This flag controls printing of displacement and energy convergence measures during the nonlinear equilibrium search. If convergence difficulty occurs, this information is helpful in determining the problem. See also the section on interactive sense switches "<ctrl-c> nlprt" and "<ctrl-c> iter".
ARCCTL	The arc length method can be controlled based on the displacement of a single node in the model. For example, in dome reversal problems the node at the center of the dome can be used. By default, the generalized arc length method is used where the norm of the global displacement vector controls the solution. This includes all nodes.
ARCLEN	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.
ARCDMP	Some static problems exhibit oscillatory response near instability points. This option numerically suppresses these oscillations, and may improve the convergence behavior of the post-buckling solution.

# \*CONTROL

---

## \*CONTROL\_IMPLICIT\_STABILIZATION

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

### Card Format

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

<u>VARIABLE</u>	<u>DESCRIPTION</u>
IAS	Artificial Stabilization flag EQ.1: active (DEFAULT for “springback” implicit analysis) EQ.2: inactive (DEFAULT for “standard” implicit analysis)
SCALE	Scale factor for artificial stabilization. Values greater than 1.0 cause less springback in the first few steps, while values less than 1.0 allow the part to springback more freely over the first few steps.
TSTART	Start time. (DEFAULT: immediately upon entering implicit mode)
TEND	End time. (DEFAULT: termination time)

### **Remarks:**

Artificial stabilization allows springback to occur over several steps. This is often necessary to obtain convergence during equilibrium iterations on problems with large springback deformation. Stabilization is introduced at the start time TSTART, and slowly removed as the end time TEND is approached. Intermediate results are not accurate representations of the fully unloaded state. The end time TEND must be reached exactly for total springback to be predicted accurately. At this time, all stabilization has been removed from the simulation.

IAS The default for IAS depends on the analysis type in \*CONTROL\_IMPLICIT\_GENERAL. For “springback” analysis, automatic time step control and artificial stabilization are activated by default.

SCALE This is a penalty scale factor similar to that used in contact interfaces. If modified, it should be changed in order-of-magnitude increments at first. Large values suppress springback until very near the termination time. Small values may not stabilize the solution enough to allow equilibrium iterations to converge.

**\*CONTROL\_OUTPUT**

Purpose: Set miscellaneous output parameters.

**Card Format**

	1	2	3	4	5	6	7	8
Variable	NPOPT	NEECHO	NREFUP	IACCOP	OPIFS	IPNINT	IKEDIT	IFLUSH
Type	I	I	I	I	F	I	I	I
Default	0	0	0	0	0.	0	100	5000

**Optional Card Format**

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

**VARIABLE****DESCRIPTION**

NPOPT

Print suppression during input phase flag for the printed output file:  
EQ.0: no suppression,  
EQ.1: nodal coordinates, element connectivities, rigid wall definitions  
and initial velocities are not printed.

NEECHO

Print suppression during input phase flag for echo file:  
EQ.0: all data printed,  
EQ.1: nodal printing is suppressed,  
EQ.2: element printing is suppressed,  
EQ.3: both node and element printing is suppressed.

# \*CONTROL

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
NREFUP	Flag to update reference node coordinates for beam elements. <b>This option requires that each reference node is unique to the beam:</b> EQ.0: no update, EQ.1: update. This is generally recommended.
IACCOP	Averaged accelerations from velocities in file “nodout” and the time history database file “d3thdt”: EQ.0: no average (default), EQ.1: averaged between output intervals.
OPIFS	Output interval for interface file ( $\Delta t$ ), see INTRODUCTION, Execution syntax.
IPNINT	Print initial time step sizes for all elements on the first cycle: EQ.0: 100 elements with the smallest time step sizes are printed. EQ.1: the governing time step sizes for each element are printed.
IKEDIT	Problem status report interval steps to the D3HSP (printed output) file. This flag is ignored if the GLSTAT file is written, see *DATABASE_GLSTAT.
IFLUSH	Number of time steps interval for flushing I/O buffers. The default value is 5000. If the I/O buffers are not emptied and an abnormal termination occurs, the output files can be incomplete. The I/O buffers for restart files are emptied automatically whenever a restart file is written so these files are not affected by this option.
IPRTF	Default print flag for RBDOUT and MATSUM files. This flag defines the default value for the print flag which can be defined in the part definition section, see *PART. This option is meant to reduce the file sizes by eliminating data which is not of interest. EQ.0: write part data into both MATSUM and RBDOUT EQ.1: write data into RBDOUT file only EQ.2: write data into MATSUM file only EQ.3: do not write data into RBDOUT and MATSUM

**\*CONTROL\_PARALLEL**

Purpose: Control parallel processing usage for shared memory computers by defining the number of processors and invoking the optional consistency of the global vector assembly.

**Card Format**

	1	2	3	4	5	6	7	8
Variable	NCPU	NUMRHS	CONST	PARA				
Type	I	I	I	I				
Default	1	0	2	0				
Remarks		1	2	3				

**VARIABLE****DESCRIPTION**

NCPU	Number of cpus used.
NUMRHS	Number of right-hand sides allocated in memory: EQ.0: same as NCPU, always recommended, EQ.1: allocate only one.
CONST	Consistency flag for parallel solution (NCPU >1). Option 2 is recommended for metal forming applications. EQ.1: on EQ.2: off, for a faster solution (default).
PARA	Flag for parallel force assembly if CONST=1. EQ.0: off EQ.1: on

**Remarks:**

1. It is recommended to always set NUMRHS=NCPU since great improvements in the parallel performance are obtained since the force assembly is then done in parallel. Setting NUMRHS to one reduces storage by one right hand side vector for each additional processor after the first. If the consistency flag is active, i.e., CONST=1, NUMRHS defaults to unity.

## \*CONTROL

---

2. For any given problem with the consistency option off, i.e.,  $CONST=2$ , slight differences in results are seen when running the same job multiple times with the same number of processors and also when varying the number of processors. Comparisons of nodal accelerations often show wide discrepancies; however, it is worth noting that the results of accelerometers often show insignificant variations due to the smoothing effect of the accelerometers which are generally attached to nodal rigid bodies. The accuracy issues are not new and are inherent in numerical simulations of automotive crash and impact problems where structural bifurcations under compressive loads are common. This problem can be easily demonstrated by using a perfectly square thin-walled tubular beam of uniform cross section under a compressive load. Typically, every run on one processor that includes a minor input change (i.e., element or hourglass formulation) will produce dramatically different results in terms of the final shape, and, likewise, if the same problem is again run on a different brand of computer. If the same problem is run on multiple processors the results can vary dramatically from run to run WITH NO INPUT CHANGE. The problem here is due to the randomness of numerical round-off which acts as a trigger in a “perfect” beam. Since summations with ( $CONST=2$ ) occur in a different order from run to run, the round-off is also random. The consistency flag,  $CONST=1$ , provides for identical results (or nearly so) whether one, two, or more processors are used while running in the shared memory parallel (SMP) mode. This is done by requiring that all contributions to global vectors be summed in a precise order independently of the number of processors used. When checking for consistent results, nodal displacements or element stresses should be compared. The NODOUT and ELOUT files should be digit to digit identical. However, the GLSTAT, SECFORC, and many of the other ASCII files will not be identical since the quantities in these files are summed in parallel for efficiency reasons and the ordering of summation operations are not enforced. The biggest drawback of this option is the CPU cost penalty which is at least 15 percent if  $PARA=0$  and is much less if  $PARA=1$  and 2 or more processors are used. Unless the  $PARA$  flag is on (for non-vector processors), parallel scaling is adversely affected. The consistency flag does not apply to MPP parallel.
3. The  $PARA$  flag will cause the force assembly for the consistency option to be performed in parallel for the shared memory parallel option. Better scaling will be obtained with the consistency option, but with more memory usage. However, the single processing speed is slightly diminished. The logic for parallelization cannot be efficiently vectorized and is not recommended for vector computers since it will degrade CPU performance. This option does not apply to MPP parallel. If  $PARA=CONST=0$  and  $NUMRHS=NCPU$  the force assembly by default is done in parallel.



**\*CONTROL\_RIGID**

Purpose: Switch the explicit rigid body joint treatment to an implicit formulation which uses Lagrange multipliers to impose prescribed kinematic boundary conditions and joint constraints. This is a new option which is underdevelopment in version 950. There is a slight cost overhead due to the assembly of sparse matrix equations which are solved using standard procedures for nonlinear problems in rigid multi-body dynamics. The generalized joint formulation can also be set on this control card.

**Card Format**

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

---

**VARIABLE**

---

**DESCRIPTION**

LMF	Lagrange multiplier flag: EQ.0: explicit penalty formulation, EQ.1: implicit formulation with Lagrange multipliers.
LMF	Generalized joint stiffness formulation (see remark): EQ.0: incremental update, EQ.1: total formulation (exact).

**Remark:**

1. As the default, the calculation of the relative angles between two coordinate systems is done incrementally. This is an approximation, in contrast to the total formulation where the angular offsets are computed exactly. The disadvantage of the latter approach is that a singularity exists when an offset angle equals 180 degrees. For most applications, the stop angles prevents this occurrence and LMF=1 should not cause a problem.

# \*CONTROL

---

## \*CONTROL\_SHELL

Purpose: Provide controls for computing shell response.

### Card Format

	1	2	3	4	5	6	7	8
Variable	WRPANG	ITRIST	IRNXX	ISTUPD	THEORY	BWC	MITER	PROJ
Type	F	I	I	I	I	I	I	I
Default	20.	2	-1	0	2	2	1	0

<u>VARIABLE</u>	<u>DESCRIPTION</u>
WRPANG	Shell element warpage angle in degrees. If a warpage greater than this angle is found, a warning message is printed. Default is 20 degrees.
ITRIST	Automatic sorting of triangular shell elements to treat degenerate quadrilateral shell elements as C0 triangular shells, see option THEORY below: EQ.1: full sorting, EQ.2: no sorting required (default).
IRNXX	Shell normal update option. This option affects the Hughes-Liu, Belytschko-Wong-Chiang, and the Belytschko-Tsay shell formulations. The latter is affected if and only if the warping stiffness option is active, i.e., BWC=1. IRNXX must be set to 2 to invoke the top or bottom surface as the reference surface for the Hughes-Liu shell elements. EQ.-2: unique nodal fibers which are incrementally updated based on the nodal rotation at the location of the fiber, EQ.-1: recompute fiber directions each cycle, EQ.0: default set to -1, EQ.1: compute on restarts, EQ.n: compute every n cycles (Hughes-Liu shells only).
ISTUPD	Shell thickness change option. This option effects all shell element formulations: EQ.0: no change. EQ.1: membrane straining causes thickness change. This option is very important in sheet metal forming or whenever membrane stretching is important.

<b>VARIABLE</b>	<b>DESCRIPTION</b>
THEORY	<p>Default shell theory:</p> <p>EQ.1: Hughes-Liu, EQ.2: Belytschko-Tsay (default), EQ.3: BCIZ triangular shell (not recommended), EQ.4: C<sub>0</sub> triangular shell, EQ.5: Belytschko-Tsay membrane, EQ.6: S/R Hughes Liu, EQ.7: S/R co-rotational Hughes Liu, EQ.8: Belytschko-Leviathan shell, EQ.9: fully integrated Belytschko-Tsay membrane, EQ.10: Belytschko-Wong-Chiang, EQ.11: Fast (co-rotational) Hughes-Liu. EQ.12: Plane stress (x-y plane), EQ.13: Plane strain (x-y plane), EQ.14: Axisymmetric solid (y-axis of symmetry) - area weighted, EQ.15: Axisymmetric solid (y-axis of symmetry) - volume weighted</p> <p>For the 2D axisymmetric solid elements, high explosive applications work best with the area weighted approach and structural applications work best with the volume weighted approach. The volume weighted approach can lead to problems along the axis of symmetry under very large deformations. Often the symmetry condition is not obeyed, and the elements will kink along the axis. The volume weighted approach must be used if 2D shell elements are used in the mesh. Type 14 and 15 elements cannot be mixed in the same calculation.</p>
BWC	<p>Warping stiffness for Belytschko-Tsay shells:</p> <p>EQ.1: Belytschko-Wong-Chiang warping stiffness added. EQ.2: Belytschko-Tsay (default).</p>
MITER	<p>Plane stress plasticity option (applies to materials 3, 18, 19, and 24):</p> <p>EQ.1: iterative plasticity with 3 secant iterations (default), EQ.2: full iterative plasticity, EQ.3: radial return noniterative plasticity. May lead to false results and has to be used with great care.</p>
PROJ	<p>Projection method for warping stiffness in the Belytschko-Tsay shell and Belytschko-Wong-Chiang elements (see remarks below).</p> <p>EQ.0: drill projection, EQ.1: full projection.</p>

**Remarks:**

1. The drill projection is used in the addition of warping stiffness to the Belytschko-Tsay and the Belytschko-Wong-Chiang shell elements. This projection generally works well and is very efficient, but to quote Belytschko and Leviathan:

"The shortcoming of the drill projection is that even elements that are invariant to rigid body rotation will strain under rigid body rotation if the drill projection is applied. On one hand, the excessive flexibility rendered by the 1-point quadrature shell element is corrected by the drill projection,

## \*CONTROL

---

but on the other hand the element becomes too stiff due to loss of the rigid body rotation invariance under the same drill projection".

They later went on to add in the conclusions:

"The projection of only the drill rotations is very efficient and hardly increases the computation time, so it is recommended for most cases. However, it should be noted that the drill projection can result in a loss of invariance to rigid body motion when the elements are highly warped. For moderately warped configurations the drill projection appears quite accurate".

In crashworthiness and impact analysis, elements that have little or no warpage in the reference configuration can become highly warped in the deformed configuration and may affect rigid body rotations if the drill projection is used. Of course it is difficult to define what is meant by "moderately warped". The full projection circumvents these problems but at a significant cost. The cost increase of the drill projection versus no projection as reported by Belytschko and Leviathan is 12 percent and by timings in LS-DYNA, 7 percent, but for the full projection they report a 110 percent increase and in LS-DYNA an increase closer to 50 percent is observed.

In Version 940.xx of LS-DYNA the drill projection was used exclusively, but in one problem the lack of invariance was observed and reported; consequently, the drill projection was replaced in the Belytschko-Leviathan shell with the full projection and the full projection is now optional for the warping stiffness in the Belytschko-Tsay and Belytschko-Wong-Chiang elements. Until this problem occurred, the drill projection seemed okay. In version 950.xx and later versions of LS-DYNA the Belytschko-Leviathan shell is somewhat slower than previously. In general in light of these problems, the drill projection cannot be recommended.

**\*CONTROL\_SOLUTION**

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

**Card Format**

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

**VARIABLE**

**DESCRIPTION**

SOLN

Analysis solution procedure:  
0: Structural analysis only,  
1: Thermal analysis only,  
2: Coupled structural thermal analysis.

# \*CONTROL

---

## \*CONTROL\_SPH

Purpose: Provide controls for computing SPH particles

### Card Format

1            2            3            4            5            6            7            8

Variable	NCBS							
Type	I							
Default	1							

---

#### VARIABLE

---

#### DESCRIPTION

NCBS

Number of cycles between particle sorting

**\*CONTROL\_STRUCTURED\_{OPTION}**

Options include:

**TERM**

Purpose: Write out a LS-DYNA structured input deck for Version 950. The name of this structured file is “dyna.str”. This input deck will not support all capabilities that are available in Version 950. As a result some data such as load curve numbers will be output in an internal numbering system. If the TERM option is activated termination will occur after the structured input file is written. This option is useful in debugging especially if problems occur in reading the input file.

# **\*CONTROL**

---

## **\*CONTROL\_SUBCYCLE**

Purpose: Control time step subcycling. This feature is described in the LS-DYNA Theoretical Manual, Section 20.2, and its use may be detrimental in cases of vectorized computation. This keyword activates subcycling. The use of mass scaling to preserve a reasonable time step size often works better than subcycling. To use mass scaling set the input parameter, DT2MS, to the negative value of the minimum acceptable time step size. See the keyword, \*CONTROL\_TIMESTEP.



**\*CONTROL\_TERMINATION**

Purpose: Stop the job.

**Card Format**

	1	2	3	4	5	6	7	8
Variable	ENDTIM	ENDCYC	DTMIN	ENDENG	ENDMAS			
Type	F	I	F	F	F			
Default	0.0	0	0.0	0.0	0.0			
Remarks	1		2					

**VARIABLE****DESCRIPTION**

ENDTIM	Termination time. Mandatory.
ENDCYC	Termination cycle. The termination cycle is optional and will be used if the specified cycle is reached before the termination time. Cycle number is identical with the time step number.
DTMIN	Reduction (or scale) factor for initial time step size to determine minimum time step, TSMIN. $TSMIN=DTSTART*DTMIN$ where DTSTART is the initial step size determined by LS-DYNA. When TSMIN is reached, LS-DYNA3D terminates with a restart dump.
ENDENG	Percent change in energy ratio for termination of calculation. If undefined, this option is inactive.
ENDMAS	Percent change in the total mass for termination of calculation. This option is relevant if and only if mass scaling is used to limit the minimum time step size, see *CONTROL_TIMESTEP variable name "DT2MS".

**Remarks:**

1. Termination by displacement may be defined in the \*TERMINATION section.
2. If the erosion flag on \*CONTROL\_TIMESTEP is set (ERODE=1), then the shell elements and solid elements with time steps falling below DTMIN will be eroded.

# \*CONTROL

---

## \*CONTROL\_THERMAL\_NONLINEAR

Purpose: Set parameters for a nonlinear thermal or coupled structural/thermal analysis. The control card, \*CONTROL\_SOLUTION, is also required.

### Card Format

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

### VARIABLE

### DESCRIPTION

REFMAX

Maximum number of matrix reformations per time step:  
EQ.0: set to 10 reformations.

TOL

Convergence tolerance for temperature:  
EQ.0.0: set to 1000 \* machine roundoff.

DCP

Divergence control parameter:  
steady state problems     $0.3 \leq \text{DCP} \leq 1.0$     default 1.0  
transient problems         $0.0 < \text{DCP} \leq 1.0$         default 0.5

**\*CONTROL\_THERMAL\_SOLVER**

Purpose: Set options for the thermal solution in a thermal only or coupled structural-thermal analysis. The control card, \*CONTROL\_SOLUTION, is also required.

**Card Format**

	1	2	3	4	5	6	7	8
Variable	ATYPE	PTYPE	SOLVER	CGTOL	GPT	EQHEAT	FWORK	
Type	I	I	I	F	I	F	F	
Default	0	0	3	1.0e-04	8	1.	1.	

---

**VARIABLE****DESCRIPTION**

---

ATYPE	Thermal analysis type: EQ.0: Steady state analysis, EQ.1: transient analysis.
PTYPE	Thermal problem type: (see *CONTROL_THERMAL_NONLINEAR if no-zero) EQ.0: linear problem, EQ.1: nonlinear problem with material properties evaluated at gauss point temperature. EQ.2: nonlinear problem with material properties evaluated at element average temperature.
SOLVER	Thermal analysis solver type: EQ.1: actol : symmetric direct solver, EQ.2: dactol : nonsymmetric direct solver, EQ.3: dscg : diagonal scaled conjugate gradient iterative (default), EQ.4: iccg : incomplete choleski conjugate gradient iterative.
CGTOL	Convergence tolerance for solver types 3 and 4. (eq.0: default 1.e-04)
GPT	Number of Gauss points to be used in the solid elements: EQ.0: the default is set to 8, EQ.1: one point quadrature is used.
EQHEAT	Mechanical equivalent of heat (e.g., 1 J / N m). (eq.0: default set to 1.)
FWORK	Fraction of mechanical work converted into heat. (eq.0: default set to 1.)

# \*CONTROL

---

## **Remark:**

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

**\*CONTROL\_THERMAL\_TIMESTEP**

Purpose: Set timestep controls for the thermal solution in a thermal only or coupled structural/thermal analysis. Also \*CONTROL\_SOLUTION, \*CONTROL\_THERMAL\_SOLVER needed.

**Card Format**

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

**VARIABLE****DESCRIPTION**

TS	Time step control: EQ.0: fixed time step, EQ.1: variable time step (may increase or decrease).
TIP	Time integration parameter: EQ.0.0: set to 0.5 - Crank-Nicholson scheme, EQ 1.0: fully implicit.
ITS	Initial thermal time step
TMIN	Minimum thermal time step: EQ.0.0: set to structural explicit timestep.
TMAX	Maximum thermal time step: EQ.0.0: set to 100 * structural explicit timestep.
DTEMP	Maximum temperature change in each time step above which the thermal timestep will be decreased: EQ.0.0: set to a temperature change of 1.0.
TSCP	Time step control parameter. The thermal time step is decreased by this factor if convergence is not obtained. $0. < TSCP < 1.0$ : EQ.0.0: set to a factor of 0.5.

# \*CONTROL

---

## \*CONTROL\_TIMESTEP

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

### Card Format

	1	2	3	4	5	6	7	8
Variable	DTINIT	TSSFAC	ISDO	TSLIMIT	DT2MS	LCTM	ERODE	MS1ST
Type	F	F	I	F	F	I	I	I
Default	-	0.9/0.67	0	0.0	0.0	0	0	0

### Card Format (This card is optional).

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

### VARIABLE

### DESCRIPTION

DTINIT

Initial time step size:  
EQ.0.0: LS-DYNA determines initial step size.

TSSFAC

Scale factor for computed time step (old name SCFT). See Remark 1 below. (Default = .90; if high explosives are used, the default is lowered to .67).

---

<b>VARIABLE</b>	<b>DESCRIPTION</b>
ISDO	<p>Basis of time size calculation for 4-node shell elements. 3-node shells use the shortest altitude for options 0,1 and the shortest side for option 2. This option has no relevance to solid elements, which use a length based on the element volume divided by the largest surface area.</p> <p>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.</p>
TSLIMIT	<p>Shell element minimum time step assignment, TSLIMIT. When a shell controls the time step, element material properties (moduli <u>not</u> masses) will be modified such that the time step does not fall below the assigned step size. Applicable only to shell elements using material models *MAT_PLASTIC_KINEMATIC, *MAT_PONER_LAW_PLASTICITY, *MAT_STRAIN_RATE_DEPENDENT_PLASTICITY, *MAT_PIECEWISE_LINEAR_PLASTICITY. The DT2MS option below applies to all materials and element classes and may be preferred.</p>
DT2MS	<p>Time step size for mass scaled solutions, DT2MS. Positive values are for quasi-static analyses or time history analyses where the inertial effects are insignificant. Default = 0.0. If negative, TSSFAC* DT2MS  is the minimum time step size permitted and mass scaling is done if and only if it is necessary to meet the Courant time step size criterion. This latter option can be used in transient analyses if the mass increases remain insignificant. See *CONTROL_TERMINATION variable name "ENDMAS".</p>
LCTM	<p>Load curve ID that limits the maximum time step size (optional). This load curve defines the maximum time step size permitted versus time. If the solution time exceeds the final time value defined by the curve the computed step size is used. If the time step size from the load curve is exactly zero, the computed time step size is also used.</p>
ERODE	<p>Erosion flag for solid and solid shell elements when DTMIN (see *CONTROL_TERMINATION) is reached. If this flag is not set the calculation will terminate:</p> <p>EQ.0: no, EQ.1: yes.</p>
MS1ST	<p>Limit mass scaling to the first step and fix the mass vector according to the time steps once. The time step will not be fixed but may drop during the calculation from the specified minimum:</p> <p>EQ.0: no, EQ.1: yes.</p>

# \*CONTROL

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
DT2MSF	Reduction (or scale) factor for initial time step size to determine the minimum time step size permitted. Mass scaling is done if it is necessary to meet the Courant time step size criterion. If this option is used DT2MS=-DT2MSF multiplied by the initial time step size, $\Delta t$ , before $\Delta t$ is scaled by TSSFAC. This option is active if and only if DT2MS=0 above.

## Remarks:

1. During the solution we loop through the elements and determine a new time step size by taking the minimum value over all elements.

$$\Delta t^{n+1} = TSSFAC \cdot \min\{\Delta t_1, \Delta t_2, \dots, \Delta t_N\}$$

where  $N$  is the number of elements. The time step size roughly corresponds to the transient time of an acoustic wave through an element using the shortest characteristic distance. For stability reasons the scale factor TSSFAC is typically set to a value of .90 (default) or some smaller value. To decrease solution time we desire to use the largest possible stable time step size. Values larger than .90 will often lead to instabilities. Some comments follow:

- The sound speed in steel and aluminum is 5mm per microsecond. Avoid excessively small elements and be aware of the effect of rotational inertia on the time step size in the Belytschko beam element. Sound speeds differ for each material, for example, consider:

AIR	331 m/s
WATER	1478
STEEL	5240
TITANIUM	5220
PLEXIGLASS	2598

- Model stiff components with rigid bodies, not by scaling Young's modulus which can substantially reduce the time step size.
- The altitude of the triangular element should be used to compute the time step size. Using the shortest side is okay only if the calculation is closely examined for possible instabilities. This is controlled by parameter ISDO.



# **\*DAMPING**

The Keyword options in this section in alphabetical order are:

**\*DAMPING\_GLOBAL**

**\*DAMPING\_PART\_MASS**

**\*DAMPING\_PART\_STIFFNESS**

## **\*DAMPING\_GLOBAL**

Purpose: Define mass weighted nodal damping that applies globally to the nodes of deformable bodies and to the mass center of the rigid bodies.

### **Card Format**

	1	2	3	4	5	6	7	8
Variable	LCID	VALDMP	STX	STY	STZ	SRX	SRY	SRZ
Type	I	F	F	F	F	F	F	F
Default	0	0.0	0.0	0.0	0.0	0.0	0.0	0.0
Remarks	1		2	2	2	2	2	2

---

**VARIABLE**

---

**DESCRIPTION**

LCID	Load curve ID which specifies node system damping: EQ.0: a contact damping factor as defined by VALDMP is used, EQ.n: system damping is given by load curve n. The damping force applied to each node is $f=-d(t)mv$ , where $d(t)$ is defined by load curve n.
VALDMP	System damping constant, d (this option is bypassed if the load curve number defined above is non zero).
STX	Scale factor on global x translational damping forces.

# \*DAMPING

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
STY	Scale factor on global y translational damping forces.
STZ	Scale factor on global z translational damping forces.
SRX	Scale factor on global x rotational damping moments.
SRY	Scale factor on global y rotational damping moments.
SRZ	Scale factor on global z rotational damping moments.

## Remarks:

1. This keyword is also used for the restart, see \*RESTART.
2. If STX=STY=STZ=SRX=SRY=SRZ=0.0 in the input above, all six values are defaulted to unity.

With mass proportional system damping the acceleration is computed as:

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

where,  $M$  is the diagonal mass matrix,  $P^n$  is the external load vector,  $F^n$  is the internal load vector, and  $F_{damp}^n$  is the force vector due to system damping. This latter vector is defined as:

$$F_{damp}^n = D_s m v$$

The best damping constant for the system is usually based on the critical damping factor for the lowest frequency mode of interest. Therefore,

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

is recommended where the natural frequency (given in radians per unit time) is generally taken as the fundamental frequency of the structure. Note that this damping applies to both translational and rotational degrees of freedom.

## \*DAMPING\_PART\_MASS

Purpose: Define mass weighted damping by part ID. Parts may be either rigid or deformable. In rigid bodies the damping forces and moments act at the center of mass.

### Card Format

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

**Card Format (This card is optional and is read if and only if FLAG=1. If this card is not read STX, STY, STZ, SRX, SRY, and SRZ default to unity.)**

Card 2	1	2	3	4	5	6	7	8
Variable	STX	STR	STZ	SRX	SRY	SRZ		
Type	F	F	F	F	F	F		
Default	0.0	0.0	0.0	0.0	0.0	0.0		

### VARIABLE

### DESCRIPTION

PID	Part ID, see *PART.
LCID	Load curve ID which specifies system damping for parts.
SF	Scale factor for load curve. This allows a simple modification of the load curve values.
FLAG	Set this flag to unity if the global components of the damping forces require separate scale factors.
STX	Scale factor on global x translational damping forces.

# \*DAMPING

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
STY	Scale factor on global y translational damping forces.
STZ	Scale factor on global z translational damping forces.
SRX	Scale factor on global x rotational damping moments.
SRY	Scale factor on global y rotational damping moments.
SRZ	Scale factor on global z rotational damping moments.

## Remarks:

Mass weighted damping damps all motions including rigid body motions. For high frequency oscillatory motion stiffness weighted damping may be preferred. With mass proportional system damping the acceleration is computed as:

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

where,  $M$  is the diagonal mass matrix,  $P^n$  is the external load vector,  $F^n$  is the internal load vector, and  $F_{damp}^n$  is the force vector due to system damping. This latter vector is defined as:

$$F_{damp}^n = D_s m v$$

The best damping constant for the system is usually based on the critical damping factor for the lowest frequency mode of interest. Therefore,

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

is recommended where the natural frequency (given in radians per unit time) is generally taken as the fundamental frequency of the structure. The damping is applied to both translational and rotational degrees of freedom. The component scale factors can be used to limit which global components see damping forces.

**\*DAMPING\_PART\_STIFFNESS**

Purpose: Assign Rayleigh stiffness damping coefficient by part ID.

**Card Format**

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

---

**VARIABLE**

---

**DESCRIPTION**

---

PID	Part ID, see *PART.
BETA	Rayleigh damping coefficient for stiffness weighted damping.

**Remarks:**

The damping matrix in Rayleigh damping is defined as:

$$C = \alpha M + \beta K$$

where C, M, and K are the damping, mass, and stiffness matrices, respectively. The constants  $\alpha$  and  $\beta$  are the mass and stiffness proportional damping constants. The mass proportional damping can be treated by system damping, see keywords: \*DAMPING\_GLOBAL and DAMPING\_PART\_MASS. Transforming C with the *i*th eigenvector  $\phi_i$  gives:

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

where  $\omega_i$  is the *i*th frequency (radians/unit time) and  $\xi_i$  is the corresponding modal damping parameter. If 10% of critical damping is sought in the *i*th 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

## **\*DAMPING**

---

frequencies and will damp rigid body motion. If large values of beta are used, it may be necessary to lower the time step size significantly. This must be done manually by reducing the time step scale factor on the \*CONTROL\_ TIMESTEP control card.

# **\*DATABASE**

The database definitions are optional, but are necessary to obtain output files containing results information. In this section the database keywords are defined in alphabetical order:

**\*DATABASE\_OPTION**

**\*DATABASE\_BINARY\_OPTION**

**\*DATABASE\_CROSS\_SECTION\_OPTION**

**\*DATABASE\_EXTENT\_OPTION**

**\*DATABASE\_FORMAT**

**\*DATABASE\_HISTORY\_OPTION**

**\*DATABASE\_NODAL\_FORCE\_GROUP**

**\*DATABASE\_SPRING\_FORWARD**

**\*DATABASE\_SUPERPLASTIC\_FORMING**

**\*DATABASE\_TRACER**

The ordering of the database definition cards in the input file is completely arbitrary.

# \*DATABASE

---

## \*DATABASE\_OPTION

Options for ASCII files include (if the file is not specified it will not be created):

<b>SECFORC</b>	Cross section forces. See *DATABASE_CROSS_SECTION_OPTION.
<b>RWFORC</b>	Wall forces.
<b>NODOUT</b>	Nodal point data. See *DATABASE_HISTORY_OPTION.
<b>ELOUT</b>	Element data. See *DATABASE_HISTORY_OPTION.
<b>GLSTAT</b>	Global data. Always obtained if SSSTAT file is activated.
<b>SSSTAT</b>	Subsystem data. See *DATABASE_EXTENT_SSSTAT.
<b>DEFORC</b>	Discrete elements.
<b>MATSUM</b>	Material energies. See Remarks 1 and 2 below.
<b>NCFORC</b>	Nodal interface forces. See *CONTACT - Card 1 (SPR and MPR)
<b>RCFORC</b>	Resultant interface forces.
<b>DEFGEO</b>	Deformed geometry file. (Note that to output this file in Chrysler format insert the following line in your <i>.cshrc</i> file: “setenv LSTC_DEFGEO chrysler”) The NASBDF file (NASTRAN Bulk Data) is created whenever the DEFGEO file is requested.
<b>SPCFORC</b>	SPC reaction forces.
<b>SWFORC</b>	Nodal constraint reaction forces (spotwelds and rivets).
<b>ABSTAT</b>	Airbag statistics.
<b>NODFOR</b>	Nodal force groups. See *DATABASE_NODAL_FORCE_GROUP.
<b>BNDOUT</b>	Boundary condition forces and energy
<b>RBDOUT</b>	Rigid body data. See Remark 2 below.
<b>GCEOUT</b>	Geometric contact entities.
<b>SLEOUT</b>	Sliding interface energy. See *CONTROL_ENERGY
<b>JNTFORC</b>	Joint force file
<b>SBTOUT</b>	Seat belt output file
<b>AVSFLT</b>	AVS database. See *DATABASE_EXTENT_OPTION.
<b>MOVIE</b>	MOVIE. See *DATABASE_EXTENT_OPTION.
<b>MPGS</b>	MPGS. See *DATABASE_EXTENT_OPTION.
<b>TRHIST</b>	Tracer particle history information. See *DATABASE_TRACER.
<b>TPRINT</b>	Thermal output from a coupled structural/thermal or thermal only analysis.



## Card Format

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

### VARIABLE

### DESCRIPTION

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

The file names and corresponding unit numbers are:

	<u>I/O UNIT #</u>	<u>FILE NAME</u>
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

# \*DATABASE

---

	<u>I/O UNIT #</u>	<u>FILE NAME</u>
Boundary conditions	i/o unit #46	BNDOUT
nodal forces and energies		
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
tracer particles	i/o unit #70	TRHIST
thermal output	i/o unit #73	TPRINT (see control card 28)

## Remarks:

1. The kinetic energy quantities in the MATSUM and GLSTAT files may differ slightly in values for several reasons. First, the rotational kinetic energy is included in the GLSTAT calculation, but is not included in MATSUM. Secondly, the energies are computed element by element in MATSUM for the deformable materials and, consequently, nodes which are merged with rigid bodies will also have their kinetic energy included in the rigid body total. Furthermore, kinetic energy is computed from nodal velocities in GLSTAT and from element midpoint velocities in MATSUM.

2. The PRINT option in the part definition allows some control over the extent of the data that is written into the MATSUM and RBDOUT files. If the print option is used the variable PRBF can be defined such that the following numbers take on the meanings:

EQ.0: default is taken from the keyword \*CONTROL\_OUTPUT,  
EQ.1: write data into RBDOUT file only

EQ.2: write data into MATSUM file only

EQ.3: do not write data into RBDOUT and MATSUM

Also see CONTROL\_OUTPUT and PART\_PRINT.

3. This keyword is also used in the restart phase, see \*RESTART. Thus, the output interval can be changed when restarting.

4. All information in the files except in AVSFLT, MOVIE, AND MPGS can also be plotted using the post-processor LS-POST. Arbitrary cross plotting of results between ASCII files is easily handled.

## \*DATABASE\_BINARY\_OPTION

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

<b>D3PLOT</b>	Dt for complete output states. See also *DATABASE_EXTENT_BINARY.
<b>D3THDT</b>	Dt for time history data of element subsets. See *DATABASE_HISTORY.
<b>D3DRLF</b>	Dynamic relaxation database.
<b>D3DUMP</b>	Binary output restart files. Define output frequency in cycles.
<b>RUNRSF</b>	Binary output restart file. Define output frequency in cycles.
<b>INTFOR</b>	Dt for output of contact interface data (file name must be given on the execution line using "S="). Also see *CONTACT variables mpr and spr.
<b>XTFILE</b>	Flag to specify output of extra time history data to XTFILE at same time as D3THDT file. The following card is left blank for this option.

The D3DUMP and the RUNRSF options create complete databases which are necessary for restarts, see \*RESTART. When RUNRSF is specified, the same file is overwritten after each interval. When D3DUMP is specified, a new restart file is created after each interval. When D3DUMP is specified, a new restart file is created after each interval, thus a “family” of files is created numbered sequentially D3DUMP01, D3DUMP02, etc. The default file names are RUNRSF and D3DUMP unless other names are specified on the execution line, see the INTRODUCTION, EXECUTION SYNTAX. Since all data held in memory is written into the restart files, these files can be quite large and care should be taken with the D3DUMP files not to create too many. If \*DATABASE\_BINARY\_D3PLOT is not specified in the keyword deck then a complete output state will be written ever timestep.

The D3PLOT, D3DRLF, and the INTFOR files contain plotting information to plot data over the three dimensional geometry of the model. These databases can be plotted with LS-POST. The D3THDT file contains time history data for element subsets as well as global information, see \*DATABASE\_HISTORY. This data can be plotted with LS-POST. The default names for the D3PLOT, D3DRLF, and the D3THDT files are D3PLOT, D3DRLF, and D3THDT. For INTFOR a unique name must be specified on the execution line with S=iff, (iff=file name), see the INTRODUCTION, EXECUTION SYNTAX. The file structure is such that each file contains the full geometry at the beginning, followed by the analysis generated output data at the specified time intervals. The default file size of 7000000 octal words may be much too small to hold one complete output state when models are very large, and an excessive number of files may be created. The limit of LS-DYNA to create files is 99 family members. Therefore, it is recommended that the file size be adjusted on the execution line with the X=scl (scl is a scale factor to enlarge the family member size).

# \*DATABASE

---

For the contents of the D3PLOT and D3THDT files see also the \*DATABASE\_EXTENT\_BINARY definition. It is possible to severely restrict the information that is dumped and consequently reduce the size of the databases. The contents of the D3THDT file are also specified with the \*DATABASE\_HISTORY definition. It should also be noted in particular that the databases can be considerably reduced for models with rigid bodies containing many elements.

## Card Format

	1	2	3	4	5	6	7	8
Variable	DT/CYCL	LCDT	BEAM	NPLTC				
Type	F	I	I	I				

<u>VARIABLE</u>	<u>DESCRIPTION</u>
DT	Time interval between outputs.
CYCL	Output interval in time steps (a time step is a cycle). For the D3DRFL file a positive number 'n' will cause plot dumps to be written at every n'th convergence check interval specified on the *CONTROL_DYNAMIC_RELAXATION card.
LCDT	Optional load curve ID specifying time interval between dumps. This option is only available for the D3PLOT, D3THDT and INTFOR files.
BEAM	Option flag for *DATABASE_BINARY_D3PLOT. EQ.0: Discrete spring and damper elements are added to the D3PLOT database where they are display as beam elements. The element global X, global Y, global Z and resultant forces are written to the database, EQ.1 No discrete spring and damper elements are added to the D3PLOT database. This option is useful when translating old LS-DYNA input decks to KEYWORD input. In older input decks there is no requirement that beam and spring elements have unique ID's, and beam elements may be created for the spring and dampers with identical ID's to existing beam elements causing a fatal error, EQ.2. Discrete spring and damper elements are added to the D3PLOT database where they are displayed as beam elements (similar to option 0). In this option the element resultant force is written to its first database position allowing beam axial forces and spring resultant forces to be plotted at the same time. This can be useful during some post-processing applications.
NPLTC	DT=ENDTIME/NPLTC applies to D3PLOT only. This overrides the DT specified in the first field.

**\*DATABASE\_CROSS\_SECTION\_OPTION**

Options include:

**PLANE  
SET**

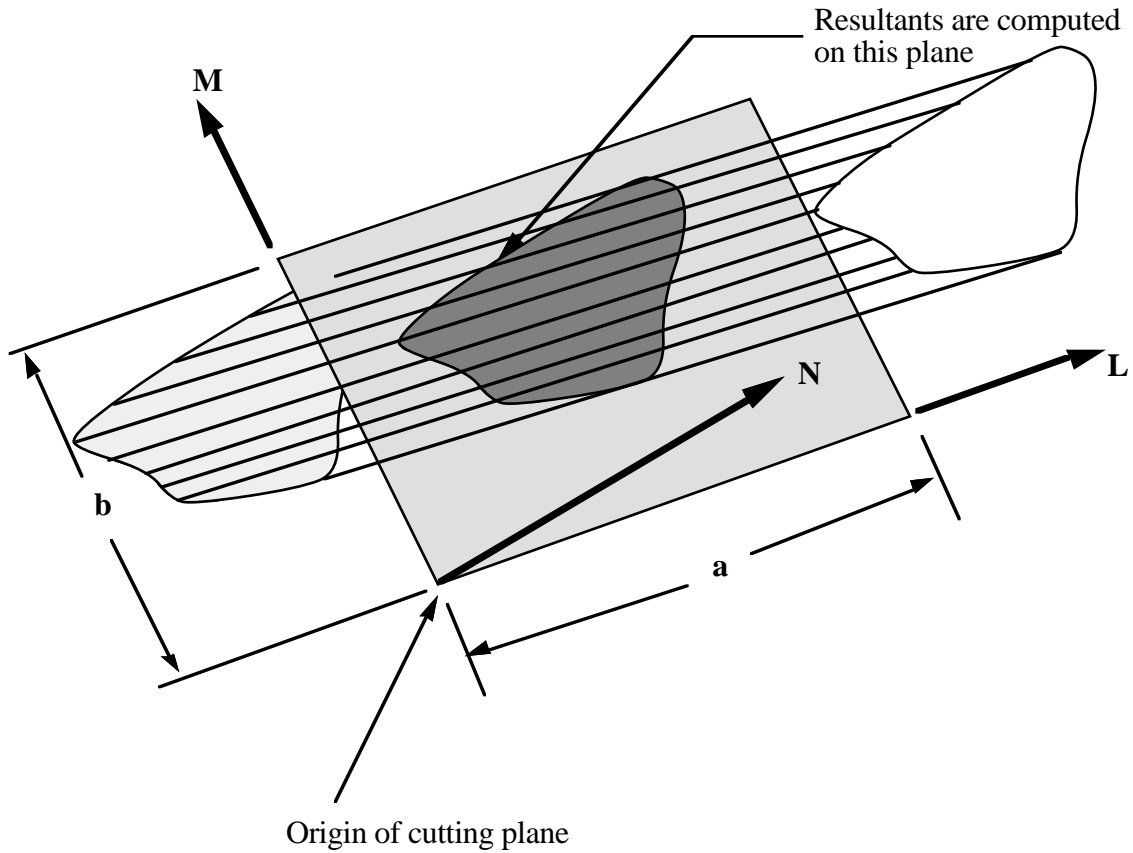
Purpose: Define a cross section for resultant forces written to ASCII file SECFORC. For the PLANE option, a set of two cards is required for each cross section. Then a cutting plane has to be defined, see Figure 9.1. If the SETS option is used, just one card is needed. In this latter case the forces in the elements belonging to the set are summed up to form the section forces.

**Format (1 of 2) for the PLANE option**

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

**Format (2 of 2) for the PLANE option**

	1	2	3	4	5	6	7	8
Variable	XHEV	YHEV	ZHEV	LENL	LENM	ID	ITYPE	
Type	F	F	F	F	F	I	I	
Default	0.	0.	0.	infinity	infinity	global	0	



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

**Format (1 of 1) for the SET option**

	1	2	3	4	5	6	7	8
Variable	NSID	HSID	BSID	SSID	TSID	DSID	ID	ITYPE
Type	I	I	I	I	I	I	I	I
Default	0	0	0	0	0	0	global	0

<u>VARIABLE</u>	<u>DESCRIPTION</u>
PSID	Part set ID. If zero all parts are included.
XCT	x-coordinate of tail of any outward drawn normal vector, <b>N</b> , originating on wall (tail) and terminating in space (head), see Figure 9.1.
YCT	y-coordinate of tail of normal vector, <b>N</b> .
ZCT	z-coordinate of tail of normal vector, <b>N</b> .
XCH	x-coordinate of head of normal vector, <b>N</b> .
YCH	y-coordinate of head of normal vector, <b>N</b> .
ZCH	z-coordinate of head of normal vector, <b>N</b> .
XHEV	x-coordinate of head of edge vector, <b>L</b> .
YHEV	y-coordinate of head of edge vector, <b>L</b> .
ZHEV	z-coordinate of head of edge vector, <b>L</b> .
LENL	Length of edge a, in <b>L</b> direction.
LENM	Length of edge b, in <b>M</b> direction.
NSID	Nodal set ID, see *SET_NODE_OPTION.
HSID	Solid element set ID, see *SET_SOLID.
BSID	Beam element set ID, see *SET_BEAM.
SSID	Shell element set ID, see *SET_SHELL_OPTION.
TSID	Thick shell element set ID, see *SET_TSHELL.
DSID	Discrete element set ID, see *SET_DISCRETE.

# \*DATABASE

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
ID	Rigid body (see *MAT_RIGID, type 20) or accelerometer ID (see *ELEMENT_SEATBELT_ACCELEROMETER). The force resultants are output in the <u>updated</u> local system of the rigid body or accelerometer.
ITYPE	Flag for local system type: EQ. 0: rigid body, EQ. 1: accelerometer.



**\*DATABASE\_EXTENT\_OPTION**

Options include:

- AVS**
- MPGS**
- MOVIE**
- BINARY**
- SSSTAT**

Purpose: Specify output database to be written. Binary applies to the data written to the D3PLOT and D3THDT files. See \*DATABASE\_BINARY\_OPTION.

**For the AVS, MPGS, and MOVIE options the following cards apply:**

Define as many cards as necessary. The created MPGS and MOVIE databases consist of a geometry file and one file for each output database.

**Card Format**

	1	2	3	4	5	6	7	8
Variable	VTYPE	COMP						
Type	I	I						

<u>VARIABLE</u>	<u>DESCRIPTION</u>
VTYPE	Variable type: EQ.0: node, EQ.1: brick, EQ.2: beam, EQ.3: shell, EQ.4: thick shell.
COMP	Component number. For the corresponding VTYPE, integer components from the following tables can be chosen: VTYPE.EQ.0: Table 9.1, VTYPE.EQ.1: Table 9.2, VTYPE.EQ.2: not supported, VTYPE.EQ.3: Table 9.3, VTYPE.EQ.4: not supported.

# \*DATABASE

---

## **Remarks:**

The AVS database consists of a title card, then a control card defining the number of nodes, brick-like elements, beam elements, shell elements, and the number of nodal vectors, NV, written for each output interval. The next NV lines consist of character strings that describe the nodal vectors. Nodal coordinates and element connectivities follow. For each state the solution time is written, followed by the data requested below. The last word in the file is the number of states. We recommend creating this file and examining its contents, since the organization is relatively transparent. The MOVIE and MPGS database are widely used and will be familiar with users who are currently using these databases.

**Table 9.1.** Nodal Quantities

Component Number	Quantity
1-3	x, y, z-displacements
4-6	x, y, z-velocities
7-9	x, y, z-accelerations
10	temperature

**Table 9.2.** Brick Element Quantities

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 9.3.** Shell and Thick Shell Element Quantities

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 (4-node shell)
23	bending moment-myy (4-node shell)
24	bending moment-mxy (4-node shell)
25	shear resultant-qxx (4-node shell)
26	shear resultant-qyy (4-node shell)
27	normal resultant-nxx (4-node shell)
28	normal resultant-nyy (4-node shell)
29	normal resultant-nxy (4-node shell)
30	thickness (4-node shell)

# \*DATABASE

---

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

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	midsurface effective stress
47	inner surface effective stress
48	outer surface effective stress
49	midsurface max. principal strain
50	through thickness strain
51	midsurface min. principal strain
52	lower surface effective strain
53	lower surface max. principal strain
54	through thickness strain
55	lower surface min. principal strain
56	lower surface effective strain
57	upper surface max. principal strain
58	through thickness strain
59	upper surface min. principal strain
60	upper surface effective strain

**Table 9.4. Beam Element Quantities**

Component Number	Quantity
1	x-force resultant
2	y-force resultant
3	z-force resultant
4	x-moment resultant
5	y-moment resultant
6	z-moment resultant

For the **BINARY** option the following cards apply:

**Card Format**

Card 1      1            2            3            4            5            6            7            8

Variable	NEIPH	NEIPS	MAXINT	STRFLG	SIGFLG	EPSFLG	RLTFLG	ENGFLG
Type	I	I	I	I	I	I	I	I
Default	0	0	3	0	1	1	1	1
Remarks			1					

Card 2      1            2            3            4            5            6            7            8

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

# \*DATABASE

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
NEIPH	Number of additional integration point history variables written to the LS-TAURUS database for solid elements. The integration point data is written in the same order that it is stored in memory-each material model has its own history variables that are stored. For user defined materials it is important to store the history data that is needed for plotting before the data which is not of interest.
NEIPS	Number of additional integration point history variables written to the LS-TAURUS database for both shell and thick shell elements for each integration point, see NEIPH above.
MAXINT	Number of shell integration points written to the LS-DYNA database, see also *INTEGRATION_SHELL. If the default value of 3 is used then results are output for the outrtmost (top) and innermost (bottom) integration points together with results for the neutral axis. If MAXINT is set to 3 and the the element has 1 integration point then all three results will be the same. If a value other than 3 is used then results for the first MAXINT integration points in the element will be output. Note: If the element has an even number of integration points and MAXINT is not set to 3 then you will not get mid-surface results. See Remarks below.
STRFLG	Set to 1 to dump strain tensors for solid, shell and thick shell elements for plotting by LS-POST and ASCII file ELOUT. For shell and thick shell elements two tensors are written, one at the innermost and one at the outermost integration point. For solid elements a single strain tensor is written.
SIGFLG	Flag for including stress tensor in the shell LS-DYNA database: EQ.1: include (default), EQ.2: exclude.
EPSFLG	Flag for including the effective plastic strains in the shell LS-DYNA database: EQ.1: include (default), EQ.2: exclude.
RLTFLG	Flag for including stress resultants in the shell LS-DYNA database: EQ.1: include (default), EQ.2: exclude.
ENGFLG	Flag for including internal energy and thickness in the LS-DYNA database: EQ.1: include (default), EQ.2: exclude.
CMPFLG	Orthotropic and anisotropic material stress output in local coordinate system for shells and thick shells. Currently, this option does not apply to solid elements with the exception of material, MAT_COMPOSITE_DAMAGE. EQ.0: global, EQ.1: local.

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
IEVERP	Every plot state for “d3plot” database is written to a separate file. This option will limit the database to 100 states: EQ.0: more than one state can be on each plotfile, EQ.1: one state only on each plotfile.
BEAMP	Number of beam integration points for output. This option does not apply to beams that use a resultant formulation.
DCOMP	Data compression to eliminate rigid body data: EQ.1: off (default), no data compression, EQ.2: on.
SHGE	Output shell hourglass energy: EQ.1: off (default), no hourglass energy written, EQ.2: on.
STSSZ	Output shell element time step: EQ.1: off (default), no shell element time step output, EQ.2: on.
N3THDT	Material energy write option for D3THDT database EQ.1: off, energy is NOT written to D3THDT database, EQ.2: on (default), energy is written to D3THDT database.

## **Remarks:**

1. If MAXINT is set to 3 then mid-surface, inner-surface and outer-surface stresses are output at the center of the element to the LS-DYNA database. For an even number of integration points, the points closest to the center are averaged to obtain the midsurface values. If multiple integration points are used in the shell plane, the stresses at the center of the element are found by computing the average of these points. For MAXINT equal to 3 LS-DYNA assumes that the data for the user defined integration rules are ordered from bottom to top even if this is not the case. If MAXINT is not equal to 3, then the stresses at the center of the element are output in the order that they are stored for the selected integration rule. If multiple points are used in plane the stresses are first averaged.
2. Beam stresses are output to the LS-DYNA database if and only if BEAMP 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,  $\sigma_{tr}$ ; the transverse shear stresses,  $\sigma_{rs}$  and  $\sigma_{tr}$ ; 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.

# \*DATABASE

---

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

Define as many cards as necessary.

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

1            2            3            4            5            6            7            8

Variable	PSID1	PSID2	PSID3	PSID4	PSID5	PSID6	PSID7	PSID8
Type	I	I	I	I	I	I	I	I

VARIABLE

DESCRIPTION

PSIDn

Part set ID for subsystem n.;, see \*SET\_PART.



**\*DATABASE\_FORMAT**

Purpose: Define the output format for binary files.

**Card Format**

	1	2	3	4	5	6	7	8
Variable	IFORM	IBINARY						
Type	I	I						
Default	0	0						
Remarks	1	2						

---

**VARIABLE**

---

**DESCRIPTION**

IFORM

Output format for D3PLOT and D3THDT files  
EQ.0: LS-DYNA database format (default),  
EQ.1: ANSYS database format,  
EQ.2: Both LS-DYNA and ANSYS database formats.

IBINARY

Word size of the binary output files (D3PLOT , D3THDT, D3DRLF and interface files for 64 bit computer such as CRAY and NEC.  
EQ.0: default 64 bit format,  
EQ.1: 32 bit IEEE format

**Remarks:**

1. This option is not available for every platform. Check LS-DYNA Banner upon execution of the program
2. By using this option one can reduce the size of the binary output files which are created by 64 bits computer such as CRAY and NEC.

# \*DATABASE

---

## \*DATABASE\_HISTORY\_OPTION

Options include:

**BEAM**  
**BEAM\_SET**  
**NODE**  
**NODE\_SET**  
**SHELL**  
**SHELL\_SET**  
**SOLID**  
**SOLID\_SET**  
**TSHELL**  
**TSHELL\_SET**

Purpose: Control which nodes or elements are output into the binary history file, D3THDT, the ASCII file NODOUT, and the ASCII file ELOUT. Define as many cards as necessary. The next “\*” card terminates the input. See also \*DATABASE\_BINARY\_OPTION and \*DATABASE\_OPTION.

### Card Format

Cards 1,2,...      1            2            3            4            5            6            7            8

Variable	ID1	ID2	ID3	ID4	ID5	ID6	ID7	ID8
Type	I	I	I	I	I	I	I	I

---

#### VARIABLE

#### DESCRIPTION

---

IDn

NODE/NODE\_SET or element/element set ID n. Elements may be BEAM/BEAM\_SET, SHELL/SHELL\_SET, SOLID/SOLID\_SET, or TSHELL/TSHELL\_SET. The contents of the files are given in Table 8.1 for nodes, Table 8.2 for solid elements, Table 8.3 for shells and thick shells, and Table 8.4 for beam elements. On the binary file D3THDT, the contents may be extended or reduced with the \*DATABASE\_EXTENT\_BINARY definition.

## \*DATABASE\_NODAL\_FORCE\_GROUP

Purpose: Define a nodal force group for output into ASCII file NODFOR and the binary file XTFIL. See also \*DATABASE\_OPTION and \*DATABASE\_BINARY\_OPTION.

### Card Format

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

### VARIABLE

### DESCRIPTION

NSID	Nodal set ID, see *SET_NODE_OPTION.
CID	Coordinate system ID for output of data in local system, see *DEFINE_COORDINATE_OPTION.

### Remarks:

1. The nodal reaction forces in the global or local (if CID is defined above) x, y, and z directions are printed into the NODFOR ascii file along with the external work which is a result of these reaction forces. The resultant force vector found by summing the reaction forces over the nodes is also written into this file. These forces can be a result of applied boundary forces such as nodal point forces and pressure boundary conditions, body forces, and contact interface forces. In the absence of body forces, interior nodes would always yield a null force resultant vector. In general this option would be used for surface nodes.

# \*DATABASE

---

## \*DATABASE\_SPRING\_FORWARD

Purpose: Create spring forward nodal force file. This option is to output resultant nodal force components of sheet metal at the end of the forming simulation into an ASCII file, "SPRING-FORWARD", for spring forward and die corrective simulations.

### Card Format

Cards 1            1            2            3            4            5            6            7            8

Variable	IFLAG								
Type	I								

---

#### VARIABLE

#### DESCRIPTION

---

IFLAG

Output type:

EQ.0: off,

EQ.1: output element nodal force vector for deformable nodes,

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

**\*DATABASE\_SUPERPLASTIC\_FORMING**

Purpose: Specify the output intervals to the superplastic forming output files. The option \*LOAD\_SUPERPLASTIC\_FORMING must be active.

**Card Format**

Cards 1            1            2            3            4            5            6            7            8

Variable	DTOUT								
Type	F								

**VARIABLE**

**DESCRIPTION**

DTOUT

Output time interval for output to “pressure”, “curve1” and “curve2” files. The “pressure” file contains general information from the analysis and the files “curve1” and “curve2” contain pressure versus time from phases 1 and 2 of the analysis. The pressure file may be plotted in Phase 3 of LS-TAURUS using the SUPERPL option.

# \*DATABASE

---

## \*DATABASE\_TRACER

Purpose: Tracer particles will save a history of either a material point or a spatial point into an ASCII file, TRHIST. This history includes positions, velocities, and stress components. The option \*DATABASE\_TRHIST must be active.

### Card Format

	1	2	3	4	5	6	7	8
Variable	TIME	TRACK	X	Y	Z			
Type	F	I	F	F	F			
Default	0.0	Lagrangian	0	0	0			

<u>VARIABLE</u>	<u>DESCRIPTION</u>
TIME	Start time for tracer particle
TRACK	Tracking option: EQ.0: particle follows material, EQ.1: particle is fixed in space.
X	Initial x-coordinate
Y	Initial y-coordinate
Z	Initial z-coordinate

# **\*DEFINE**

The keyword **\*DEFINE** provides a way of defining boxes, coordinate systems, load curves, tables, and orientation vectors for various uses. The keyword cards in this section are defined in alphabetical order:

- \*DEFINE\_BOX**
- \*DEFINE\_BOX\_ADAPTIVE**
- \*DEFINE\_BOX\_DRAWBEAD**
- \*DEFINE\_COORDINATE\_NODES**
- \*DEFINE\_COORDINATE\_SYSTEM**
- \*DEFINE\_COORDINATE\_VECTOR**
- \*DEFINE\_CURVE**
- \*DEFINE\_CURVE\_FEEDBACK**
- \*DEFINE\_CURVE\_TRIM**
- \*DEFINE\_SD\_ORIENTATION**
- \*DEFINE\_TABLE**
- \*DEFINE\_VECTOR**

Examples for the **\*DEFINE** keyword can be found at the end of this section.

# \*DEFINE

---

## \*DEFINE\_BOX

Purpose: Define a box-shaped volume. Two diagonally opposite corner points of a box are specified in global coordinates. The box volume is then used for various specifications, e.g., velocities, contact, etc.

### Card Format

	1	2	3	4	5	6	7	8
Variable	BOXID	XMN	XXM	YMN	YMX	ZMN	ZMX	
Type	I	F	F	F	F	F	F	
Default	0	0.0	0.0	0.0	0.0	0.0	0.0	
Remarks								

<u>VARIABLE</u>	<u>DESCRIPTION</u>
BOXID	Box ID. Define unique numbers.
XMN	Minimum x-coordinate.
XXM	Maximum x-coordinate.
YMN	Minimum y-coordinate.
YMX	Maximum y-coordinate.
ZMN	Minimum z-coordinate.
ZMX	Maximum z-coordinate.



**\*DEFINE\_BOX\_ADAPTIVE**

Purpose: Define a box-shaped volume enclosing the elements where the adaptive level is to be specified. If the midpoint of the element falls within the box the adaptive level is reset. Elements falling outside of this volume use the value, MAXLVL, on the \*CONTROL\_ADAPTIVE control cards.

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	BOXID	XMN	XMN	YMN	YMX	ZMN	ZMX	
Type	I	F	F	F	F	F	F	
Default	none	0.0	0.0	0.0	0.0	0.0	0.0	

Card 2            1            2            3            4            5            6            7

Variable	PID	LEVEL						
Type	I	I						
Default	0	none						

---

**VARIABLE****DESCRIPTION**

---

BOXID	Box ID. Define unique numbers.
XMN	Minimum x-coordinate.
XMN	Maximum x-coordinate.
YMN	Minimum y-coordinate.
YMX	Maximum y-coordinate.
ZMN	Minimum z-coordinate.
ZMX	Maximum z-coordinate.

# \*DEFINE

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
PID	Part ID. If zero, all active element within box are considered.
LEVEL	Maximum number of refinement levels for elements that are contained in the box. Values of 1, 2, 3, 4,... allow a maximum of 1, 4, 16, 64, ... elements, respectively, to be created for each original element.

**\*DEFINE\_BOX\_DRAWBEAD**

Purpose: Define a specific box-shaped volume around a drawbead. The box will contain the drawbead nodes and elements between the bead and the outer edge of the blank. Elements directly under the bead are also included.

**Card Format**

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

**VARIABLE****DESCRIPTION**

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

# \*DEFINE

---

## \*DEFINE\_COORDINATE\_NODES

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

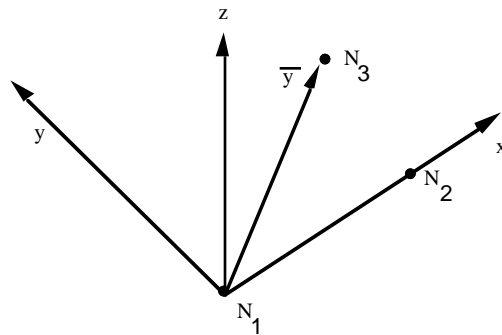
### Card Format

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

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

### Remark:

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



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

**\*DEFINE\_COORDINATE\_SYSTEM**

Purpose: Define a local coordinate system with three points. The same procedure as described in Figure 10.1, see \*DEFINE\_COORDINATE\_NODES, is used. The coordinates of the nodes are given instead.  $N_1$  is defined by  $(X_0, Y_0, Z_0)$ ,  $N_2$  is defined by  $(X_L, Y_L, Z_L)$ , and  $N_3$  by  $(X_P, Y_P, Z_P)$ .

**Card 1 of 2 - Required.**

	1	2	3	4	5	6	7	8
Variable	CID	XO	YO	ZO	XL	YL	ZL	
Type	I	F	F	F	F	F	F	
Default	0	0.0	0.0	0.0	0.0	0.0	0.0	
Remarks								

**Card 2 of 2 - Required.**

	1	2	3	4	5	6	7	8
Variable	XP	YP	ZP					
Type	F	F	F					
Default	0.0	0.0	0.0					
Remarks								

## \*DEFINE

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
CID	Coordinate system ID. A unique number has to be defined.
XO	X-coordinate of origin
YO	Y-coordinate of origin
ZO	Z-coordinate of origin
XL	X-coordinate of point on local x-axis
YL	Y-coordinate of point on local x-axis
ZL	Z-coordinate of point on local x-axis
XP	X-coordinate of point in local x-y plane
YP	Y-coordinate of point in local x-y plane
ZP	Z-coordinate of point in local x-y plane

**Remark:**

1. The coordinates of the points must be separated by a reasonable distance and not colinear to avoid numerical inaccuracies.

**\*DEFINE\_COORDINATE\_VECTOR**

Purpose: Define a local coordinate system with two vectors, see Figure 10.2. The vector cross product,  $xy \times x = z$ , determines the z-axis. The y-axis is then given by  $y = z \times x$ .

**Card Format**

	1	2	3	4	5	6	7	8
Variable	CID	XX	YX	ZX	XV	YV	ZV	
Type	I	F	F	F	F	F	F	
Default	0	0.0	0.0	0.0	0.0	0.0	0.0	

---

**VARIABLE**

---

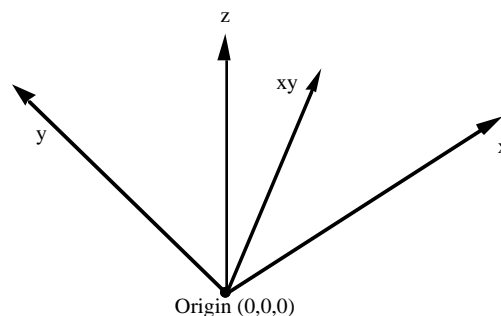
**DESCRIPTION**

---

CID	Coordinate system ID. A unique number has to be defined.
XX	X-coordinate on local x-axis. Origin lies at (0,0,0).
YX	Y-coordinate on local x-axis
ZX	Z-coordinate on local x-axis
XV	X-coordinate of local x-y vector
YV	Y-coordinate of local x-y vector
ZV	Z-coordinate of local x-y vector

**Remark:**

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



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

# \*DEFINE

---

## \*DEFINE\_CURVE

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

### Card Format

	1	2	3	4	5	6	7	8
Variable	LCID	SIDR	SFA	SFO	OFFA	OFFO	DATTYP	
Type	I	I	F	F	F	F	I	
Default	none	0	1.	1.	0.	0.	0	

**Card 2, 3, 4, etc. Put one pair of points per card (2E20.0) Input is terminated when a “\*” card is found. (Use only two points for applying loads if the implicit arc-length method is active.)**

	1	2	3	4	5	6	7	8
Variable	A1		O1					
Type	F		F					
Default	0.0		0.0					

### VARIABLE

### DESCRIPTION

LCID

Load curve ID. Tables (see \*DEFINE\_TABLE) and load curves may not share common ID's. LS-DYNA3D allows load curve ID's and table ID's to be used interchangeably. A unique number has to be defined. **Note: The magnitude of LCID is restricted to 5 significant digits. This limitation will be removed in a future release of LS-DYNA3D.**

SIDR

Stress initialization by dynamic relaxation:  
 EQ.0: load curve used in transient analysis only or for other applications,  
 EQ.1: load curve used in stress initialization but not transient analysis,  
 EQ.2: load curve applies to both initialization and transient analysis.



---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
SFA	Scale factor for abscissa value. This is useful for simple modifications.
SFO	Scale factor for ordinate value (function). This is useful for simple modifications.
OFFA	Offset for abscissa values, see explanation below.
OFFO	Offset for ordinate values (function), see explanation below.
DATTYP	Data type. Usually 0, set to 1 <u>only</u> for general xy data. This affects how offsets are applied. General xy data curves refer to curves whose abscissa values do not increase monotonically. Generally, DATTYP=0 for time dependent curves, force versus displacement curves, and stress strain curves.
A1, A2,...	Abscissa values. Only pairs have to be defined, see remarks below.
O1, O2,...	Ordinate (function) values. Only pairs have to be defined, see remarks below.

**Remarks:**

1. **Warning:** In the definition of Load Curves used in the constitutive models, reasonable spacing of the points should always be observed, i.e., never set a single point off to a value approaching infinity. LS-DYNA uses internally discretized curves to improve efficiency in the constitutive models. Also, since the constitutive models extrapolate the curves, it is important to ensure that extrapolation does not lead to physically meaningless values, such as a negative flow stress.
2. The load curve values are scaled after the offsets are applied, i.e.:

$$\text{Abscissa value} = SFA \cdot (\text{Defined value} + OFFA)$$

$$\text{Ordinate value} = SFO \cdot (\text{Defined value} + OFFO)$$

3. Positive offsets for the load curves (DATTYP=0) are intended for time versus function curves since two additional points are generated automatically at time zero and at time .999\*OFFA with the function values set to zero. If DATTYP=1, then the offsets do not create these additional points. Negative offsets for the abscissa simply shifts the abscissa values without creating additional points.
4. Load curves are not extrapolated by LS-DYNA for applied loads such as pressures, concentrated forces, displacement boundary conditions, etc. Function values are set to zero if the time, etc., goes off scale. Therefore, extreme care must be observed when defining load curves. In the constitutive models, extrapolation is employed if the values on the abscissa go off scale.
5. The load curve offsets and scale factors are ignored during restarts if the curve is redefined. See \*CHANGE\_CURVE\_DEFINITION in the restart section.

# \*DEFINE

---

## \*DEFINE\_CURVE\_FEEDBACK

Purpose: Define information that is used as the solution evolves to scale the ordinate values of the specified load curve ID. One application for this capability is in sheet metal stamping.

### Card Format

Card 1            1            2            3            4            5            6            7            8

Variable	LCID	PID	BOXID	FLDID				
Type	I	I	I	I				
Default	none	none	0	none				

Card 2

Variable	FSL	TSL	SFF	SFT	BIAS			
Type	F	F	F	F	F			
Default	none	none	1.0	1.0	0.0			

<u>VARIABLE</u>	<u>DESCRIPTION</u>
LCID	ID number for load curve to be scaled.
PID	Active part ID for load curve control
BOXID	Box ID. Elements of specified part ID contained in box are checked. If the box ID is set to zero the all elements of the active part are checked.
FLDID	Load curve ID which defines the flow limit diagram as shown in Figure 10.3. If the product of <i>FSL</i> and the ordinate value of the maximum principal strain is exceeded the scale factor for flow, <i>SF</i> , is active.
FSL	If the strain ratio, $\epsilon_{major_{workpiece}} / \epsilon_{major_{fld}}$ , is exceeded the scale factor for flow, <i>SF</i> , is active.

---

<b>VARIABLE</b>	<b>DESCRIPTION</b>
TSL	Thickness strain limit. If the through thickness strain is exceeded the scale factor for thickening, $ST$ , is active.
SFF	Scale factor for the flow limit diagram, $SF$ (Default=1.0).
SFT	Scale factor for thickening, $ST$ (Default=1.0).
BIAS	Bias for combined flow and thickening, $S$ , $-1 \leq S \leq 1$ .

**Remarks:**

The scale factor for the load curve ordinate value is updated as:

$$S_{load\ curve}^{n+1} = S_{load\ curve}^n \cdot S_{final}$$

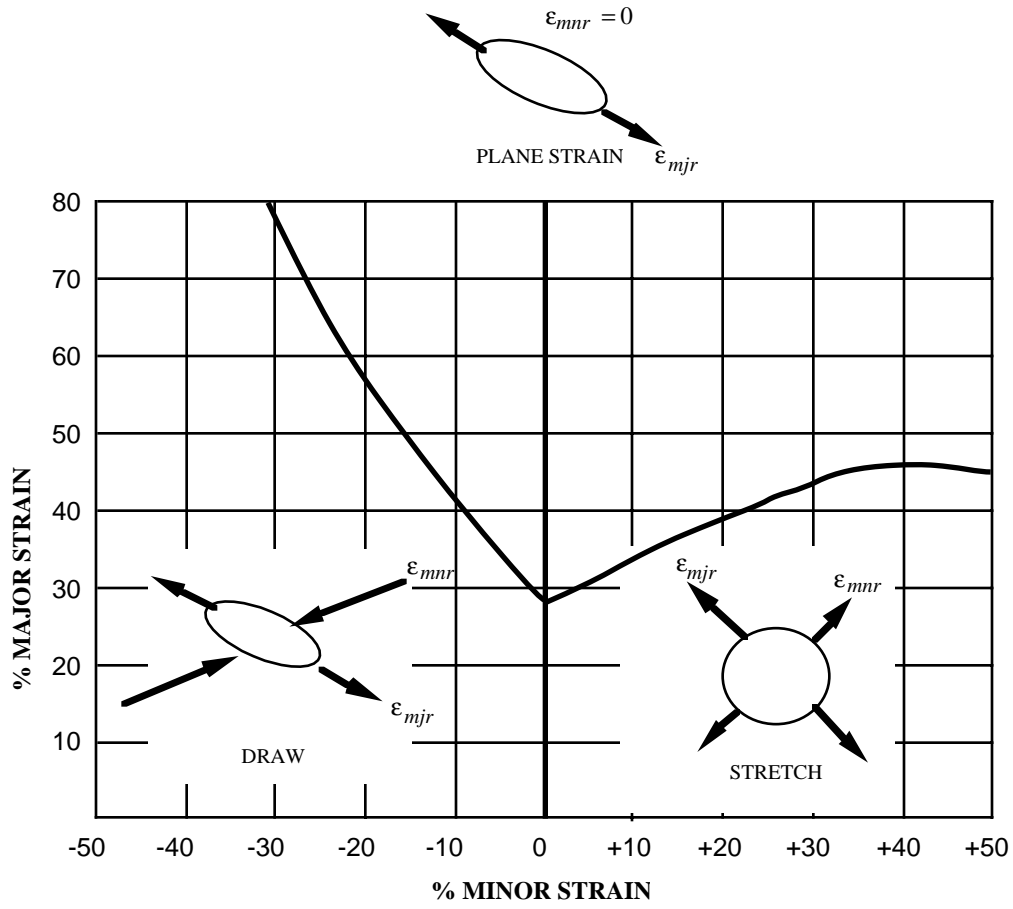
where  $S_{final}$  is equal to  $SF$  if the strain ratio is exceeded or to  $ST$  if the thickness strain limit is exceeded. The bias value determines the final scale factor,  $S_{final}$ , in the event that the thickness and flow limit diagram criteria both satisfied. In this case the scale factor for the load curve is given by:

$$S_{final} = \frac{1}{2}(1 - S) \cdot SF + \frac{1}{2}(1 + S)ST$$

Generally,  $SF$  is slightly less than unity and  $ST$  is slightly greater than unity so that  $S_{loadcurve}$  changes insignificantly from time step to time step.

# \*DEFINE

---



**Figure 10.3.** Flow limit diagram.

**\*DEFINE\_CURVE\_TRIM**

Purpose: Define a curve for trimming. Also, see \*INTERFACE\_SPRINGBACK.

**Card Format**

	1	2	3	4	5	6	7	8
Variable	TCID	TCTYPE	TFLG	TDIR	TCTOL			
Type	I	I	F	F	F			
Default	none	none	none	none	0.25			
Remarks	1,2,3			figure 10.3	4			

**Card 2, 3, 4, etc. defined if and only if TCTYPE=1. Put one pair of points per card (2E20.0) Input is terminated when a “\*” card is found.**

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

**Card 2 defined if and only if TCTYPE=2.**

	1	2	3	4	5	6	7	8
Variable	FILENAME							
Type	C							

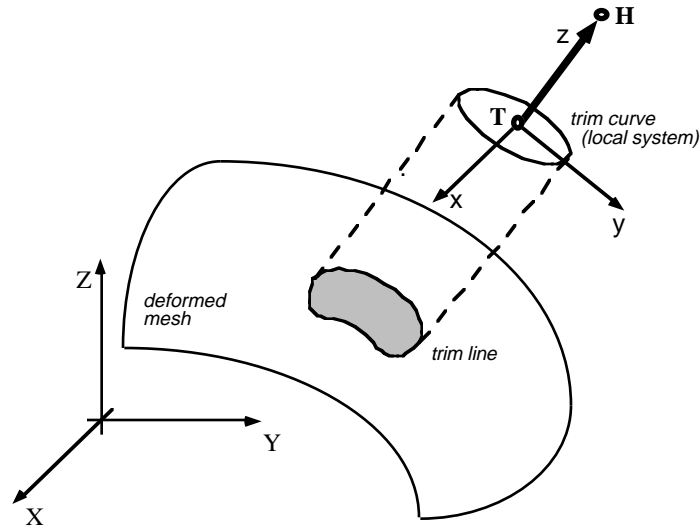
# \*DEFINE

---

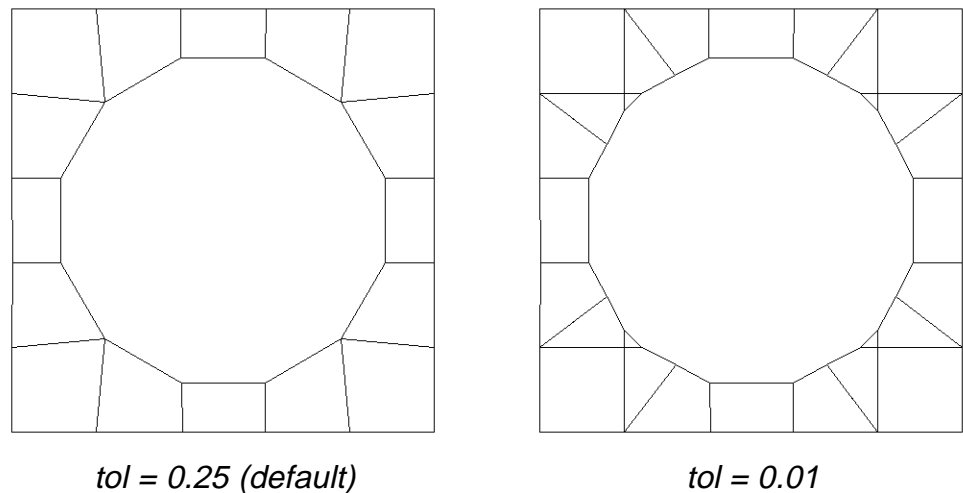
<u>VARIABLE</u>	<u>DESCRIPTION</u>
TCID	ID number for trim curve.
TCTYPE	Trim curve type: EQ.1: digitized curve provided, EQ.2: IGES trim curve.
TFLG	Element removal option: EQ.-1: remove material outside curve. EQ. 1: remove material inside curve,
TDIR	ID of vector (*DEFINE_VECTOR) giving direction of projection for trim curve (see Figure 10.4). EQ. 0: default vector (0,0,1) is used. Curve is defined in global XY plane, and projected onto mesh in global Z-direction to define trim line.
TCTOL	Tolerance limiting size of small elements created during trimming (see Figure 10.5).
CX	x-coordinate of trim curve Defined if and only if TCTYPE=1.
CY	y-coordinate of trim curve Defined if and only if TCTYPE=1.
FILENAME	Name of IGES database containing trim curve(s). Defined if and only if TCTYPE=2.

## **Remarks:**

1. This command in combination with \*ELEMENT\_TRIM trims the requested parts before the job starts.
2. If the command \*ELEMENT\_TRIM does not exist the parts are trimmed after the job is terminated.
3. Pre-trimming (\*ELEMENT\_TRIM + \*DEFINE\_CURVE\_TRIM) can handle adaptive mesh and post-trimming. The keyword \*DEFINE\_CURVE\_TRIM by itself cannot deal with an adaptive mesh. See the detailed procedure outlined in the Remarks in the Section \*INTERFACE\_SPRINGBACK.
4. The trimming tolerance TCTOL limits the size of the smallest element created during trimming. A value of 0.0 places no limit on element size. A value of 0.5 restricts new elements to be at least half of the size of the parent element. A value of 1.0 allows no new elements to be generated, only repositioning of existing nodes to lie on the trim curve.



**Figure 10.4. 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 10.5 Trimming Tolerance.** The tolerance limits the size of the small elements generated during trimming. The default tolerance (left) produces large elements. Using a tolerance of 0.01 (right) allows smaller elements, and more detail in the trim line.

# \*DEFINE

---

## \*DEFINE\_SD\_ORIENTATION

Purpose: Define orientation vectors for discrete springs and dampers. These orientation vectors are optional for this element class. Four alternative options are possible. With the first two options, IOP= 0 or 1, the vector is defined by coordinates and is fixed permanently in space. The third and fourth options orient the vector based on the motion of two nodes, so that the direction can change as the line defined by the nodes rotates.

### Card Format

	1	2	3	4	5	6	7	8
Variable	VID	IOP	XT	YT	ZT	NID1	NID2	
Type	I	I	F	F	F	I	I	
Default	0	0	0.0	0.0	0.0	0	0	
Remarks	none	1	IOP=0,1	IOP=0,1	IOP=0,1	IOP=2,3	IOP=2,3	

---

### VARIABLE

### DESCRIPTION

VID	Orientation vector ID. A unique ID number must be used.
IOP	Option: EQ.0: deflections/rotations are measured and forces/moments applied along the following orientation vector.  EQ.1: deflections/rotations are measured and forces/moments applied along the axis between the two spring/damper nodes projected onto the plane normal to the following orientation vector.  EQ.2: deflections/rotations are measured and forces/moments applied along a vector defined by the following two nodes.  EQ.3: deflections/rotations are measured and forces/moments applied along the axis between the two spring/damper nodes projected onto the plane normal to the a vector defined by the following two nodes.



<u>VARIABLE</u>	<u>DESCRIPTION</u>
XT	x-value of orientation vector. Define if IOP=0,1.
YT	y-value of orientation vector. Define if IOP=0,1.
ZT	z-value of orientation vector. Define if IOP=0,1.
NID1	Node 1 ID. Define if IOP=2,3.
NID2	Node 2 ID. Define if IOP=2, 3.

**Remarks:**

1. The orientation vectors defined by options 0 and 1 are fixed in space for the duration of the simulation. Options 2 and 3 allow the orientation vector to change with the motion of the nodes. Generally, the nodes should be members of rigid bodies, but this is not mandatory. When using nodes of deformable parts to define the orientation vector, care must be taken to ensure that these nodes will not move past each other. If this happens, the direction of the orientation vector will immediately change with the result that initiate severe instabilities can develop.

# \*DEFINE

---

## \*DEFINE\_TABLE

Purpose: Define a table. This input section is somewhat unique in that another keyword, **\*DEFINE\_CURVE**, is used as part of the input in this section. A table consists of a **\*DEFINE\_TABLE** card followed by n lines of input. Each of the n additional lines define a numerical value in ascending order corresponding to a **\*DEFINE\_CURVE** input which follows the **\*DEFINE\_TABLE** keyword and the related input. For example, to define strain rate dependency where it is desired to provide a stress versus strain curve for each strain rate, n strain rates would be defined following the **\*DEFINE\_TABLE** keyword. The curves then follow which make up the table. There are no rules for defining the n curves, i.e., each curve may have a different origin, spacing, and number of points in their definition. (Load curve ID's defined for the table may be referenced elsewhere in the input.) This rather awkward input is done for efficiency reasons related to the desire to avoid indirect addressing in the inner loops used in the constitutive model stress evaluation.

### Card Format

1            2            3            4            5            6            7            8

Variable	TBID							
Type	I							
Default	none							

**Card 2, 3, 4, etc. Put one point per card (E20.0). Input is terminated when a “\*DEFINE\_CURVE” card is found.**

1            2            3            4            5            6            7            8

Variable	VALUE							
Type	F							
Default	0.0							

Insert one \***DEFINE\_CURVE** input section here for each point defined above.

<u>VARIABLE</u>	<u>DESCRIPTION</u>
TBID	Table ID. Tables and Load curves may not share common ID's. LS-DYNA3D allows load curve ID's and table ID's to be used interchangeably.
VALUE	Load curve will be defined corresponding to this value, e.g., this value could be a strain rate, see purpose above.

**Remark:**

1. If for example, 10 stress-strain curves for 10 different strain rates are given, 10 cards with the ascending values of strain rate then follow the first card. Afterwards, 10 corresponding \***DEFINE\_CURVE** specifications have to follow.

# \*DEFINE

---

## \*DEFINE\_VECTOR

Purpose: Define a vector by defining the coordinates of two points.

### Card Format

	1	2	3	4	5	6	7	8
Variable	VID	XT	YT	ZT	XH	YH	ZH	
Type	I	F	F	F	F	F	F	
Default	0	0.0	0.0	0.0	0.0	0.0	0.0	
Remarks								

<u>VARIABLE</u>	<u>DESCRIPTION</u>
VID	Vector ID
XT	X-coordinate of tail of vector
YT	Y-coordinate of tail of vector
ZT	Z-coordinate of tail of vector
XH	X-coordinate of head of vector
YH	Y-coordinate of head of vector
ZH	Z-coordinate of head of vector

### **Remark:**

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

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

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

# \*DEFINE

---

```

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

```

```

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

```

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

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

# \*DEFINE

---

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



## **\*DEFORMABLE\_TO\_RIGID**

The cards in this section are defined in alphabetical order and are as follows:

**\*DEFORMABLE\_TO\_RIGID**

**\*DEFORMABLE\_TO\_RIGID\_AUTOMATIC**

**\*DEFORMABLE\_TO\_RIGID\_INERTIA**

If one of these cards is defined, then any deformable part defined in the model may be switched to rigid during the calculation. Parts that are defined as rigid (\*MAT\_RIGID) in the input are permanently rigid and cannot be changed to deformable.

Deformable parts may be switched to rigid at the start of the calculation by specifying them on the \*DEFORMABLE\_TO\_RIGID card.

Part switching may be specified on a restart (see RESTART section of this manual) or it may be performed automatically by use of the \*DEFORMABLE\_TO\_RIGID\_AUTOMATIC cards.

The \*DEFORMABLE\_TO\_RIGID\_INERTIA cards allow inertial properties to be defined for deformable parts that are to be swapped to rigid at a later stage.

It is not possible to perform part material switching on a restart if it was not flagged in the initial analysis. The reason for this is that extra memory needs to be set up internally to allow the switching to take place. If part switching is to take place on a restart, but no parts are to be switched at the start of the calculation, no inertia properties for switching and no automatic switching sets are to be defined, then just define one \*DEFORMABLE\_TO\_RIGID card without further input.

# **\*DEFORMABLE\_TO\_RIGID**

---

## **\*DEFORMABLE\_TO\_RIGID**

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

### **Card Format**

1            2            3            4            5            6            7            8

Variable	PID	MRB						
Type	I	I						
Default	none	0						

---

#### **VARIABLE**

#### **DESCRIPTION**

---

PID

Part ID of the part which is switched to a rigid material, also see \*PART.

MRB

Part ID of the master rigid body to which the part is merged. If zero, the part becomes either an independent or master rigid body.

# \*DEFORMABLE\_TO\_RIGID

## \*DEFORMABLE\_TO\_RIGID\_AUTOMATIC

Purpose: Define a set of parts to be switched to rigid or to deformable at some stage in the calculation.

### Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	SWSET	CODE	TIME 1	TIME 2	TIME 3	ENTNO	RELSW	PAIRED
Type	I	I	F	F	F	I	I	I
Default	none	0	0.	1.0E20	0.	0.	0	0
Remark		1				1,2		3

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

## **\*DEFORMABLE\_TO\_RIGID**

---

<b>VARIABLE</b>	<b>DESCRIPTION</b>
SWSET	Set number for this automatic switch set. Must be unique.
CODE	Activation switch code. Defines the test to activate the automatic material switch of the part: EQ.0: switch takes place at time 1, EQ.1: switch takes place between time 1 and time 2 if rigid wall force (specified below ) is zero, EQ.2: switch takes place between time 1 and time 2 if contact surface force (specified below ) is zero, EQ.3: switch takes place between time 1 and time 2 if rigid wall force (specified below ) is non-zero, EQ.4: switch takes place between time 1 and time 2 if contact surface force (specified below ) is non-zero.
TIME 1	Switch will not take place before this time.
TIME 2	Switch will not take place after this time: EQ.0 Time 2 set to 1.0e20.
TIME 3	Delay period. After this part switch has taken place, another automatic switch will not take place for the duration of the delay period. If set to zero a part switch may take place immediately after this switch.
ENTNO	Rigid wall/contact surface number for switch codes 1, 2, 3, 4.
RELSW	Related switch set. The related switch set is another automatic switch set that must be activated before this part switch can take place: EQ.0: no related switch set.
PAIRED	Define a pair of related switches. EQ. 0: not paired EQ. 1: paired with switch set RELSW and is the Master switch. EQ.-1: paired with switch set RELSW and is the Slave switch .
NRBF	Flag to delete or activate nodal rigid bodies. If nodal rigid bodies or generalized, weld definitions are active in the deformable bodies that are switched to rigid, then the definitions should be deleted to avoid instabilities: EQ.0: no change, EQ.1: delete, EQ.2: activate.
NCSF	Flag to delete or activate nodal constraint set. If nodal constraint/spotweld definitions are active in the deformable bodies that are switched to rigid, then the definitions should be deleted to avoid instabilities: EQ.0: no change, EQ.1: delete, EQ.2: activate.

# \*DEFORMABLE\_TO\_RIGID

VARIABLE	DESCRIPTION
RWF	Flag to delete or activate rigid walls: EQ.0: no change, EQ.1: delete, EQ.2: activate.
DTMAX	Maximum permitted time step size after switch.
D2R	Number of deformable parts to be switched to rigid plus number of rigid parts for which new master/slave rigid body combinations will be defined: EQ.0: no parts defined.
R2D	Number of rigid parts to be switched to deformable: EQ.0: no parts defined.

## Remarks:

1. Only surface to surface and node node to surface contacts can be used to activate an automatic part switch.
2. Contact surface and rigid wall numbers are the order in which they are defined in the deck. The first rigid wall and the first contact surface encountered in the input deck will have an entity number of 1.
3. Switch sets may be paired together to allow a pair of switches to be activated more than once. Each pair of switches should use constant values for CODE, i.e 1&3 or 2&4. Within each pair of switches the related switch ,RELSW, should be set to the ID of the other switch in the pair. The Master switch (PAIRED = 1) will be activated before the Slave switch (PAIRED = -1).
4. If the delete switch is activated, ALL corresponding constraints are deactivated regardless of their relationship to a switched part. By default, constraints which are directly associated with a switched part are deactivated/activated as necessary.

```

$ Define a pair or related switches that will be activated by (no)force on
$ Contact 3. To start with switch set 20 will be activated (PAIRED=1) swapping
$ the PARTS to RIGID. When the contact force is none zero switch set 10 will be
$ activated swapping the PARTS to DEFORMABLE. If the contact force returns to
$ zero switch set 20 will be activated again making the PARTS RIGID.
$

```

```

*DEFORMABLE_TO_RIGID_AUTOMATIC
$.>...1.>...2.>...3.>...4.>...5.>...6.>...7.>...8
$  swset      code   time 1   time 2   time 3   entno   relsw   paired
$    20         2         1         2         3         3       10      1
$  nrbf      ncsf      rwf    dtmax    D2R      R2D
$                                     1

```

```

*DEFORMABLE_TO_RIGID_AUTOMATIC
$.>...1.>...2.>...3.>...4.>...5.>...6.>...7.>...8
$  swset      code   time 1   time 2   time 3   entno   relsw   paired
$    10         2         1         2         3         3       20     -1
$  nrbf      ncsf      rwf    dtmax    D2R      R2D

```

1

# **\*DEFORMABLE\_TO\_RIGID**

---

Define D2R cards below:

## **Card Format**

1            2            3            4            5            6            7            8

Variable	PID	MRB						
Type	I	I						
Default	none	0						

---

### **VARIABLE**

### **DESCRIPTION**

PID

Part ID of the part which is switched to a rigid material.

MRB

Part ID of the master rigid body to which the part is merged. If zero, the part becomes either an independent or master rigid body.

Define R2D cards below:

## **Card Format**

1            2            3            4            5            6            7            8

Variable	PID							
Type	I							
Default	none							

---

### **VARIABLE**

### **DESCRIPTION**

PID

Part ID of the part which is switched to a deformable material.

# \*DEFORMABLE\_TO\_RIGID

## \*DEFORMABLE\_TO\_RIGID\_INERTIA

Purpose: Inertial properties can be defined for the new rigid bodies that are created when the deformable parts are switched. These can only be defined in the initial input if they are needed in a later restart. Unless these properties are defined, LS-DYNA will recompute the new rigid body properties from the finite element mesh. The latter requires an accurate mesh description. **When rigid bodies are merged to a master rigid body, the inertial properties defined for the master rigid body apply to all members of the merged set.**

### Card Format

Card 1            1            2            3            4            5            6            7            8

Variable	PID							
Type	I							
Default	none							

Card 2            1            2            3            4            5            6            7            8

Variable	XC	YC	ZC	TM				
Type	F	F	F	F				

Card 3            1            2            3            4            5            6            7            8

Variable	IXX	IXY	IXZ	IYY	IYZ	IZZ		
Type	F	F	F	F	F	F		
Default	none	0.0	0.0	none	0.0	none		

## **\*DEFORMABLE\_TO\_RIGID**

---

<b>VARIABLE</b>	<b>DESCRIPTION</b>
PID	Part ID, see *PART.
XC	x-coordinate of center of mass
YC	y-coordinate of center of mass
ZC	z-coordinate of center of mass
TM	Translational mass
IXX	$I_{xx}$ , xx component of inertia tensor
IXY	$I_{xy}$
IXZ	$I_{xz}$
IYY	$I_{yy}$
IYZ	$I_{yz}$
IZZ	$I_{zz}$



# **\*ELEMENT**

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

- \*ELEMENT\_BEAM\_{OPTION}**
- \*ELEMENT\_DISCRETE**
- \*ELEMENT\_INERTIA**
- \*ELEMENT\_MASS**
- \*ELEMENT\_SEATBELT**
- \*ELEMENT\_SEATBELT\_ACCELEROMETER**
- \*ELEMENT\_SEATBELT\_PRETENSIONER**
- \*ELEMENT\_SEATBELT\_RETRACTOR**
- \*ELEMENT\_SEATBELT\_SENSOR**
- \*ELEMENT\_SEATBELT\_SLIPRING**
- \*ELEMENT\_SHELL\_{OPTION}**
- \*ELEMENT\_SOLID\_{OPTION}**
- \*ELEMENT\_SPH**
- \*ELEMENT\_TRIM**
- \*ELEMENT\_TSHELL**

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

# \*ELEMENT

---

\*ELEMENT\_BEAM\_{OPTION}

Available options include:

**<BLANK>**  
**THICKNESS**  
**PID**

Purpose: Define two node elements including 3D beams, trusses, 2D axisymmetric shells, and 2D plane strain beam elements. The type of the element and its formulation is specified through the part ID (see \*PART) and the section ID (see \*SECTION\_BEAM).

Two alternative methods are available for defining the cross sectional property data. The THICKNESS option is provided for the user to override the \*SECTION\_BEAM data which is taken as the default if the THICKNESS option is not used. End release conditions are imposed used constraint equations, and caution must be used with this option as discussed in remark 2 below.

The PID option is used by the type 9 spot weld element only and is ignored for all other beam types. When the PID option is active an additional card is read that gives two part ID's that are tied by the spot weld element. If the PID option is inactive for the type 9 element the nodal points of the spot weld are located to the two nearest segments. The surface of each segment should project to the other and in the most typical case the node defining the weld, assuming only one node is used, should lie in the middle; however, this is not a requirement. Note that with the spotweld elements only one node is needed to define the weld, and two nodes are optional.

## Card Format (10I8)

1      2      3      4      5      6      7      8      9      10

Variable	EID	PID	N <sub>1</sub>	N <sub>2</sub>	N <sub>3</sub>	RT1	RR1	RT2	RR2	LOCAL
Type	I	I	I	I	I	I	I	I	I	I
Default	none	none	none	none	none	0	0	0	0	2
Remarks					1	2,3	2,3	2,3	2,3	2,3

# \*ELEMENT

## Optional Card (Required if THICKNESS is specified after the keyword)

1 2 3 4 5 6 7 8 9 10

Variable	PARM1	PARM2	PARM3	PARM4	PARM5
Type	F	F	F	F	F
Remarks	4	5	5	5	6

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

1 2 3 4 5 6 7 8 9 10

Variable	PID1	PID2								
Type	I	I								
Default	none	none								
Remarks										

### VARIABLE

### DESCRIPTION

EID	Element ID. A unique number has to be specified.
PID	Part ID, see *PART.
N1	Nodal point (end) 1.
N2	Nodal point (end) 2. This node is optional for the spot weld, beam type 9, since if it not defined it will be created automatically and given a nonconflicting nodal point ID. Nodes N1 and N2 are automatically positioned for the spot weld beam element.
N3	Nodal point 3. The third node, N3, is optional for beam types 3, 6, 7, 8, and 9 if the latter, type 9, has a non-circular cross section.

# \*ELEMENT

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
RT1, RT2	Release conditions for translations at nodes N1 and N2, respectively: EQ.0: no translational degrees-of-freedom are released EQ.1: x-translational degree-of-freedom EQ.2: y-translational degree-of-freedom EQ.3: z-translational degree-of-freedom EQ.4: x and y-translational degrees-of-freedom EQ.5: y and z-translational degrees-of-freedom EQ.6: z and x-translational degrees-of-freedom EQ.7: x, y, and z-translational degrees-of-freedom (3DOF) This option does not apply to the spot weld, beam type 9.
RR1, RR2	Release conditions for rotations at nodes N1 and N2, respectively: EQ.0: no rotational degrees-of-freedom are released EQ.1: x-rotational degree-of-freedom EQ.2: y-rotational degree-of-freedom EQ.3: z-rotational degree-of-freedom EQ.4: x and y-rotational degrees-of-freedom EQ.5: y and z-rotational degrees-of-freedom EQ.6: z and x-rotational degrees-of-freedom EQ.7: x, y, and z-rotational degrees-of-freedom (3DOF) This option does not apply to the spot weld, beam type 9.
LOCAL	Coordinate system option: EQ.1: global coordinate system EQ.2: local coordinate system (default)
PARM1	Based on beam type: Type.EQ.1: beam thickness, s direction at node 1 Type.EQ.2: area Type.EQ.3: area Type.EQ.4: beam thickness, s direction at node 1 Type.EQ.5: beam thickness, s direction at node 1 Type.EQ.6: volume Type.EQ.7: beam thickness, s direction at node 1 Type.EQ.8: beam thickness, s direction at node 1 Type.EQ.9: beam thickness, s direction at node 1
PARM2	Based on beam type: Type.EQ.1: beam thickness, s direction at node 2 Type.EQ.2: $I_{ss}$ Type.EQ.3: not used Type.EQ.4: beam thickness, s direction at node 2 Type.EQ.5: beam thickness, s direction at node 2 Type.EQ.6: geometric inertia Type.EQ.6: volume Type.EQ.7: beam thickness, s direction at node 2 Type.EQ.8: beam thickness, s direction at node 2 Type.EQ.9: beam thickness, s direction at node 2

---

<b>VARIABLE</b>	<b>DESCRIPTION</b>
PARM3	Based on beam type: Type.EQ.1: beam thickness, t direction at node 1 Type.EQ.2: $I_{tt}$ Type.EQ.3: not used Type.EQ.4: beam thickness, t direction at node 1 Type.EQ.5: beam thickness, t direction at node 1 Type.EQ.6: local coordinate ID Type.EQ.7: not used. Type.EQ.8: not used. Type.EQ.9: beam thickness, t direction at node 1
PARM4	Based on beam type: Type.EQ.1: beam thickness, t direction at node 2 Type.EQ.2: $I_{rr}$ Type.EQ.3: not used Type.EQ.4: beam thickness, t direction at node 2 Type.EQ.5: beam thickness, t direction at node 2 Type.EQ.6: area Type.EQ.7: not used. Type.EQ.8: not used. Type.EQ.9: beam thickness, t direction at node 2
PARM5	Based on beam type: Type.EQ.1: not used Type.EQ.2: shear area Type.EQ.3: not used Type.EQ.4: not used Type.EQ.5: not used Type.EQ.6: offset Type.EQ.7: not used. Type.EQ.8: not used. Type.EQ.9: not used
PID1	Optional part ID for spot weld element type 9.
PID2	Optional part ID for spot weld element type 9.

**Remarks:**

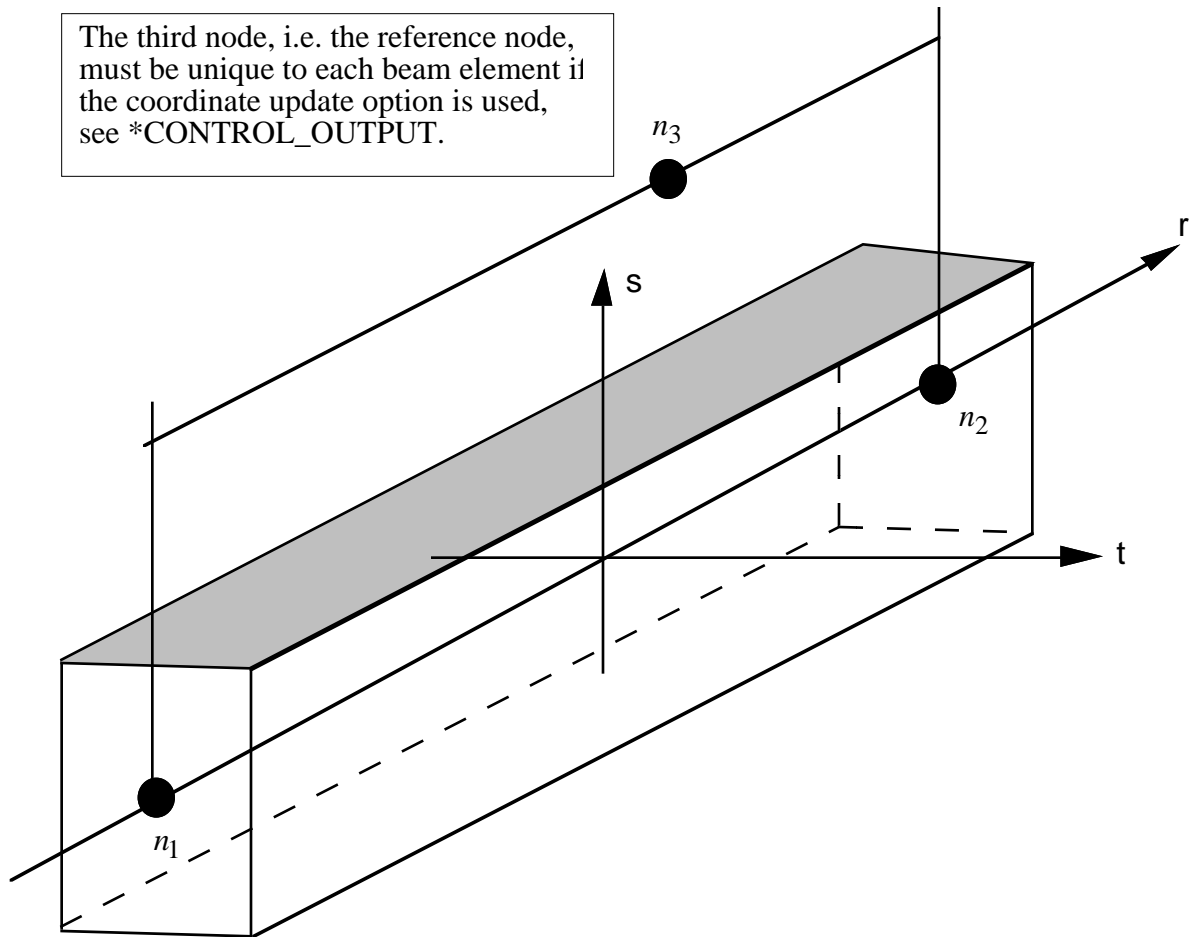
1. A plane through  $N_1$ ,  $N_2$ , and  $N_3$  defines the orientation of the principal r-s plane of the beam, see Figure 12.1.
2. This option applies to all three-dimensional beam elements. The released degrees-of-freedom can be either global, the default, or local relative to the local beam coordinate system, see Figure 12.1. A local coordinate system is stored for each node of the beam element and the orientation of the local coordinate systems rotates with the node. To properly track the response, the nodal points with a released resultant are automatically replaced with new nodes to accommodate the added degrees-of-freedom. Then constraint equations are used to join the nodal points together with the proper release conditions imposed. **Consequently, nodal points which belong to beam elements which have release conditions**

## **\*ELEMENT**

---

**applied cannot be subjected to other constraints such as applied displacement /velocity/acceleration boundary conditions, nodal rigid bodies, nodal constraint sets, or any of the constraint type contact definitions.** Force type loading conditions and penalty based contact algorithms may be used with this option.

3. Please note that this option may lead to nonphysical constraints if the translational degrees-of-freedom are released, but this should not be a problem if the displacements are infinitesimal.
4. If the second card is not defined for the resultant beam or if the area, A, is not defined the properties are taken from the cross section cards, see \*SECTION\_BEAM.
5. Do not define for discrete beams (beam type 6), see \*SECTION\_BEAM.
6. Define for resultant beam elements only, see \*SECTION\_BEAM.
7. The stress resultants are output in local coordinate system for the beam. Stress information is optional and is also output in the local system for the beam.



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

# \*ELEMENT

---

## \*ELEMENT\_DISCRETE

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

Note: These elements enter into the time step calculations. Care must be taken to ensure that the nodal masses connected by the springs and dampers are defined and unrealistically high stiffness and damping values must be avoided. **All rotations are in radians.**

### Card Format (5I8,E16.0,I8,E16.0)

1            2            3            4            5            6            7            8            9            10

Variable	EID	PID	N1	N2	VID	S	PF	OFFSET
Type	I	I	I	I	I	F	I	F
Default	none	none	none	none	0	1.	0	0
Remarks								positive for tension

### VARIABLE

### DESCRIPTION

EID	Element ID. A unique number has to be used.
PID	Part ID, see *PART.
N1	Nodal point 1.
N2	Nodal point 2. If zero, the spring/damper connects node N1 to ground.
VID	Orientation option: EQ.0: the spring/damper acts along the axis from node N1 to N2, NE.0: the spring/damper acts along the axis defined by the orientation vector, VID defined in the *DEFINE_SD_ORIENTATION section.
S	Scale factor on forces.
PF	Print flag: EQ.0: forces are printed in DEFORC file, see *DATABASE_OPTION, EQ.1: forces are not printed DEFORC file.
OFFSET	Initial offset. The initial offset is a displacement or rotation at time zero. For example, a positive offset on a translational spring will lead to a tensile force being developed at time zero.



# \*ELEMENT

## \*ELEMENT\_INERTIA

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

### Card Format (10I8)

	1	2	3	4	5	6	7	8	9	10
Variable	EID	NID	CSID							
Type	I	I	I							
Default	none	none	none							
Remarks			1							

### Card Format (8I10)

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

#### VARIABLE

#### DESCRIPTION

EID	Element ID. A unique number must be used.
NID	Node ID. Node to which the mass is assigned.
CSID	Coordinate set ID EQ.0: global inertia tensor EQ.1: principal moments of inertias with orientation vectors defined by Coordinate set CSID. See *DEFINE_COORDINATE_SYSTEM and *DEFINE_COORDINATE_VECTOR.

# **\*ELEMENT**

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
IXX	XX component of inertia tensor.
IXY	XY component of inertia tensor.
IXZ	XZ component of inertia tensor.
IYY	YY component of inertia tensor.
IYZ	YZ component of inertia tensor.
IZZ	ZZ component of inertia tensor.

## **Remarks:**

1. The coordinate system cannot be defined using the option \*DEFINE\_COORDINATE\_NODE for this element.
2. If CSID is defined then IXY, IXZ and IYZ are set to zero. The nodal inertia tensor must be positive definite, i.e., its determinant must be greater than zero, since its inverse is required. This check is done after the nodal inertia is added to the defined inertia tensor.

# \*ELEMENT

## \*ELEMENT\_MASS

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

### Card Format (2I8,E16.0)

1      2      3      4      5      6      7      8      9      10

Variable	EID	NID	MASS						
Type	I	I	F						
Default	none	none	0.						
Remarks									

### VARIABLE

### DESCRIPTION

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

# \*ELEMENT

---

## \*ELEMENT\_SEATBELT

Purpose: Define a seat belt element.

### Card Format (5I8,E16.0)

1      2      3      4      5      6      7      8      9      10

Variable	EID	PID	N <sub>1</sub>	N <sub>2</sub>	SBRID	SLEN			
Type	I	I	I	I	I	F			
Default	none	none	none	none	none	0.0			
Remarks									

---

#### VARIABLE

#### DESCRIPTION

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

### **Remarks:**

1. The retractor ID should be defined only if the element is initially **inside** a retractor, see \*ELEMENT\_SEATBELT\_RETRACTOR.
2. Belt elements are single degree of freedom elements connecting two nodes. When the strain in an element is positive (i.e. the current length is greater than the unstretched length), a tension force is calculated from the material characteristics and is applied along the current axis of the element to oppose further stretching. The unstretched length of the belt is taken as the initial distance between the two nodes defining the position of the element plus the initial slack length.

**\*ELEMENT\_SEATBELT\_ACCELEROMETER**

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

**Card Format**

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

---

**VARIABLE**

---

**DESCRIPTION**

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

**Remarks:**

The presence of the accelerometer means that the accelerations and velocities of node 1 will be output to **all** output files in local instead of global coordinates.

The local coordinate system is defined by the three nodes as follows:

- local **x** from node 1 to node 2,
- local **z** perpendicular to the plane containing nodes, 1, 2, and 3 ( $\mathbf{z} = \mathbf{x} \times \mathbf{a}$ ), where **a** is from node 1 to node 3),
- local  $\mathbf{y} = \mathbf{z} \times \mathbf{x}$ .

The three nodes should all be part of the same rigid body. The local axis then rotates with the body.

# \*ELEMENT

---

## \*ELEMENT\_SEATBELT\_PRETENSIONER

Purpose: Define seat belt pretensioner. A combination with sensors and retractors is also possible.

### Card Format

1            2            3            4            5            6            7            8

Variable	SBPRID	SBPRTY	SBSID1	SBSID2	SBSID3	SBSID4		
Type	I	I	I	I	I	I		
Default	0	0	0	0	0	0		
Remarks			1					

### Second Card

1            2            3            4            5            6            7            8

Variable	SBRID	TIME	PTLCID					
Type	I	F	I					
Default	0	0.0	0					
Remarks								

---

#### VARIABLE

---

#### DESCRIPTION

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

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
SBSID1	Sensor 1, see *ELEMENT_SEATBELT_SENSOR.
SBSID2	Sensor 2, see *ELEMENT_SEATBELT_SENSOR.
SBSID3	Sensor 3, see *ELEMENT_SEATBELT_SENSOR.
SBSID4	Sensor 4, see *ELEMENT_SEATBELT_SENSOR.
SBRID	Retractor number (SBPRTY = 1) or spring element number (SBPRTY = 2 or 3).
TIME	Time between sensor triggering and pretensioner acting.
PTLCID	Load curve for pretensioner (Time after activation, Pull-in) (SBPRTY = 1).

**Remarks:**

1. At least one sensor should be defined.

Pretensioners allow modeling of three types of active devices which tighten the belt during the initial stages of a crash. The first type represents a pyrotechnic device which spins the spool of a retractor, causing the belt to be reeled in. The user defines a pull-in versus time curve which applies once the pretensioner activates. The remaining types represent preloaded springs or torsion bars which move the buckle when released. The pretensioner is associated with any type of spring element including rotational. Note that the preloaded spring, locking spring and any restraints on the motion of the associated nodes are defined in the normal way; the action of the pretensioner is merely to cancel the force in one spring until (or after) it fires. With the second type, the force in the spring element is canceled out until the pretensioner is activated. In this case the spring in question is normally a stiff, linear spring which acts as a locking mechanism, preventing motion of the seat belt buckle relative to the vehicle. A preloaded spring is defined in parallel with the locking spring. This type avoids the problem of the buckle being free to 'drift' before the pretensioner is activated.

To activate the pretensioner, the following sequence of events must occur:

1. Any one of up to four sensors must be triggered.
2. Then a user-defined time delay occurs.
3. Then the pretensioner acts.

# \*ELEMENT

---

## \*ELEMENT\_SEATBELT\_RETRACTOR

Purpose: Define seat belt retractor.

### Card Format

	1	2	3	4	5	6	7	8
Variable	SBRID	SBRNID	SBID	SID1	SID2	SID3	SID4	
Type	I	I	I	I	I	I	I	
Default	0	0	0	0	0	0	0	
Remarks		1		2				

### Second Card

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

#### VARIABLE

#### DESCRIPTION

SBRID	Retractor ID. A unique number has to be used.
SBRNID	Retractor node ID
SBID	Seat belt element ID
SID1	Sensor ID 1



---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
SID2	Sensor ID 2
SID3	Sensor ID 3
SID4	Sensor ID 4
TDEL	Time delay after sensor triggers.
PULL	Amount of pull-out between time delay ending and retractor locking, a length value.
LLCID	Load curve for loading (Pull-out, Force), see Figure 12.3.
ULCID	Load curve for unloading (Pull-out, Force), see Figure 12.3.
LFED	Fed length, see explanation below.

**Remarks:**

1. The retractor node should not be on any belt elements. The element defined should have one node coincident with the retractor node but should not be inside the retractor.
2. At least one sensor should be defined.
3. The first point of the load curve should be  $(0, T_{\min})$ .  $T_{\min}$  is the minimum tension. All subsequent tension values should be greater than  $T_{\min}$ .
4. The unloading curve should start at zero tension and increase monotonically (i.e., no segments of negative or zero slope).

Retractors allow belt material to be paid out into a belt element. Retractors operate in one of two regimes: unlocked when the belt material is paid out, or reeled in under constant tension and locked when a user defined force-pullout relationship applies.

The retractor is initially unlocked, and the following sequence of events must occur for it to become locked:

1. Any one of up to four sensors must be triggered. (The sensors are described below.)
2. Then a user-defined time delay occurs.
3. Then a user-defined length of belt must be paid out (optional).
4. Then the retractor locks and once locked, it remains locked.

In the unlocked regime, the retractor attempts to apply a constant tension to the belt. This feature allows an initial tightening of the belt and takes up any slack whenever it occurs. The tension value is taken from the first point on the force-pullout load curve. The maximum

## \*ELEMENT

---

rate of pull out or pull in is given by  $0.01 \times \text{fed length}$  per time step. Because of this, the constant tension value is not always achieved.

In the locked regime, a user-defined curve describes the relationship between the force in the attached element and the amount of belt material paid out. If the tension in the belt subsequently relaxes, a different user-defined curve applies for unloading. The unloading curve is followed until the minimum tension is reached.

The curves are defined in terms of initial length of belt. For example, if a belt is marked at 10mm intervals and then wound onto a retractor, and the force required to make each mark emerge from the (locked) retractor is recorded, the curves used for input would be as follows:

- 0     Minimum tension (should be > zero)
- 10mm   Force to emergence of first mark
- 20mm   Force to emergence of second mark
- •
- •
- •

Pyrotechnic pretensions may be defined which cause the retractor to pull in the belt at a predetermined rate. This overrides the retractor force-pullout relationship from the moment when the pretensioner activates.

If desired, belt elements may be defined which are initially inside the retractor. These will emerge as belt material is paid out, and may return into the retractor if sufficient material is reeled in during unloading.

Elements e2, e3 and e4 are initially inside the retractor, which is paying out material into element e1. When the retractor has fed  $L_{\text{crit}}$  into e1, where

$$L_{\text{crit}} = \text{fed length} - 1.1 \times \text{minimum length}$$

(minimum length defined on belt material input)

(fed length defined on retractor input)

element e2 emerges with an unstretched length of  $1.1 \times \text{minimum length}$ ; the unstretched length of element e1 is reduced by the same amount. The force and strain in e1 are unchanged; in e2, they are set equal to those in e1. The retractor now pays out material into e2.

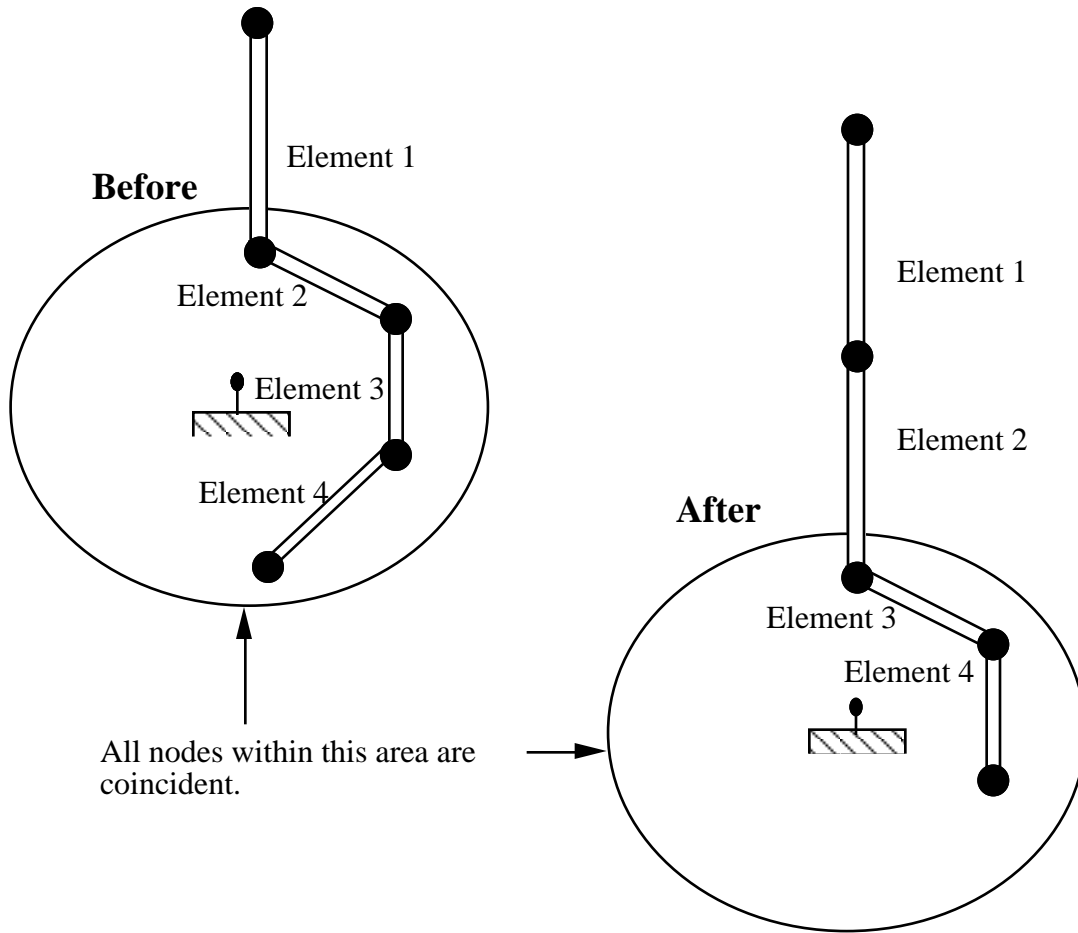
If no elements are inside the retractor, e2 can continue to extend as more material is fed into it.

As the retractor pulls in the belt (for example, during initial tightening), if the unstretched length of the mouth element becomes less than the minimum length, the element is taken into the retractor.

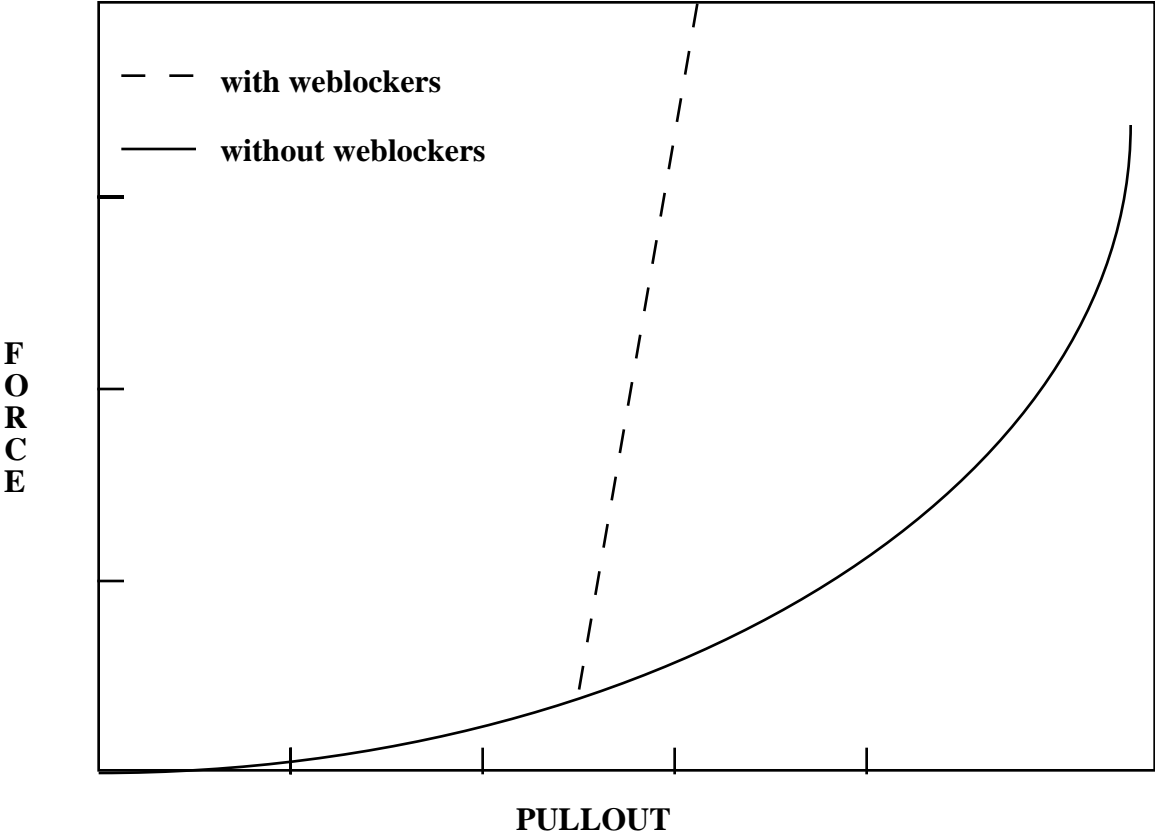
To define a retractor, the user enters the retractor node, the ‘mouth’ element (into which belt material will be fed), e1 in Figure 11.2, up to 4 sensors which can trigger unlocking, a time delay, a payout delay (optional), load and unload curve numbers, and the fed length. The retractor node is typically part of the vehicle structure; belt elements should not be connected to this node directly, but any other feature can be attached including rigid bodies. The mouth element should have a node coincident with the retractor but should not be inside the retractor. The fed length would typically be set either to a typical element initial length, for the distance between painted marks on a real belt for comparisons with high speed film. The fed length should be at least three times the minimum length.

If there are elements initially inside the retractor (e2, e3 and e4 in the Figure) they should not be referred to on the retractor input, but the retractor should be identified on the element input for these elements. Their nodes should all be coincident with the retractor node and should not be restrained or constrained. Initial slack will automatically be set to  $1.1 \times$  minimum length for these elements; this overrides any user-defined value.

Webblockers can be included within the retractor representation simply by entering a ‘locking up’ characteristic in the force pullout curve, see Figure 12.3. The final section can be very steep (but must have a finite slope).



**Figure 12.2.** Elements in a retractor.



**Figure 12.3.** Retractor force pull characteristics.

# \*ELEMENT

---

## \*ELEMENT\_SEATBELT\_SENSOR

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

### Card Format

1            2            3            4            5            6            7            8

Variable	SBSID	SBSTYP	SBSFL					
Type	I	I	I					
Default	0	0	0					
Remarks								

### Second Card if SBSTYP=1

1            2            3            4            5            6            7            8

Variable	NID	DOF	ACC	ATIME				
Type	I	I	F	F				
Default	0	0	0.0	0.0				
Remarks	1							

**Second Card if SBSTYP=2**

1 2 3 4 5 6 7 8

Variable	SBRID	PULRAT	PULTIM					
Type	I	F	F					
Default	0	0.0	0.0					
Remarks								

**Second Card if SBSTYP=3**

1 2 3 4 5 6 7 8

Variable	TIME							
Type	F							
Default	0.0							
Remarks								

**Second Card if SBSTYP=4**

1 2 3 4 5 6 7 8

Variable	NID1	NID2	DMX	DMN				
Type	I	I	F	F				
Default	0	0	0.0	0.0				
Remarks			2	2				

## \*ELEMENT

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
SBSID	Sensor ID. A unique number has to be used.
SBSTYP	Sensor type: EQ.1: acceleration of node, EQ.2: retractor pull-out rate, EQ.3: time, EQ.4: distance between nodes.
SBSFL	Sensor flag: EQ.0: sensor active during dynamic relaxation, EQ.1: sensor can be triggered during dynamic relaxation.
NID	Node ID of sensor
DOF	Degree of freedom: EQ.1: x, EQ.2: y, EQ.3: z.
ACC	Activating acceleration
ATIME	Time over which acceleration must be exceeded
SBRID	Retractor ID, see *ELEMENT_SEATBELT_RETRACTOR.
PULRAT	Rate of pull-out (length/time units)
PULTIM	Time over which rate of pull-out must be exceeded
TIME	Time at which sensor triggers
NID1	Node 1 ID
NID2	Node 2 ID
DMX	Maximum distance
DMN	Minimum distance

### **Remarks:**

1. Node should not be on rigid body, velocity boundary condition, or other 'imposed motion' feature.
2. Sensor triggers when the distance between the two nodes is  $d \geq d_{\max}$  or  $d \leq d_{\min}$ .

Sensors are used to trigger locking of retractors and activate pretensioners. Four types of sensors are available which trigger according to the following criteria:



**Type 1** – When the magnitude of x-, y-, or z- acceleration of a given node has remained above a given level continuously for a given time, the sensor triggers. This does not work with nodes on rigid bodies.

**Type 2** – When the rate of belt payout from a given retractor has remained above a given level continuously for a given time, the sensor triggers.

**Type 3** – The sensor triggers at a given time.

**Type 4** – The sensor triggers when the distance between two nodes exceeds a given maximum or becomes less than a given minimum. This type of sensor is intended for use with an explicit mass/spring representation of the sensor mechanism.

By default, the sensors are inactive during dynamic relaxation. This allows initial tightening of the belt and positioning of the occupant on the seat without locking the retractor or firing any pretensioners. However, a flag can be set in the sensor input to make the sensors active during the dynamic relaxation phase.

# \*ELEMENT

---

## \*ELEMENT\_SEATBELT\_SLIPRING

Purpose: Define seat belt slip ring.

### Card Format

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

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

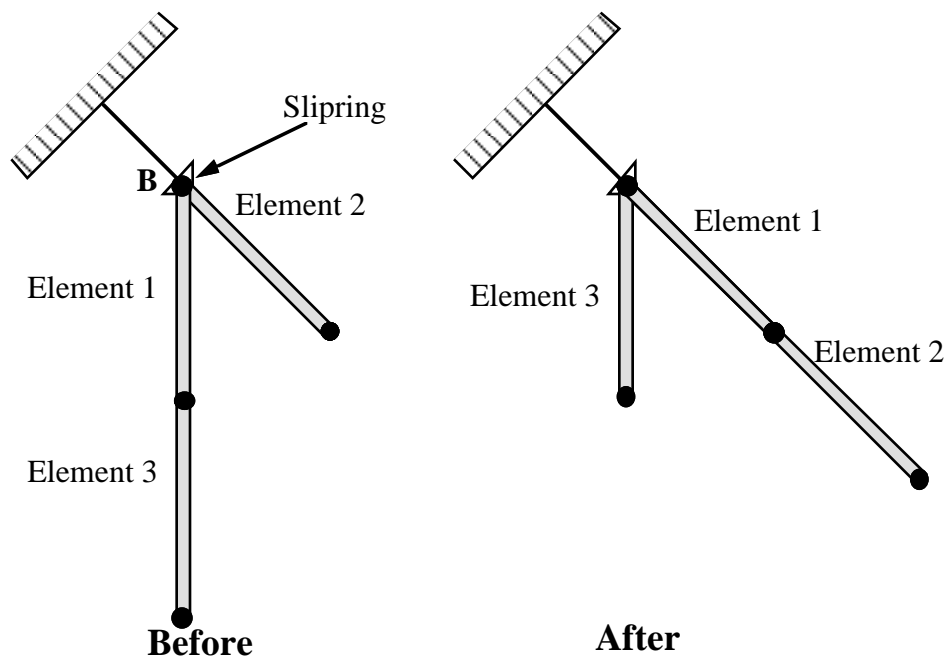
### Remarks:

Elements 1 and 2 should share a node which is coincident with the slip ring node. The slip ring node should not be on any belt elements.

Sliprings allow continuous sliding of a belt through a sharp change of angle. Two elements (1 & 2 in Figure 12.4) meet at the slipring. Node B in the belt material remains attached to the slipring node, but belt material (in the form of unstretched length) is passed from element 1 to element 2 to achieve slip. The amount of slip at each timestep is calculated from the ratio of forces in elements 1 and 2. The ratio of forces is determined by the relative angle between elements 1 and 2 and the coefficient of friction,  $\mu$ . The tension in the belts are taken as  $T_1$  and  $T_2$ , where  $T_2$  is on the high tension side and  $T_1$  is the force on the low tension side. Thus, if  $T_2$  is sufficiently close to  $T_1$ , no slip occurs; otherwise, slip is just sufficient to reduce the ratio  $T_2/T_1$  to  $e^{\mu\Theta}$ . No slip occurs if both elements are slack. The out-of-balance force at node B is reacted on the slipring node; the motion of node B follows that of slipring node.

If, due to slip through the sliping, 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 sliping and reappears on the other side (see Figure 12.4). 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 sliping.

To define a sliping, the user identifies the two belt elements which meet at the sliping, the friction coefficient, and the sliping node. The two elements must have a common node coincident with the sliping node. No attempt should be made to restrain or constrain the common node for its motion will automatically be constrained to follow the sliping node. Typically, the sliping node is part of the vehicle body structure and, therefore, belt elements should not be connected to this node directly, but any other feature can be attached, including rigid bodies.



**Figure 12.4.** Elements passing through sliping.

# \*ELEMENT

---

\*ELEMENT\_SHELL\_{*OPTION*}

Available options include:

**<BLANK>**  
**THICKNESS**  
**BETA**

Purpose: Define three and four noded elements including 3D shells, membranes, 2D plane stress, plane strain, and axisymmetric solids. The type of the element and its formulation is specified through the part ID (see \*PART) and the section ID (see \*SECTION\_SHELL). Also, the thickness of each element can be specified when applicable on the element cards or else a default thickness value is used from the section definition. For orthotropic and anisotropic materials a local material angle (variable PSI) can be defined which is cumulative with the integration point angles specified in \*SECTION\_SHELL.

## Card Format (10I8)

Card 1            1            2            3            4            5            6            7            8            9            10

Variable	EID	PID	N1	N2	N3	N4				
Type	I	I	I	I	I	I				
Default	none	none	none	none	none	none				
Remarks			3	3	3	3				

## \*ELEMENT

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

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

### VARIABLE

### DESCRIPTION

EID	Element ID. Chose a unique number with respect to other elements.
PID	Part ID, see *PART.
N1	Nodal point 1
N2	Nodal point 2
N3	Nodal point 3
N4	Nodal point 4
THIC1	Shell thickness at node 1
THIC2	Shell thickness at node 2
THIC3	Shell thickness at node 3
THIC4	Shell thickness at node 4
PSI	Orthotropic material angle offset measured from the reference (1-2 element side) axis, see remark 6 below. The angle is given in degrees.

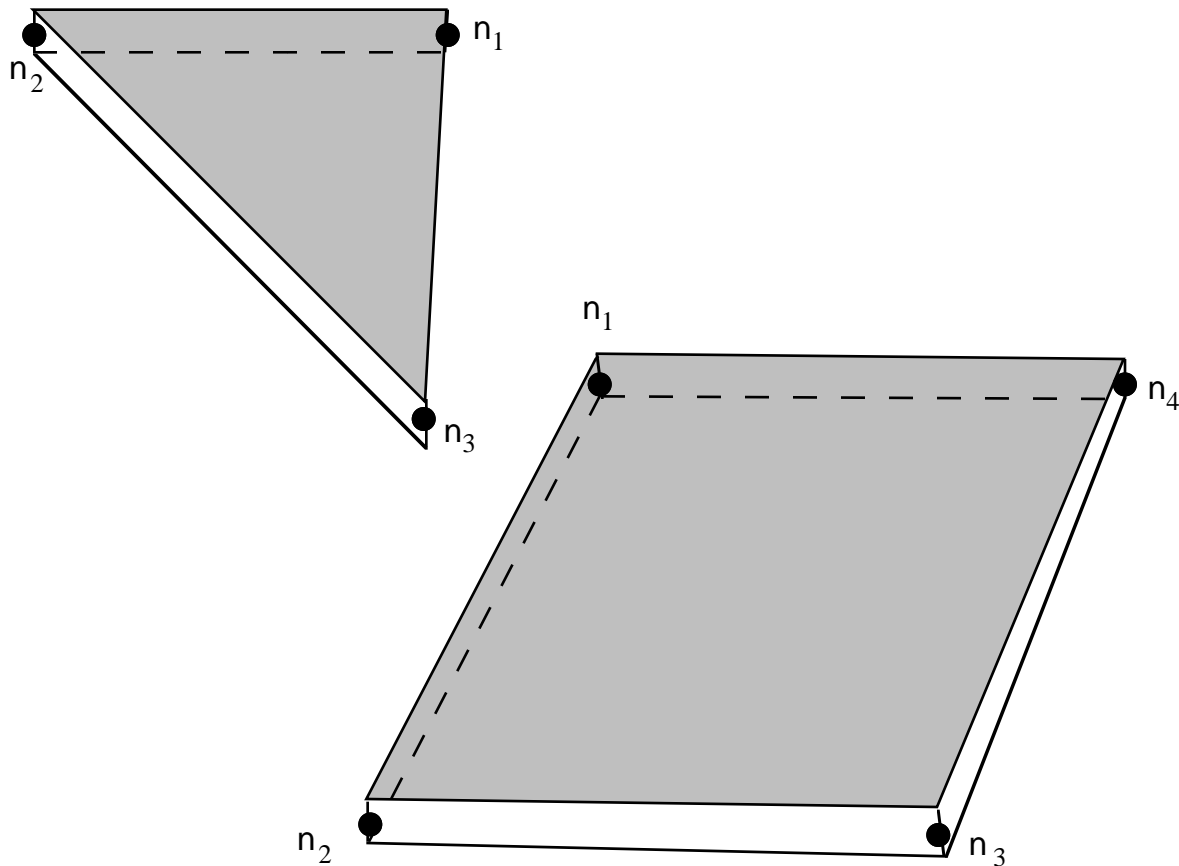
### **Remarks:**

1. Default values in place of zero shell thicknesses are taken from the cross-section property definition of the PID, see \*SECTION\_SHELL.
2. PSI is defined only for orthotropic and anisotropic materials.

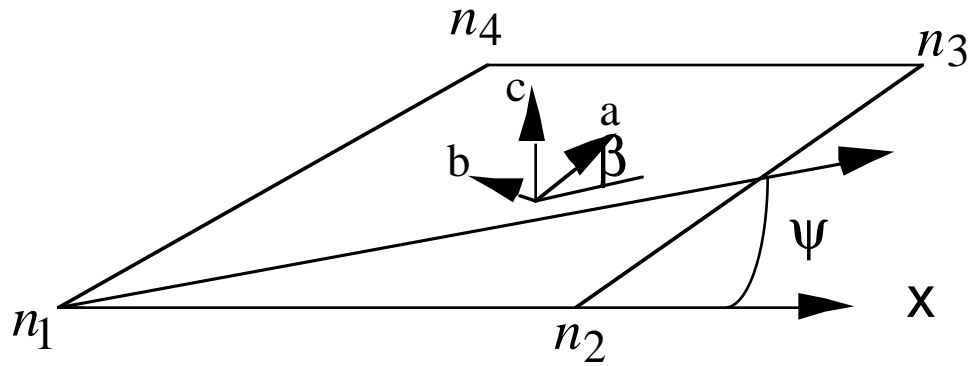
## \*ELEMENT

---

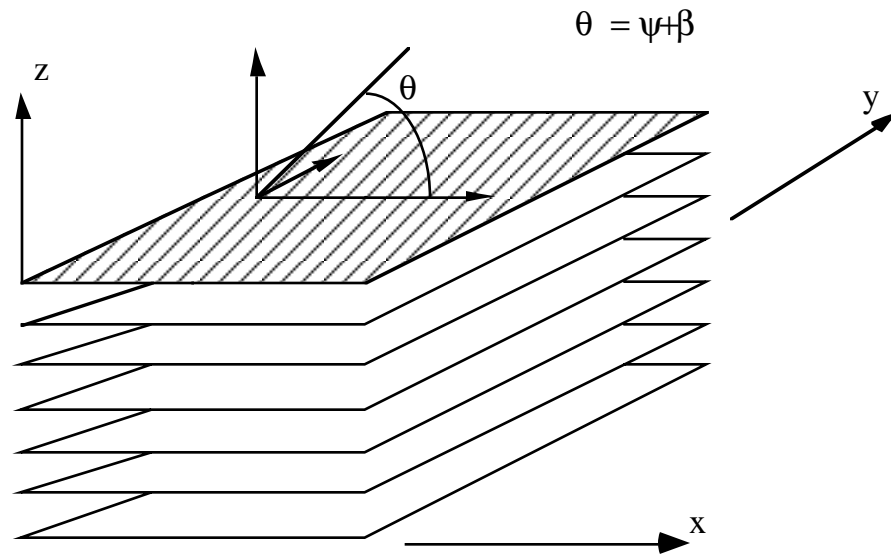
3. Counterclockwise node numbering determines the top surface, see Figure 12.5.
4. Stresses and strain output in the binary databases are by default given in the global coordinate system. Stress resultants are output in the local coordinate system for the shell element.
5. Interior angles must be less than 180 degrees.
6. To allow for an arbitrary orientation of the shell elements within the finite element mesh, each ply in the composite has a unique material orientation angle which measures the offset from some reference in the element. Each integration point through the shell thickness, typically though not limited to one point per ply, requires the definition of the orientation angle at that point. The reference is determined by the angle  $\psi$ , which can be defined for each element on the element card, and is measured from the 1-2 element side. Figures 12.6 and 12.7 depict these angles.



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



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



**Figure 12.7.** A multi-layer laminate can be defined. The angle  $\beta_i$  is defined for the  $i$ th lamina (integration point), see \*SECTION\_SHELL.

# \*ELEMENT

---

\*ELEMENT\_SOLID\_{*OPTION*}

Available options include:

<BLANK>

ORTHO

Purpose: Define three dimensional solid elements including 4 noded tetrahedrons and 8-noded hexahedrons. The type of solid element and its formulation is specified through the part ID (see \*PART) and the section ID (see \*SECTION\_SOLID\_ *OPTION*). Also, a local coordinate system for orthotropic and anisotropic materials can be defined by using the ORTHO option.

## Card Format (10I8)

1 2 3 4 5 6 7 8 9 10

Variable	EID	PID	N1	N2	N3	N4	N5	N6	N7	N8
Type	I	I	I	I	I	I	I	I	I	I
Default	none	none	none	none	none	none	none	none	none	none
Remarks	1									

## Optional Cards (Required if ORTHO is specified after the keyword)

Optional card 1 1 2 3 4 5 6 7 8 9 10

Variable	A1	A2	A3		
Type	F	F	F		
Default	0.	0.	0.		
Remarks	2				



# **\*ELEMENT**

Optional card 2      1            2            3            4            5            6            7            8            9            10

Variable	D1	D2	D3		
Type	F	F	F		
Default	0.	0.	0.		
Remarks	2				

## **VARIABLE**

## **DESCRIPTION**

EID	Element ID. A unique number has to be chosen.
PID	Part ID, see *PART.
N1	Nodal point 1
N2	Nodal point 2
N3	Nodal point 3
.	.
.	.
N8	Nodal point 8
A1	x-component of local material direction a.
A2	y-component of local material direction a.
A3	z-component of local material direction a.
D1	x-component of vector in the plane of the material vectors a and b.
D2	y-component of vector in the plane of the material vectors a and b.
D3	z-component of vector in the plane of the material vectors a and b.

## \*ELEMENT

---

### Remarks:

1. Four, six, and eight node elements are depicted in Figure 12.8 where the ordering of the nodal points is shown. This ordering must be followed or code termination will occur during the initialization phase with a negative volume message. Input of nodes on the element cards for the two degenerate elements is given by:

4-node N1, N2, N3, N4, N4, N4, N4, N4

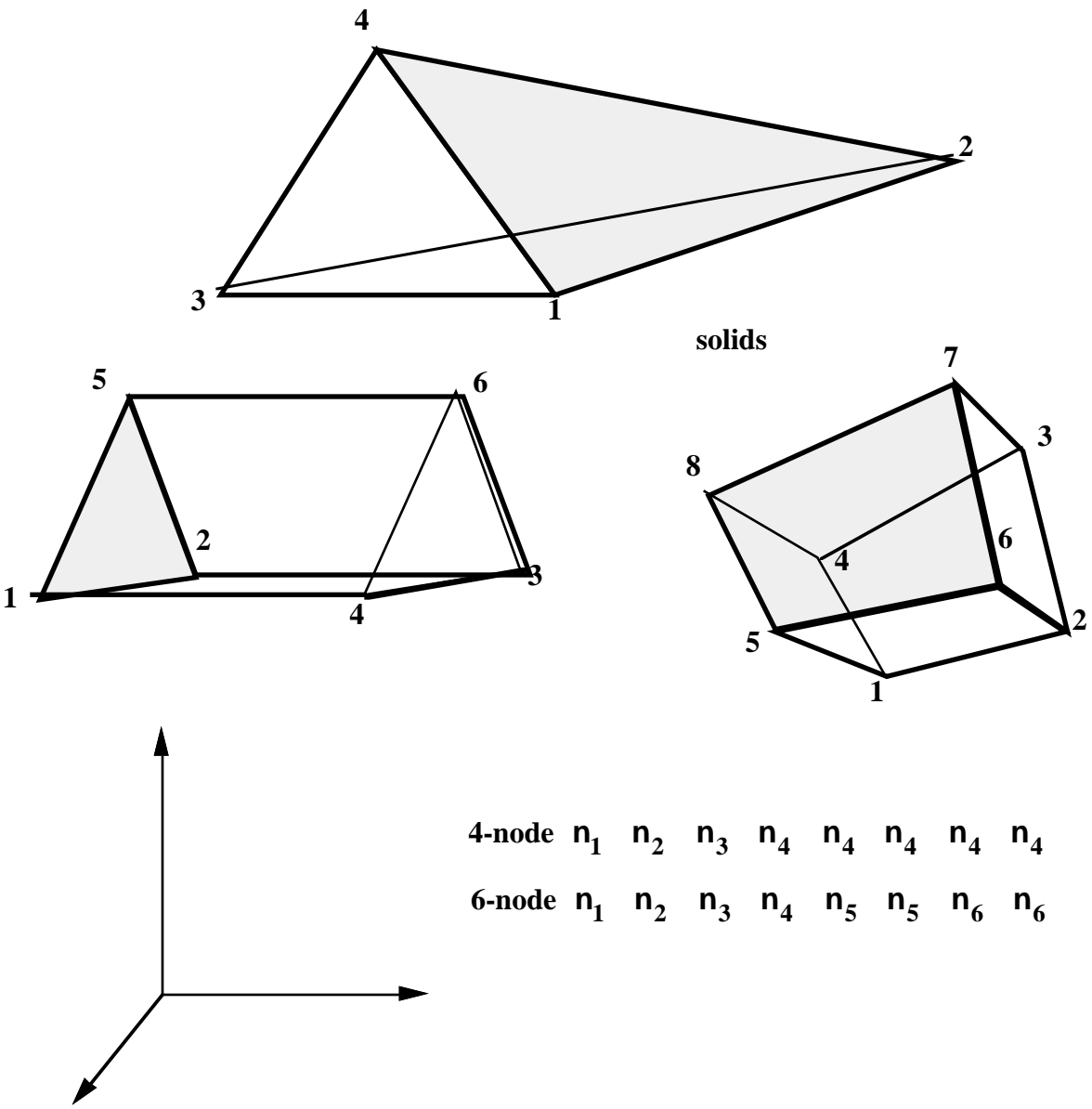
6-node N1, N2, N3, N4, N5, N5, N6, N6

2. For the orthotropic and anisotropic material models the local directions may be defined on the second card following the element connectivity definition. The local directions are then computed from the two vectors such that (see Figure 12.9):

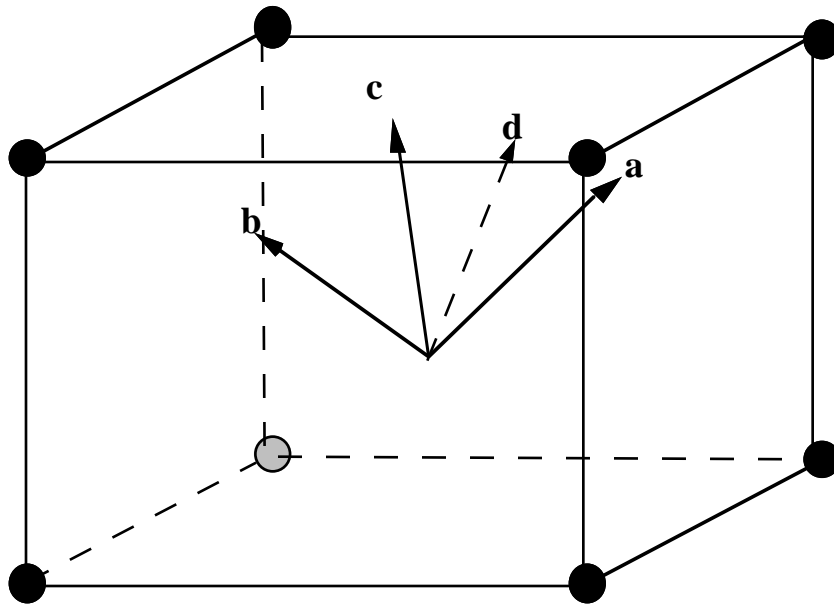
$$\underline{\underline{c}} = \underline{\underline{a}} \times \underline{\underline{d}} \text{ and } \underline{\underline{b}} = \underline{\underline{c}} \times \underline{\underline{a}}.$$

These vectors are internally normalized within LS-DYNA.

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



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



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

# **\*ELEMENT**

---

## **\*ELEMENT\_SPH**

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

### **Card Format (2I8,E16.0)**

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

#### VARIABLE

#### DESCRIPTION

NID	Node ID and Element ID are the same for the SPH option.
PID	Part ID to which this node (element) belongs.
MASS	Mass value

# **\*ELEMENT**

---

## **\*ELEMENT\_TRIM**

Purpose: Define a part subset to be trimmed by \*DEFINE\_CURVE\_TRIM.

### **Card Format**

Card 1            1            2            3            4            5            6            7            8            9            10

Variable	PSID										
Type	I										
Default	none										
Remarks	1,2,3										

---

#### **VARIABLE**

#### **DESCRIPTION**

PSID

Part set ID for trimming, see \*SET\_PART.

### **Remarks:**

1. This command in combination with \*DEFINE\_CURVE\_TRIM trims the requested parts before the job starts.
2. In case this command does not exist and only \*DEFINE\_CURVE\_TRIM is available in the input, the related parts are trimmed after the job is terminated.
3. Pre-trimming (\*ELEMENT\_TRIM + \*DEFINE\_CURVE\_TRIM) can handle adaptive mesh and post-trimming. The keyword \*DEFINE\_CURVE\_TRIM by itself cannot deal with an adaptive mesh. See the detailed procedure outlined in the Remarks in the section \*INTERFACE\_SPRINGBACK.

**\*ELEMENT\_TSHELL**

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

**Card Format (10I8)**

	1	2	3	4	5	6	7	8	9	10
Variable	EID	PID	N <sub>1</sub>	N <sub>2</sub>	N <sub>3</sub>	N <sub>4</sub>	N <sub>5</sub>	N <sub>6</sub>	N <sub>7</sub>	N <sub>8</sub>
Type	I	I	I	I	I	I	I	I	I	I
Default	none	none	none	none	none	none	none	none	none	none
Remarks			1							

**VARIABLE****DESCRIPTION**

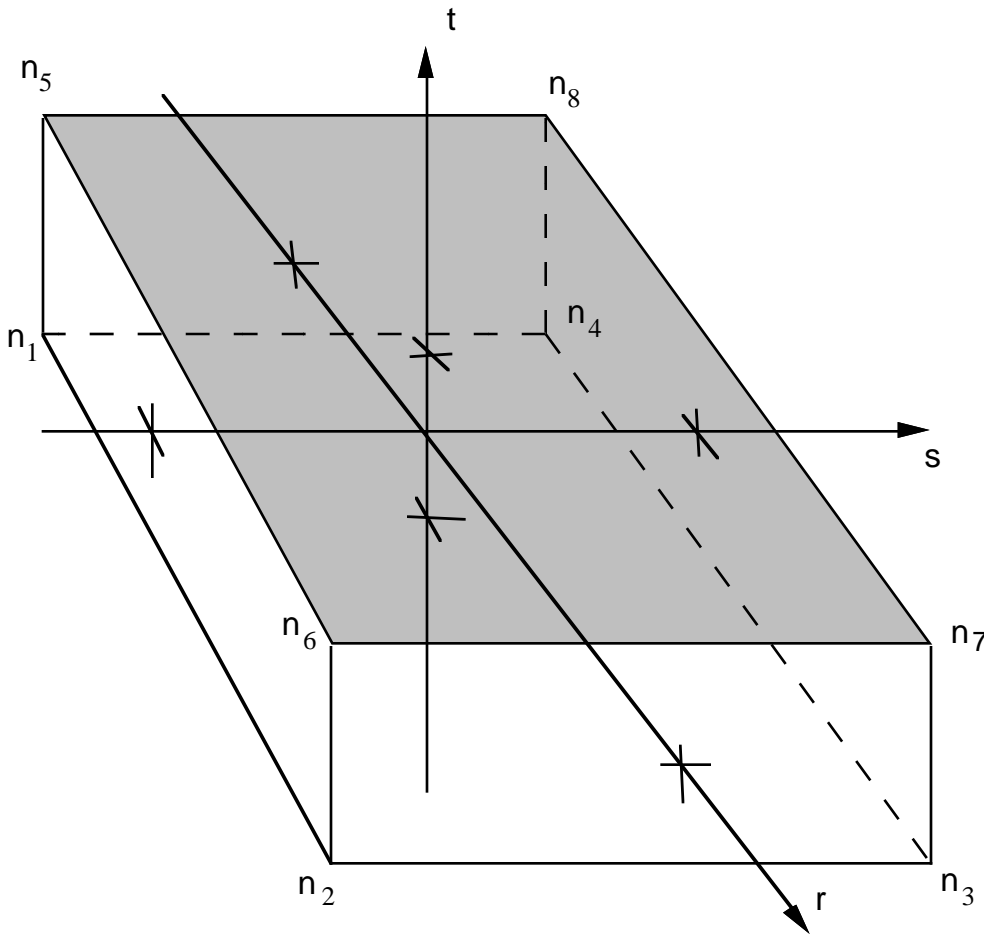
EID	Element ID. Unique numbers have to be used.
PID	Part ID, see *PART.
N1	Nodal point 1
N2	Nodal point 2
N3	Nodal point 3
.	.
.	.
N8	Nodal point 8

# \*ELEMENT

---

## Remarks:

1. The correct numbering of the nodes is essential for correct use. Nodes  $n_1$  to  $n_4$  define the lower surface, and nodes  $n_5$  to  $n_8$  define the upper surface. If one point integration is used (see \*SECTION\_TSHELL), the integration points then lie along the  $t$ -axis as depicted in Figure 12.10. Two by two selective reduced integration is also available. Extreme care must be used in defining the connectivity to insure proper orientation.
2. The stresses for this shell element are output in the global coordinate system.
3. To define a thick shell wedge element nodal pairs  $n_3$  &  $n_4$  and  $n_7$  &  $n_8$  are repeated. The ordering is then  $n_1, n_2, n_3, n_3, n_4, n_5, n_6, n_6$ , where nodes  $n_1, n_2, n_3$  form the lower triangular face and nodes  $n_4, n_5, n_6$  for the upper triangular face of the wedge.



**Figure 12.10.** Solid 8-node Shell Element.



# **\*EOS**

LS-DYNA has historically referenced equations of state by type identifiers. Below these identifiers are given with the corresponding keyword name in the order that they appear in the manual. The equations of state can be used with a subset of the materials that are available for solid elements.

<b>TYPE 1:</b>	<b>*EOS_LINEAR_POLYNOMIAL</b>
<b>TYPE 2:</b>	<b>*EOS_JWL</b>
<b>TYPE 3:</b>	<b>*EOS_SACK_TUESDAY</b>
<b>TYPE 4:</b>	<b>*EOS_GRUNEISEN</b>
<b>TYPE 5:</b>	<b>*EOS_RATIO_OF_POLYNOMIALS</b>
<b>TYPE 6:</b>	<b>*EOS_LINEAR_POLYNOMIAL_WITH_ENERGY_LEAK</b>
<b>TYPE 7:</b>	<b>*EOS_IGNITION_AND_GROWTH_OF_REACTION_IN_HE</b>
<b>TYPE 8:</b>	<b>*EOS_TABULATED_COMPACTION</b>
<b>TYPE 9:</b>	<b>*EOS_TABULATED</b>
<b>TYPE 10:</b>	<b>*EOS_PROPELLANT_DEFLAGRATION</b>
<b>TYPE 11:</b>	<b>*EOS_TENSOR_PORE_COLLAPSE</b>
<b>TYPE 14:</b>	<b>*EOS_JWLB</b>

# \*EOS

---

## \*EOS\_LINEAR\_POLYNOMIAL

Purpose: Define coefficients for linear polynomial EOS.

### Card Format

Card 1            1            2            3            4            5            6            7            8

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

Card 2

Variable	E0	V0						
Type	F	F						

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

**Remarks:**

The linear polynomial equation of state is linear in internal energy. The pressure is given by:

$$P = C_0 + C_1\mu + C_2\mu^2 + C_3\mu^3 + (C_4 + C_5\mu + C_6\mu^2) E.$$

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  $\gamma$  is the ratio of specific heats. The pressure is then given by:

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

The units of E are the units of pressure.

# \*EOS

---

## \*EOS\_JWL

This is Equation of state Form 2.

### Card Format

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

<u>VARIABLE</u>	<u>DESCRIPTION</u>
EOSID	Equation of state label
A	
B	
R1	
R2	
OMEG	
E0	
VO	Initial realtive volume.

### **Remarks:**

The JWL equation of state defines the pressure as

$$p = A \left( 1 - \frac{\omega}{R_1 V} \right) e^{-R_1 V} + B \left( 1 - \frac{\omega}{R_2 V} \right) e^{-R_2 V} + \frac{\omega E}{V},$$

and is usually used for detonation products of high explosives.

**\*EOS\_SACK\_TUESDAY**

This is Equation of state Form 3.

**Card Format**

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

**VARIABLE****DESCRIPTION**

EOSID	Equation of state label
A1	
A2	
A3	
B1	
B2	
E0	Initial internal energy
V0	Initial relative volume

**Remarks:**

The Sack equation of state defines pressure as

$$p = \frac{A_3}{V^{A_1}} e^{-A_2 V} \left( 1 - \frac{B_1}{V} \right) + \frac{B_2}{V} E$$

and is used for detonation products of high explosives.

# \*EOS

---

## \*EOS\_GRUNEISEN

This is Equation of state Form 4.

### Card Format

1            2            3            4            5            6            7            8

Variable	EOSID	C	S1	S2	S3	GAMAO	A	E0
Type	I	F	F	F	F	F	F	F

Card 2

Variable	V0							
Type	F							

<u>VARIABLE</u>	<u>DESCRIPTION</u>
EOSID	Equation of state ID
C	
S1	
S2	
S3	
GAMAO	
A	
E0	Initial internal energy
V0	Initial relative volume

**Remarks:**

The Gruneisen equation of state with cubic shock velocity-particle velocity defines pressure for compressed materials as

$$p = \frac{\rho_0 C^2 \mu \left[ 1 + \left( 1 - \frac{\gamma_0}{2} \right) \mu - \frac{a}{2} \mu^2 \right]}{\left[ 1 - (S_1 - 1) \mu - S_2 \frac{\mu^2}{\mu + 1} - S_3 \frac{\mu^3}{(\mu + 1)^2} \right]^2} + (\gamma_0 + a\mu) E.$$

and for expanded materials as

$$p = \rho_0 C^2 \mu + (\gamma_0 + a\mu) E.$$

where C is the intercept of the  $v_s$ - $v_p$  curve;  $S_1$ ,  $S_2$ , and  $S_3$  are the coefficients of the slope of the  $v_s$ - $v_p$  curve;  $\gamma_0$  is the Gruneisen gamma; a is the first order volume correction to  $\gamma_0$ ; and  $\mu = \frac{\rho}{\rho_0} - 1$ .

# \*EOS

---

## \*EOS\_RATIO\_OF\_POLYNOMIALS

This is Equation of state Form 5.

**Card Format (I10)** for card 1, **(4E20.0)** all following cards.

Card 1                    1

Variable	EOSID
Type	I

Card 2                    1                    2                    3                    4

Variable	A10	A11	A12	A13
Type	F	F	F	F

Card 3                    1                    2                    3                    4

Variable	A20	A21	A22	A23
Type	F	F	F	F

Card 4                    1                    2                    3                    4

Variable	A30	A31	A32	A33
Type	F	F	F	F

Card 5                    1                    2                    3                    4

Variable	A40	A41	A42	A43
Type	F	F	F	F



---

Card 6                    1                    2                    3                    4

Variable	A50	A51	A52	A53
Type	F	F	F	F

Card 7                    1                    2                    3                    4

Variable	A60	A61	A62	A63
Type	F	F	F	F

Card 8                    1                    2                    3                    4

Variable	A70	A71	A72	A73
Type	F	F	F	F

Card 9                    1                    2

Variable	A14	A24		
Type	F	F		

Card 10                    1                    2                    3                    4

Variable	ALPH	BETA	E0	V0
Type	F	F	F	F

# \*EOS

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
EOSID	Equation of state ID
A10	
A11	
A12	
A13	
A20	
A21	
A22	
A23	
A30	
A31	
A32	
A33	
A40	
A41	
A42	
A43	
A50	
A51	
A52	
A53	
A60	
A61	
A62	
A63	
A70	
A71	

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
A72	
A73	
A14	
A24	
ALPHA	$\alpha$
BETA	$\beta$
E0	Initial internal energy
V0	Initial relative volume

**Remarks:**

The ratio of polynomials equation of state defines the pressure as

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

where

$$F_i = \sum_{j=0}^n A_{ij} \mu^j \quad n = 4 \text{ if } i < 3$$

$$\mu = \frac{\rho}{\rho_0} - 1 \quad n = 3 \text{ if } i \geq 3$$

In expanded elements  $F_1$  is replaced by  $F'_1 = F_1 + \beta \mu^2$ . By setting coefficient  $A_{10} = 1.0$ , the delta-phase pressure modeling for this material will be initiated. The code will reset it to 0.0 after setting flags.

# \*EOS

---

## \*EOS\_LINEAR\_POLYNOMIAL\_WITH\_ENERGY\_LEAK

### Card Format

Card 1            1            2            3            4            5            6            7            8

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

Card 2

Variable	E0	V0	LCID					
Type	F	F	I					

<u>VARIABLE</u>	<u>DESCRIPTION</u>
EOSID	Equation of state label
C0	
C1	
C2	
C3	
C4	
C5	
C6	
E0	Initial internal energy
V0	Initial relative volume
LCID	Load curve ID defining the energy deposition rate

**Remarks:**

This polynomial equation of state, linear in the internal energy per initial volume,  $E$ , is given by

$$p = C_0 + C_1\mu + C_2\mu^2 + C_3\mu^3 + (C_4 + C_5\mu + C_6\mu^2)E$$

in which  $C_0$ ,  $C_1$ ,  $C_2$ ,  $C_3$ ,  $C_4$ ,  $C_5$ , and  $C_6$  are user defined constants and

$$\mu = \frac{1}{V} - 1.$$

where  $V$  is the relative volume. In expanded elements, we set the coefficients of  $\mu^2$  to zero, i.e.,

$$C_2 = C_6 = 0$$

Internal energy,  $E$ , is increased according to an energy deposition rate versus time curve whose ID is defined in the input.

# \*EOS

---

## \*EOS\_IGNITION\_AND\_GROWTH\_OF\_REACTION\_IN\_HE

### Card Format

Card 1            1            2            3            4            5            6            7            8

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

Card 2

Variable	R2	R3	R5	R6	FMXIG	FREQ	GROW1	EM
Type	F	F	F	F	F	F	F	F

Card 3

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

Card 4

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

---

### VARIABLE

### DESCRIPTION

EOSID            Equation of state label  
A  
B  
XP1

---

<b>VARIABLE</b>	<b>DESCRIPTION</b>
XP2	
FRER	
G	
R1	
R2	
R3	
R5	
R6	
FMXIG	
FREQ	
GROW1	
EM	
AR1	
ES1	
CVP	Heat capacity of reaction products
CVR	Heat capacity of unreacted HE
EETAL	
CCRIT	
ENQ	Heat of reaction
TMP0	Initial temperature (°K)
GROW2	
AR2	
ES2	
EN	
FMXGR	
FMNGR	

---

# \*EOS

---

## Remarks:

Equation of State Form 7 is used to calculate the shock initiation (or failure to initiate) and detonation wave propagation of solid high explosives. It should be used instead of the ideal HE burn options whenever there is a question whether the HE will react, there is a finite time required for a shock wave to build up to detonation, and/or there is a finite thickness of the chemical reaction zone in a detonation wave. At relatively low initial pressures (<2-3 GPa), this equation of state should be used with material type 10 for accurate calculations of the unreacted HE behavior. At higher initial pressures, material type 9 can be used. A JWL equation of state defines the pressure in the unreacted explosive as

$$P_e = r_1 e^{-r_5 V_e} + r_2 e^{-r_6 V_e} + r_3 \frac{T_e}{V_e} \quad (r_3 = \omega_e \text{ cvr})$$

where  $V_e$  and  $T_e$  are the relative volume and temperature, respectively, of the unreacted explosive. Another JWL equation of state defines the pressure in the reaction products as

$$P_p = a e^{-x p_1 V_p} + b e^{-x p_2 V_p} + \frac{g T_p}{V_p} \quad (g = \omega_p \text{ cvp})$$

where  $V_p$  and  $T_p$  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  $F$  ( $F=0$  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.,

$$V = (1-F) V_e + V_p$$

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

$$\frac{\partial F}{\partial t} = \text{freq} (1-F)^{\text{frer}} (V_e^{-1} - 1 - \text{ccrit})^{\text{cetal}} \quad (\text{Ignition})$$

$$+ \text{grow1} (1-F)^{\text{es1}} F^{\text{ar1}} p^{\text{em}} \quad (\text{Growth})$$

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

The ignition rate is set equal to zero when  $F \geq \text{fmxig}$ , the growth rate is set equal to zero when  $F \geq \text{fmxgr}$ , and the completion rate is set equal to zero when  $F \leq \text{fmngr}$ .

Details of the computational methods and many examples of one and two dimensional shock initiation and detonation wave calculation can be found in the references. Unfortunately, sufficient experimental data has been obtained for only two solid explosives to develop very reliable shock initiation models: PBX-9504 (and the related HMX-based explosives LX-14, LX-10, LX-04, etc.) and LX-17 (the insensitive TATB-based explosive). Reactive flow models have been developed for



other explosives (TNT, PETN, Composition B, propellants, etc.) but are based on very limited experimental data.

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

# \*EOS

---

## \*EOS\_TABULATED\_COMPACTON

### Card Format

Card 1            1            2            3            4            5            6            7            8

Variable	EOSID	GAMA	E0	V0				
Type	I	F	F	F				

### Card Format (5E16.0)

Card 2            1            2            3            4            5

Variable	EV1	EV2	EV3	EV4	EV5
Type	F	F	F	F	F

Card 3

Variable	EV6	EV7	EV8	EV9	EV10
Type	F	F	F	F	F

Repeat Cards 2 and 3 for  $C_i$ ,  $T_i$ , and  $K_i$ . A total of 9 cards must be defined.

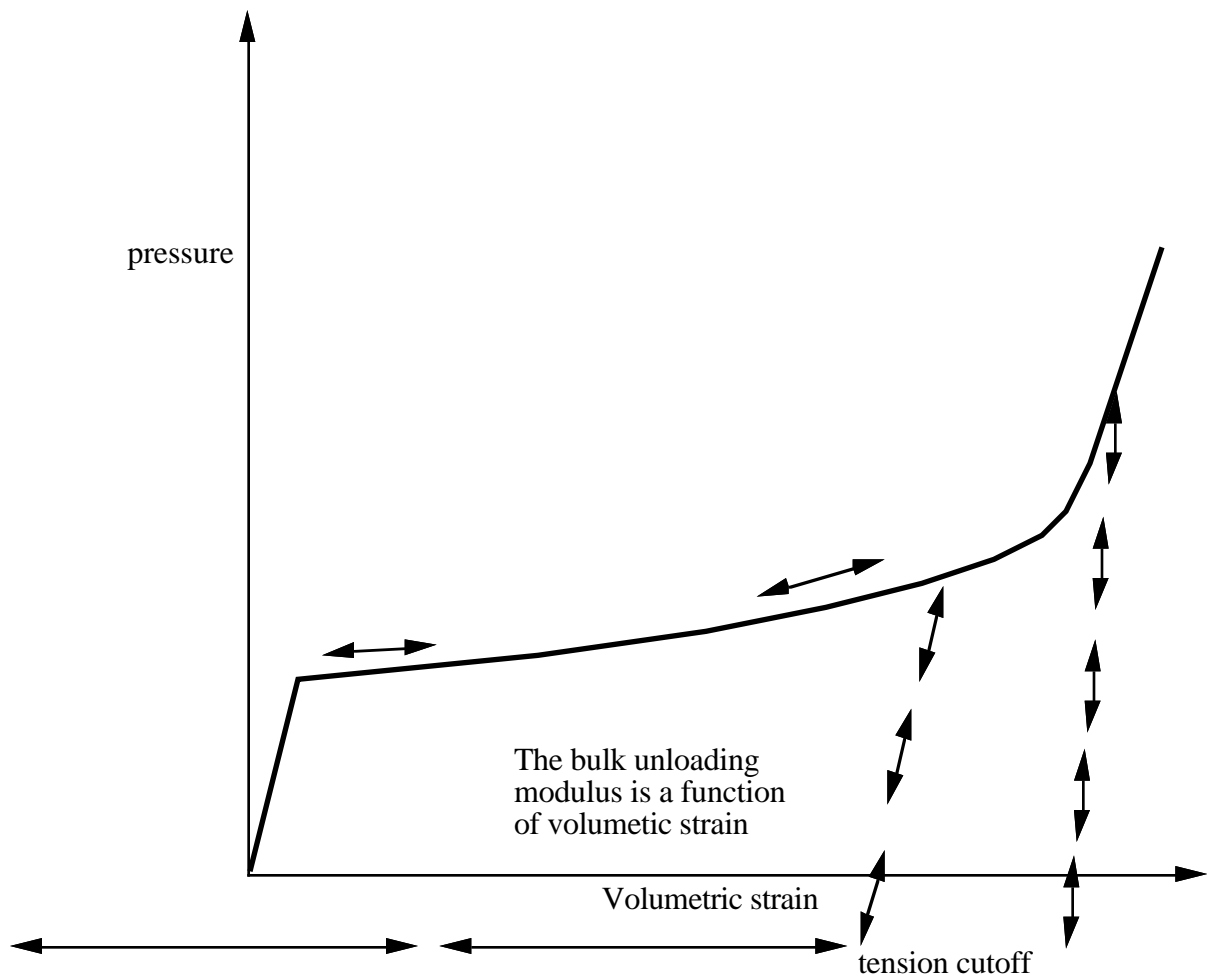
<u>VARIABLE</u>	<u>DESCRIPTION</u>
EOSID	Equation of state label
$\epsilon V1, \epsilon V2, \dots, \epsilon VN$	$\ln V$
$C1, C2, \dots, CN$	
$T1, T2, \dots, TN$	
$K1, K2, \dots, KN$	
GAMA	$\gamma$
E0	Initial internal energy
V0	Initial relative volume

**Remarks:**

The tabulated compaction model is linear in internal energy. Pressure is defined by

$$p = C(\epsilon_v) = \gamma T(\epsilon_v)E$$

in the loading phase. The volumetric strain,  $\epsilon_v$  is given by the natural logarithm of the relative volume. Unloading occurs along the unloading bulk modulus to the pressure cutoff. Reloading always follows the unloading path to the point where unloading began, and continues on the loading path, see Figure 13.1. Up to 10 points and as few as 2 may be used when defining the tabulated functions. LS-DYNA will extrapolate to find the pressure if necessary.



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

# \*EOS

---

## \*EOS\_TABULATED

This is Equation of state Form 9.

### Card Format

Card 1            1            2            3            4            5            6            7            8

Variable	EOSID	GAMA	E0	VO				
Type	I	F	F	F				

### Card Format (5E16.0)

Card 2            1            2            3            4            5

Variable	EV1	EV2	EV3	EV4	EV5
Type	F	F	F	F	F

Card 3

Variable	EV6	EV7	EV8	EV9	EV10
Type	F	F	F	F	F

Repeat Cards 2 and 3 for  $C_i$  and  $T_i$ . A total of 7 cards must be defined.

<u>VARIABLE</u>	<u>DESCRIPTION</u>
EOSID	Equation of state label
$\epsilon V1, \epsilon V2, \dots, \epsilon VN$	$\ln V$
$C1, C2, \dots, CN$	
$T1, T2, \dots, TN$	

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
GAMA	$\gamma$
E0	Initial internal energy
V0	Initial relative volume

**Remarks:**

The tabulated equation of state model is linear in internal energy. Pressure is defined by

$$P = C(\epsilon_V) + \gamma T(\epsilon_V) E$$

The volumetric strain,  $\epsilon_V$  is given by the natural logarithm of the relative volume. Up to 10 points and as few as 2 may be used when defining the tabulated functions. LS-DYNA will extrapolate to find the pressure if necessary.

# \*EOS

---

## \*EOS\_PROPELLANT\_DEFLAGRATION

This Equation of state (10) has been added to model airbag propellants.

### Card Format

Card 1            1            2            3            4            5            6            7            8

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

Card 2

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

Card 3

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

Card 4

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

Card 5

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

---

<b>VARIABLE</b>	<b>DESCRIPTION</b>
A	Product JWL coefficient
B	Product JWL coefficient
XP1	Product JWL coefficient
XP2	Product JWL coefficient
FRER	Unreacted Co-volume
G	Product $wC_V$
R1	Unreacted JWL coefficient
R2	Unreacted JWL coefficient
R3	Unreacted $wC_V$
R5	Unreacted JWL coefficient
R6	Unreacted JWL coefficient
FMXIG	Initial Fraction Reacted $F_0$
FREQ	Initial Pressure $P_0$
GROW1	First burn rate coefficient
EM	Pressure Exponent (1 <sup>st</sup> term)
AR1	Exponent on F (1 <sup>st</sup> term)
ES1	Exponent on (1-F) (1 <sup>st</sup> term)
CVP	Heat capacity products
CVR	Heat capacity unreacted
EETAL	Extra, not presently used
CCRIT	Product co-volume
ENQ	Heat of Reaction
TMP0	Initial Temperature (298°K)
GROW2	Second burn rate coefficient

## \*EOS

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
AR2	Exponent on F (2 <sup>nd</sup> term)
ES2	Exponent on (1-F) (2 <sup>nd</sup> term)
EN	Pressure Exponent (2 <sup>nd</sup> term)
FMXGR	Maximum F for 1 <sup>st</sup> term
FMNGR	Minimum F for 2 <sup>nd</sup> term

### Remarks:

A deflagration (burn rate) reactive flow model requires an unreacted solid equation of state, a reaction product equation of state, a reaction rate law and a mixture rule for the two (or more) species. The mixture rule for the standard ignition and growth model [Lee and Tarver 1980] assumes that both pressures and temperatures are completely equilibrated as the reaction proceeds. However, the mixture rule can be modified to allow no thermal conduction or partial heating of the solid by the reaction product gases. For this relatively slow process of airbag propellant burn, the thermal and pressure equilibrium assumptions are valid. The equations of state currently used in the burn model are the JWL, Gruneisen, the van der Waals co-volume, and the perfect gas law, but other equations of state can be easily implemented. In this propellant burn, the gaseous nitrogen produced by the burning sodium azide obeys the perfect gas law as it fills the airbag but may have to be modelled as a van der Waal's gas at the high pressures and temperatures produced in the propellant chamber. The chemical reaction rate law is pressure, particle geometry and surface area dependant, as are most high pressure burn processes. When the temperature profile of the reacting system is well known, temperature dependent Arrhenius chemical kinetics can be used.

Since the airbag propellant composition and performance data are company private information, it is very difficult to obtain the required information for burn rate modeling. However, Imperial Chemical Industries (ICI) Corporation supplied pressure exponent, particle geometry, packing density, heat of reaction, and atmospheric pressure burn rate data which allowed us to develop the numerical model presented here for their  $\text{NaN}_3 + \text{Fe}_2\text{O}_3$  driver airbag propellant. The deflagration model, its implementation, and the results for the ICI propellant are presented in [Hallquist, et.al., 1990].

The unreacted propellant and the reaction product equations of state are both of the form:

$$p = Ae^{-R_1V} + Be^{-R_2V} + \frac{\omega C_v T}{V - d}$$

where  $p$  is pressure (in Mbars),  $V$  is the relative specific volume (inverse of relative density),  $\omega$  is the Gruneisen coefficient,  $C_v$  is heat capacity (in Mbars -cc/cc<sup>o</sup>K),  $T$  is temperature in <sup>o</sup>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  $\omega$  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:

$$\frac{\partial F}{\partial t} = Z(1-F)^y F^x p^w + V(1-F)^u Frp^s$$

for  $0 < F < F_{limit1}$                       for  $F_{limit2} < F < 1$

where  $F$  is the fraction reacted ( $F = 0$  implies no reaction,  $F = 1$  is complete reaction),  $t$  is time, and  $p$  is pressure (in Mbars),  $r, s, u, w, x, y, F_{limit1}$  and  $F_{limit2}$  are constants used to describe the pressure dependance and surface area dependence of the reaction rates. Two (or more) pressure dependant reaction rates are included in case the propellant is a mixture or exhibited a sharp change in reaction rate at some pressure or temperature. Burning surface area dependences can be approximated using the  $(1-F)^y F^x$  terms. Other forms of the reaction rate law, such as Arrhenius temperature dependent  $e^{-E/RT}$  type rates, can be used, but these require very accurate temperatures calculations. Although the theoretical justification of pressure dependent burn rates at kilobar type pressures is not complete, a vast amount of experimental burn rate versus pressure data does demonstrate this effect and hydrodynamic calculations using pressure dependent burn accurately simulate such experiments.

The deflagration reactive flow model is activated by any pressure or particle velocity increase on one or more zone boundaries in the reactive material. Such an increase creates pressure in those zones and the decomposition begins. If the pressure is relieved, the reaction rate decreases and can go to zero. This feature is important for short duration, partial decomposition reactions. If the pressure is maintained, the fraction reacted eventually reaches one and the material is completely converted to product molecules. The deflagration front rates of advance through the propellant calculated by this model for several propellants are quite close to the experimentally observed burn rate versus pressure curves.

To obtain good agreement with experimental deflagration data, the model requires an accurate description of the unreacted propellant equation of state, either an analytical fit to experimental compression data or an estimated fit based on previous experience with similar materials. This is also true for the reaction products equation of state. The more experimental burn rate, pressure production and energy delivery data available, the better the form and constants in the reaction rate equation can be determined.

Therefore, the equations used in the burn subroutine for the pressure in the unreacted propellant

$$P_u = R1 \cdot e^{-R5 \cdot V_u} + R2 \cdot e^{-R6 \cdot V_u} + \frac{R3 \cdot T_u}{V_u - FRER}$$

where  $V_u$  and  $T_u$  are the relative volume and temperature respectively of the unreacted propellant. The relative density is obviously the inverse of the relative volume. The pressure  $Pp$  in the reaction products is given by:

$$P_p = A \cdot e^{-XP1 \cdot V_p} + B \cdot e^{-XP2 \cdot V_p} + \frac{G \cdot T_p}{V_p - CCRIT}$$

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:

$$V = (1 - F) \cdot V_u + F \cdot V_p$$

## \*EOS

---

where  $V$  is the total relative volume. Other mixture assumptions can and have been used in different versions of DYNA2D/3D. The reaction rate law has the form:

$$\frac{\partial F}{\partial t} = \text{GROW1}(\text{P} + \text{FREQ})^{\text{EM}} (\text{F} + \text{FMXIG})^{\text{AR1}} (1 - \text{F} + \text{FMXIG})^{\text{ES1}} \\ + \text{GROW2}(\text{P} + \text{FREQ})^{\text{EN}} (\text{F} + \text{FMXIG})^{\text{AR2}} (1 - \text{F} + \text{FMXIG})^{\text{ES2}}$$

If  $F$  exceeds  $\text{FMXGR}$ , the  $\text{GROW1}$  term is set equal to zero, and, if  $F$  is less than  $\text{FMNGR}$ , the  $\text{GROW2}$  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,  $\text{ENQ}$ , is assumed to be a constant and the same at all values of  $F$  but more complex energy release laws could be implemented.

**\*EOS\_TENSOR\_PORE\_COLLAPSE**

This is Equation of state Form 11.

**Card Format**

Card 1            1            2            3            4            5            6            7            8

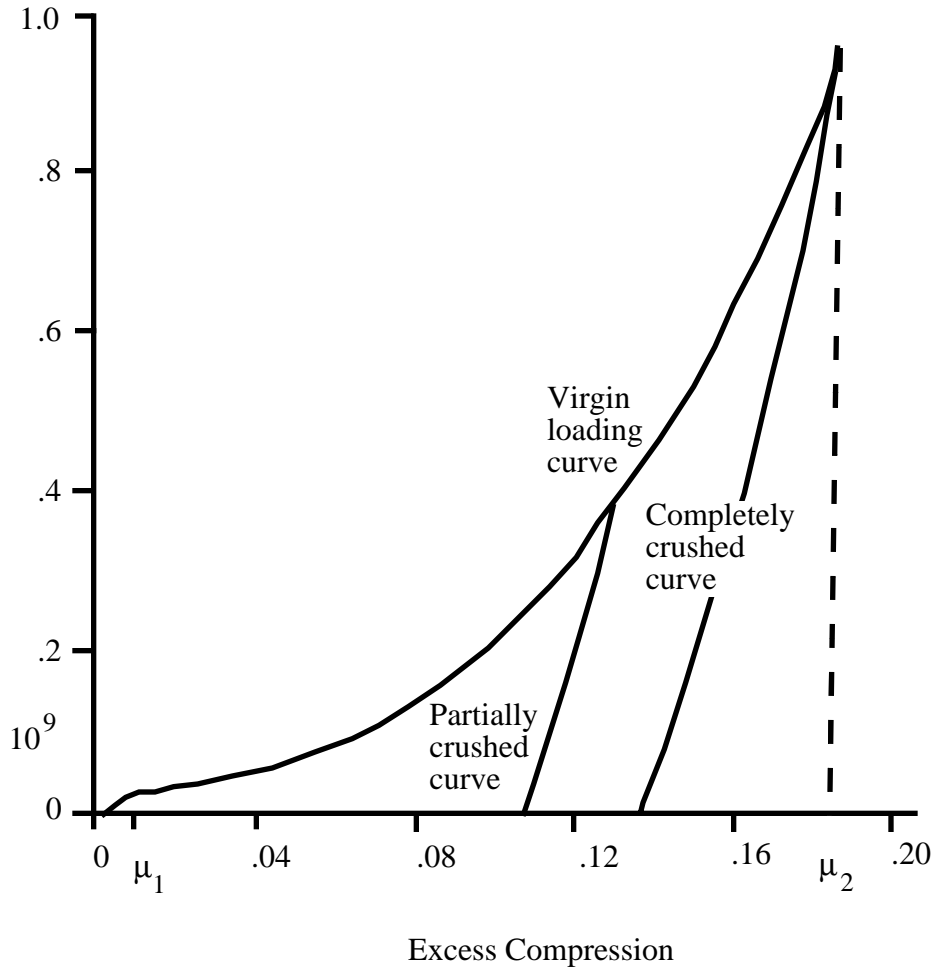
Variable	EOSID	NLD	NCR	MU1	MU2	IE0	EC0	
Type	I	F	F	F	F	F	F	

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

<u>VARIABLE</u>	<u>DESCRIPTION</u>
EOSID	Equation of state label
NLD	Virgin loading load curve ID
NCR	Completely crushed load curve ID
MU1	Excess Compression required before any pores can collapse
MU2	Excess Compression point where the Virgin Loading Curve and the Completely Crushed Curve intersect
IE0	Initial Internal Energy
EC0	Initial Excess Compression

**Remarks:**

The pore collapse model described in the TENSOR manual [23] is no longer valid and has been replaced by a much simpler method. This is due in part to the lack of experimental data required for the more complex model. It is desired to have a close approximation of the TENSOR model in the DYNA code to enable a quality link between them. The TENSOR model defines two curves, the virgin loading curve and the completely crushed curve as shown in Figure 13.2. It also defines the excess compression point required for pore collapse to begin ( $\mu_1$ ), and the excess compression point required to completely crush the material ( $\mu_2$ ). From this data and the maximum excess compression the material has attained ( $\mu_{max}$ ), the pressure for any excess compression ( $\mu$ ) can be determined.



**Figure 13.2.** Pressure versus compaction curve.

Unloading occurs along the virgin loading curve until the excess compression surpasses  $\mu_1$ . 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  $\mu_2$ , then all unloading will follow the completely crushed curve.

For unloading between  $\mu_1$  and  $\mu_2$  a partially crushed curve is determined by the relationship:

$$P_{pc}(\mu) = P_{cc} \left( \frac{(1 + \mu_B)(1 + \mu)}{1 + \mu_{\max}} - 1 \right).$$

where

$$\mu_B = P_{cc}^{-1}(P_{\max})$$

and the subscripts  $pc$  and  $cc$  refer to the partially crushed and completely crushed states, respectively. This is more readily understood in terms of the relative volume ( $V$ ).

$$V = \frac{1}{1 + \mu}$$

$$P_{pc}(V) = P_{cc} \left( \frac{V_B}{V_{\min}} V \right)$$

This representation suggests that for a fixed  $V_{\min} \left( = \frac{1}{\mu_{\max} + 1} \right)$  the partially crushed curve will separate linearly from the completely crushed curve as  $V$  increases to account for pore recovery in the material.

The bulk modulus  $K$  is determined to be the slope of the current curve times one plus the excess compression:

$$K = \frac{\partial P}{\partial \mu} (1 + \mu)$$

The slope  $\frac{\partial P}{\partial \mu}$  for the partially crushed curve is obtained by differentiation as:

$$\frac{\partial P}{\partial \mu} = \frac{\partial P_{cc} \left( \frac{(1 + \mu_B)(1 + \mu)}{1 + \mu_{\max}} \right) (1 + \mu_B)}{\partial \mu (1 + \mu_{\max})}$$

Simplifying,

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

where

$$\mu_a = \frac{(1 + \mu_B)(1 + \mu)}{1 + \mu_{\max}} - 1.$$

The bulk sound speed is determined from the slope of the completely crushed curve at the current pressure to avoid instabilities in the time step.

The virgin loading and completely crushed curves are modeled with monotonic cubic-splines. An optimized vector interpolation scheme is then used to evaluate the cubic-splines. The bulk modulus and sound speed are derived from a linear interpolation on the derivatives of the cubic-splines.

# \*EOS

---

## \*EOS\_JWL

This is Equation of state Form 14. The JWL (Jones-Wilkens-Lee-Baker) equation of state, developed by Baker [1991] and further described by Baker and Orosz [1991], describes the high pressure regime produced by overdriven detonations while retaining the low pressure expansion behavior required for standard acceleration modeling. The derived form of the equation of state is based on the JWL form due to its computational robustness and asymptotic approach to an ideal gas at high expansions. Additional exponential terms and a variable Gruneisen parameter have been added to adequately describe the high pressure region above the Chapman-Jouguet state.

### Card Format

Card 1            1            2            3            4            5            6            7            8

Variable	EOSID	A1	A2	A3	A4	A5		
Type	I	F	F	F	F	F		

Card 2            1            2            3            4            5            6            7            8

Variable	R1	R2	R3	R4	R5			
Type	F	F	F	F	F			

Card 3

Variable	AL1	AL2	AL3	AL4	AL5			
Type	F	F	F	F	F			

Card 4            2            3            4            5            6

Variable	BL1	BL2	BL3	BL4	BL5			
Type	F	F	F	F	F			

Card 5            1            2            3            4            5            6            7            8

Variable	RL1	RL2	RL3	RL4	RL5			
Type	F	F	F	F	F			

Card 6

Variable	C	OMEGA	E	V0				
Type	I	F	F	F				

**VARIABLE**

**DESCRIPTION**

EOSID	Equation of state label
A1	Equation of state coefficient, see below.
A2	Equation of state coefficient, see below.
A3	Equation of state coefficient, see below.
A4	Equation of state coefficient, see below.
A5	Equation of state coefficient, see below.
R1	Equation of state coefficient, see below.
R2	Equation of state coefficient, see below.
R3	Equation of state coefficient, see below.
R4	Equation of state coefficient, see below.
R5	Equation of state coefficient, see below.
AL1	$A\lambda_1$ , equation of state coefficient, see below.
AL2	$A\lambda_2$ , equation of state coefficient, see below.
AL3	$A\lambda_3$ , equation of state coefficient, see below.
AL4	$A\lambda_4$ , equation of state coefficient, see below.

# \*EOS

---

VARIABLE	DESCRIPTION
AL5	$A_{\lambda 5}$ , equation of state coefficient, see below.
BL1	$B_{\lambda 1}$ , equation of state coefficient, see below.
BL2	$B_{\lambda 2}$ , equation of state coefficient, see below.
BL3	$B_{\lambda 3}$ , equation of state coefficient, see below.
BL4	$B_{\lambda 4}$ , equation of state coefficient, see below.
BL5	$B_{\lambda 5}$ , equation of state coefficient, see below.
RL1	$R_{\lambda 1}$ , equation of state coefficient, see below.
RL2	$R_{\lambda 2}$ , equation of state coefficient, see below.
RL3	$R_{\lambda 3}$ , equation of state coefficient, see below.
RL4	$R_{\lambda 4}$ , equation of state coefficient, see below.
RL5	$R_{\lambda 5}$ , equation of state coefficient, see below.
C	Equation of state coefficient, see below.
OMEGA	Equation of state coefficient, see below.
E	Energy density per unit initial volume
V0	Initial relative volume.

## **Remarks:**

The JWLB equation-of-state defines the pressure as

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

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

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  $\omega$  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
$\rho_0$ (g/cc)	1.800	1.821	1.765	1.631	1.803
E0 (Mbar)	.07040	.10205	.10910	.06656	.09590
DCJ (cm/ $\mu$ s)	.76794	.86619	.83041	.67174	.82994
PCJ (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
R1	16.688	34.636	44.169	40.713	52.106
R2	6.8050	8.2176	8.7877	9.6754	8.3998
R3	2.0737	20.401	25.072	2.4350	2.1339
R4	2.9754	2.0616	2.2251	.15564	.18592
C (Mbar)	.00776	.01251	.01570	.00710	.00968
$\omega$	.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



# **\*HOURLASS**

## **\*HOURLASS**

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

### **Card Format**

Card 1            1            2            3            4            5            6            7            8

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

---

**VARIABLE**

---

**DESCRIPTION**

HGID	Hourglass ID. Unique numbers have to be specified.
IHQ	Hourglass control type. For solid elements six options are available. For quadrilateral shell and membrane elements the hourglass control is based on the formulation of Belytschko and Tsay, i.e., options 1-3 are identical, and options 4-6 are identical: EQ.0:default=1, EQ.1:standard LS-DYNA viscous form, EQ.2:Flanagan-Belytschko viscous form, EQ.3: Flanagan-Belytschko viscous form with exact volume integration for solid elements, EQ.4:Flanagan-Belytschko stiffness form, EQ.5:Flanagan-Belytschko stiffness form with exact volume integration for solid elements. EQ.6:Belytschko-Bindeman [1993] assumed strain co-rotational stiffness form for 2D and 3D solid elements only. This form is available for explicit and IMPLICIT solution methods. In fact, type 6 is mandatory for the implicit options.

# \*HOURGLASS

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
	<p>EQ:8:Applicable to the type 16 fully integrated shell element. IHQ=8 activates the warping stiffness for accurate solutions. A speed penalty of 25% is common for this option.</p> <p>A discussion of the viscous and stiffness hourglass control for shell elements follows at the end of this section.</p>
QM	Hourglass coefficient. Values of QM that exceed .15 (for IHQ<6) may cause instabilities. The recommended default applies to all options. The stiffness forms, however, can stiffen the response especially if deformations are large and therefore should be used with care. For the shell and membrane elements QM is taken as the membrane hourglass coefficient, the bending as QB, and warping as QW. These coefficients can be specified independently, but generally, QM=QB=QW, is adequate. For type 6 solid element hourglass control, QM=1.0 gives an accurate coarse mesh bending stiffness that does not lock in the incompressible limit. For type 6 values such as 0.001-0.01 will avoid an overly stiff response.
IBQ	Bulk viscosity type (See remark 3. below.): EQ.1: standard LS-DYNA.
Q2	Quadratic bulk viscosity coefficient.
Q1	Linear bulk viscosity coefficient.
QB	Hourglass coefficient for shell bending. The default: QB=QM. See remark 4. below.
QW	Hourglass coefficient for shell warping. The default: QB=QW.

## **Remarks:**

1. Viscous hourglass control is recommended for problems deforming with high velocities. Stiffness control is often preferable for lower velocities, especially if the number of time steps are large. For solid elements the exact volume integration provides some advantage for highly distorted elements.
2. For automotive crash the stiffness form of the hourglass control with a coefficient of 0.05 is preferred by many users.
3. Bulk viscosity is necessary to propagate shock waves in solid materials and therefore applies only to solid elements. Generally, the default values are okay except in problems where pressures are very high, larger values may be desirable. In low density foams, it may be necessary to reduce the viscosity values since the viscous stress can be significant. It is not advisable to reduce it by more than an order of magnitude.
4. In part, the computational efficiency of the Belytschko-Lin-Tsay and the under integrated Hughes-Liu shell elements are derived from their use of one-point quadrature in the plane of the element. To suppress the hourglass deformation modes that accompany one-point quadrature, hourglass viscous or stiffness based stresses are added to the physical stresses at the local

element level. The discussion of the hourglass control that follows pertains to all one point quadrilateral shell and membrane elements in LS-DYNA.

The hourglass shape vector  $\tau_I$  is defined as

$$\tau_I = h_I - (h_J \hat{x}_{aJ}) B_{aI}$$

where,  $\hat{x}_{aJ}$  are the element coordinates in the local system at the Ith element node,  $B_{aI}$  is the strain displacement matrix, and hourglass basis vector is:

$$h = \begin{bmatrix} +1 \\ -1 \\ +1 \\ -1 \end{bmatrix}$$

is the basis vector that generates the deformation mode that is neglected by one-point quadrature. In the above equations and the remainder of this subsection, the Greek subscripts have a range of 2, e.g.,  $\hat{x}_{aI} = (\hat{x}_{1I}, \hat{x}_{2I}) = (\hat{x}_I, \hat{y}_I)$ .

The hourglass shape vector then operates on the generalized displacements to produce the generalized hourglass strain rates

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

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

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

where the superscripts M, B, and W denote membrane, bending, and warping modes, respectively. The corresponding hourglass stress rates are then given by

$$\dot{Q}_\alpha^M = \frac{QM \cdot EtA}{8} B_{\beta I} B_{\beta I} \dot{q}_\alpha^M$$

$$\dot{Q}_\alpha^B = \frac{QB \cdot Et^3 A}{192} B_{\beta I} B_{\beta I} \dot{q}_\alpha^B$$

$$\dot{Q}_3^W = \frac{QW \cdot \kappa Gt^3 A}{12} B_{\beta I} B_{\beta I} \dot{q}_3^B$$

where t is the shell thickness. The hourglass coefficients: QM, QB, and QW are generally assigned values between 0.05 and 0.10.

Finally, the hourglass stresses which are updated using the time step,  $\Delta t$ , from the stress rates in the usual way, i.e.,

## \*HOURGLASS

---

$$Q^{n+1} = Q^n + \Delta t \dot{Q}$$

and the hourglass resultant forces are then

$$\hat{f}_{\alpha I}^H = \tau_I Q_\alpha^M$$

$$\hat{m}_{\alpha I}^H = \tau_I Q_\alpha^B$$

$$\hat{f}_{3I}^H = \tau_I Q_3^W$$

where the superscript H emphasizes that these are internal force contributions from the hourglass deformations.

# **\*INCLUDE**

## **\*INCLUDE**

Purpose: File to be included in this keyword file. The file contents are placed directly at the location of the \*INCLUDE line.

### **Card Format**

Card 1

1

Variable	FILENAME
Type	C

---

**VARIABLE**

---

**DESCRIPTION**

FILENAME

File name of file to be included in this keyword file, 80 characters maximum.

### **Remarks:**

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

```
*TITLE
full car model
*INCLUDE
carfront.k
*INCLUDE
carback.k
*INCLUDE
occupantcompartment.k
*INCLUDE
dummy.k
*INCLUDE
bag.k
*CONTACT
...
*END
```

# **\*INCLUDE**

---

Note that the command \*END terminates the include file.

The carfront.k file can again be subdivided into rightrail.k, leftrail.k, battery.k, wheelhouse.k, shotgun.k, etc.. Each \*.k file can include nodes, elements, boundary conditions, initial conditions, and so on.

```
*INCLUDE
rightrail.k
*INCLUDE
leftrail.k
*INCLUDE
battery.k
*INCLUDE
wheelhouse.k
*INCLUDE
shotgun.k
...
*END
```



# **\*INITIAL**

The keyword **\*INITIAL** provides a way of initializing velocities and detonation points. The keyword control cards in this section are defined in alphabetical order:

**\*INITIAL\_DETONATION**  
**\*INITIAL\_FOAM\_REFERENCE\_GEOMETRY**  
**\*INITIAL\_MOMENTUM**  
**\*INITIAL\_STRAIN\_SHELL**  
**\*INITIAL\_STRESS\_BEAM**  
**\*INITIAL\_STRESS\_SHELL**  
**\*INITIAL\_STRESS\_SOLID**  
**\*INITIAL\_TEMPERATURE\_OPTION**  
**\*INITIAL\_VEHICLE\_KINEMATICS**

Two mutually exclusive methods are available for initial velocity generation:

**\*INITIAL\_VELOCITY**  
**\*INITIAL\_VELOCITY\_NODE**

and:

**\*INITIAL\_VELOCITY\_GENERATION**

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

**\*INITIAL\_VOID\_OPTION**

# \*INITIAL

---

## \*INITIAL\_DETONATION

Purpose: Define points to initiate the location of high explosive detonations in part ID's which use the material (type 8) \*MAT\_HIGH\_EXPLOSIVE\_BURN. Also see \*CONTROL\_EXPLOSIVE\_SHADOW.

### Card Format

Card 1            1            2            3            4            5            6            7            8

Variable	PID	X	Y	Z	LT			
Type	I	F	F	F	F			
Default	all HE	0.	0.	0.	0,			

### Optional card required if and only if PID=-1.

Card 2            1            2            3            4            5            6            7            8

Variable	PEAK	DECAY	XS	YS	ZS	NID		
Type	F	F	F	F	F	I		
Remark	1	1						

VARIABLE	DESCRIPTION
PID	Part ID of high explosive material to be lit, see *PART. However, two other options are available: EQ.-1: an acoustic boundary, also *BOUNDARY_USA_SURFACE, EQ. 0: all high explosive materials are considered.
X	x-coordinate of detonation point, see Figure 16.1.
Y	y-coordinate of detonation point.
Z	z-coordinate of detonation point.

---

<b>VARIABLE</b>	<b>DESCRIPTION</b>
LT	Lighting time for detonation point. This time is ignored for an acoustic boundary.
PEAK	Peak pressure, $p_o$ , of incident pressure pulse, see remark below.
DECAY	Decay constant, $\tau$
XS	x-coordinate of standoff point, see Figure 16.1.
YS	y-coordinate of standoff point
ZS	z-coordinate of standoff point
NID	Reference node ID near structure

**Remarks:**

For solid elements (not acoustic) two options are available. If the control card option, \*CONTROL\_EXPLOSIVE\_SHADOW, is not used the lighting time for an explosive element is computed using the distance from the center of the element to the nearest detonation point,  $L_d$ ; the detonation velocity,  $D$ ; and the lighting time for the detonator,  $t_d$ :

$$t_L = t_d + \frac{L_d}{D}$$

The detonation velocity for this default option is taken from the element whose lighting time is computed and does not account for the possibilities that the detonation wave may travel through other explosives with different detonation velocities or that the line of sight may pass outside of the explosive material.

If the control card option, \*CONTROL\_EXPLOSIVE\_SHADOW, is defined, the lighting time is based on the shortest distance through the explosive material. If inert obstacles exist within the explosive material, the lighting time will account for the extra time required for the detonation wave to travel around the obstacles. The lighting times also automatically accounts for variations in the detonation velocity if different explosives are used. No additional input is required for this option but care must be taken when setting up the input. This option works for two and three dimensional solid elements. It is recommended that for best results:

1. Keep the explosive mesh as uniform as possible with elements of roughly the same dimensions.
2. Inert obstacle such as wave shapers within the explosive must be somewhat larger than the characteristic element dimension for the automatic tracking to function properly. Generally, a factor of two should suffice. The characteristic element dimension is found by checking all explosive elements for the largest diagonal
3. The deonation points should be either within or on the boundary of the explosive. Offset points may fail to initiate the explosive.

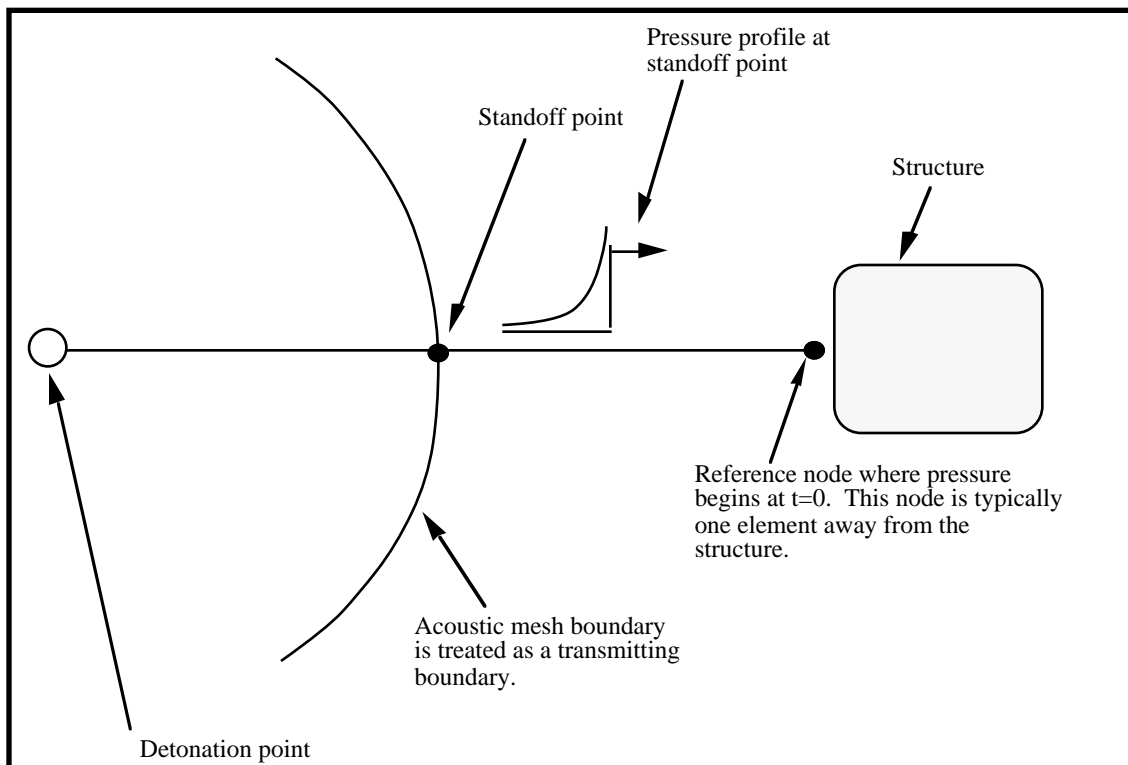
## \*INITIAL

4. Check the computed lighting times in the post processor LS-POST. The lighting times may be displayed at time=0., state 1, by plotting component 7 (a component normally reserved for plastic strain) for the explosive material. The lighting times are stored as negative numbers. The negative lighting time is replaced by the burn fraction when the element ignites.

Line detonations may be approximated by using a sufficient number of detonation points to define the line. Two many detonation points may result in significant initialization cost.

The pressure versus time curve for the acoustic option is defined by:

$$p(t) = p_0 e^{-\frac{t}{\tau}}$$



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

**\*INITIAL\_FOAM\_REFERENCE\_GEOMETRY**

Purpose: The reference configuration allows stresses to be initialized in the material model, \*MAT\_LOW\_DENSITY\_FOAM To use this option, the geometry of the foam material is defined in a deformed configuration. The stresses in the low density foam then depend only on the deformation gradient matrix:

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

where  $X_j$  is the undeformed configuration. By using this option, dynamic relaxation can be avoided once a deformed configuration is obtained usually on the first run of a particular problem.

**Card Format (I8,3E16.0)**

Card 2,...            1            2            3            4            5            6            7            8            9            10

Variable	NID	X	Y	Z						
Type	I	F	F	F						
Default	none	0.	0.	0.						
Remarks										

**VARIABLE****DESCRIPTION**

BIRTH	Time at which the reference geometry activates (default=0.0)
NID	Node number
X	x coordinate
Y	y coordinate
Z	z coordinate

# \*INITIAL

---

## \*INITIAL\_MOMENTUM

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

### Card Format

	1	2	3	4	5	6	7	8
Variable	EID	MX	MY	MZ	DEPT			
Type	I	F	F	F	F			
Default	none	0.	0.	0.	0,			

### VARIABLE

### DESCRIPTION

EID	Element ID
MX	Initial x-momentum
MY	Initial y-momentum
MZ	Initial z-momentum
DEPT	Deposition time

**\*INITIAL\_STRAIN\_SHELL**

Purpose: Initialize strain tensor and inner and outer through thickness integration points at element center. This option is primarily for multi-stage metal forming operations where the accumulated strain is of interest.

Define as many shell elements in this section as desired. The input is assumed to terminate when a new keyword is detected. These strain tensors are defined at the inner and outer integration points and are used for post-processing only. There is no interpolation with this option and the strains are defined in the global cartesian coordinate system. The \*DATABASE\_EXTENT\_BINARY flag STRFLG must be set to unity for this option to work.

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	EID							
Type	I							
Default	none							

**Define two cards below, one for the inner integration point and the other for the outer integration point, respectively.**

Card 2,...        1            2            3            4            5            6            7            8

Variable	EPSxx	EPSyy	EPSzz	EPSxy	EPSyz	EPSzx		
Type	F	F	F	F	F	F		

---

**VARIABLE**

---

**DESCRIPTION**

EID

Element ID

EPSij

Define the ij strain component. The strains are defined in the GLOBAL cartesian system.

# \*INITIAL

---

## \*INITIAL\_STRESS\_BEAM

Purpose: Initialize stresses and plastic strains in the Hughes-Liu beam elements.

Define as many beams in this section as desired. The input is assumed to terminate when a new keyword is detected.

### Card Format

Card 1            1            2            3            4            5            6            7            8

Variable	EID	RULE	NPTS					
Type	I	I	I					
Default	none	none	none					

Define NTPS cards below, one per integration point.

Card 2,...        1            2            3            4            5            6            7            8

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

---

### VARIABLE

### DESCRIPTION

---

EID

Element ID

RULE

Integration rule type number:

- EQ.1.0: truss element or discrete beam element,
- EQ.2.0: 2 × 2 Gauss quadrature (default beam),
- EQ.3.0: 3 × 3 Gauss quadrature,
- EQ.4.0: 3 × 3 Lobatto quadrature,
- EQ.5.0: 4 × 4 Gauss quadrature.



<b><u>VARIABLE</u></b>	<b><u>DESCRIPTION</u></b>
NPTS	Number of integration points output.
SIGIJ	Define the IJ stress component.
EPS	Effective plastic strain

# \*INITIAL

---

## \*INITIAL\_STRESS\_SHELL

Purpose: Initialize stresses and plastic strains for shell elements.

Define as many shell elements in this section as desired. The input is assumed to terminate when a new keyword is detected. It is not necessary for the location of the through thickness integration points to match those used in the elements which are initialized. The data will be interpolated by LS-DYNA3D.

### Card Format

Card 1            1            2            3            4            5            6            7            8

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

Define **NPLANE X NTHICK** cards below, one per integration point. For each through thickness point define **NPLANE** points. **NPLANE** should be either 1 or 4 corresponding to either 1 or 4 Gauss integration points. If four integration points are specified, they should be ordered such that their in plane parametric coordinates are at:

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

respectively.

Card 2,...        1            2            3            4            5            6            7            8

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

---

<b>VARIABLE</b>	<b>DESCRIPTION</b>
EID	Element ID
NPLANE	Number of in plane integration points being output.
NTHICK	Number of through thickness integration points.
T	Parametric coordinate of through thickness integration point. between -1 and 1 inclusive.
SIGij	Define the ij stress component. The stresses are defined in the GLOBAL cartesian system.
EPS	Effective plastic strain

# \*INITIAL

---

## \*INITIAL\_STRESS\_SOLID

Purpose: Initialize stresses and plastic strains for solid elements.

Define as many solid elements in this section as desired. The input is assumed to terminate when a new keyword is detected. If eight points are defined for 1 point LS-DYNA solid elements, the average value will be taken.

### Card Format

Card 1            1            2            3            4            5            6            7            8

Variable	EID	NINT						
Type	I	I						
Default	none	none						

Define NINT cards below, one per integration point. NINT should be either 1 or 8. If eight Gauss integration points are specified, they should be ordered such that their parametric coordinates are located at:

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

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

respectively.

Card 2,...            1            2            3            4            5            6            7            8

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

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

# \*INITIAL

---

## \*INITIAL\_TEMPERATURE\_OPTION

Available options are:

### NODE SET

Purpose: Define initial nodal point temperatures using nodal set ID's or node numbers. These initial temperatures are used in a thermal only analysis or a coupled thermal/structural analysis. See also \*CONTROL\_THERMAL\_SOLVER, \*CONTROL\_THERMAL\_TIMESTEP, and CONTROL\_THERMAL\_NONLINEAR.

For thermal loading in a structural only analysis, see \*LOAD\_THERMAL\_option.

### Card Format

Card 1            1            2            3            4            5            6            7            8

Variable	NSID/NID	TEMP							
Type	I	I							
Default	none	0.							
Remark	1								

---

#### VARIABLE

#### DESCRIPTION

NSID/NID

Nodal set ID or nodal point ID, see also \*SET\_NODES:  
EQ.0: all nodes are included (set option only).

TEMP

Temperature at node or node set.

### Remark:

1. If a nodal temperature is specified on more than one input card, then the last set input will determine its temperature unless it is specified on a \*INITIAL\_TEMPERATURE\_NODE card.

**\*INITIAL\_VEHICLE\_KINEMATICS**

Purpose: Define initial kinematical information for a vehicle. In its initial orientation, the vehicle's yaw, pitch, and roll axes must be aligned with the global axes. Successive simple rotations are taken about these body fixed axes.

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	GRAV	PSID	XO	YO	ZO	XF	YF	ZF
Type	I	I	F	F	F	F	F	F
Default	none	none	0.	0.	0.	0.	0.	0.

Card 2            1            2            3            4            5            6            7            8

Variable	VX	VY	VZ	AAXIS	BAXIS	CAXIS		
Type	F	F	F	I	I	I		
Default	0.	0.	0.	0	0	0		

Card 3            1            2            3            4            5            6            7            8

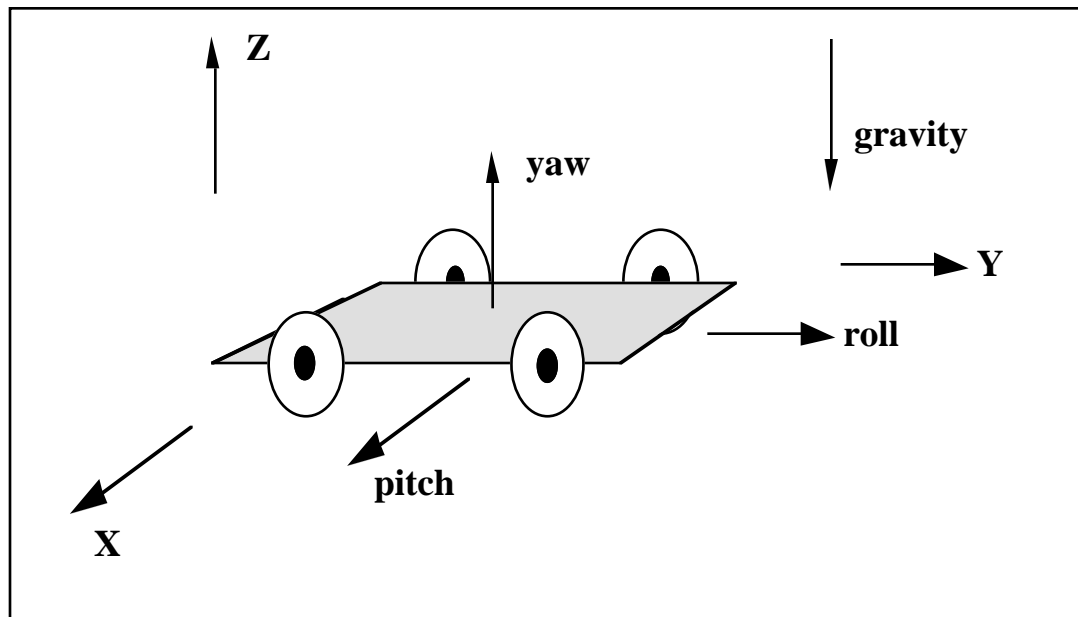
Variable	AANG	BANG	CANG	WA	WB	WC		
Type	F	F	F	F	F	F		
Default	0.	0.	0.	0.	0.	0.		

# \*INITIAL

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
GRAV	Gravity direction code. EQ.1. Global +x direction. EQ.-1. Global -x direction. EQ.2. Global +y direction. EQ.-2. Global -y direction. EQ.3 Global +z direction. EQ.-3 Global -z direction. Note: this must be the same for all vehicles present in the model.
PSID	Part set ID.
XO	x-coordinate of initial position of mass center.
YO	y-coordinate of initial position of mass center.
ZO	z-coordinate of initial position of mass center.
XF	x-coordinate of final position of mass center.
YF	y-coordinate of final position of mass center.
ZF	z-coordinate of final position of mass center.
VX	x-component of mass center velocity.
VY	y-component of mass center velocity.
VZ	z-component of mass center velocity.
AAXIS	First rotation axis code. EQ.1. Initially aligned with global x-axis. EQ.2. Initially aligned with global y-axis. EQ.3. Initially aligned with global z-axis.
BAXIS	Second rotation axis code.
CAXIS	Third rotation axis code.
AANG	Rotation angle about the first rotation axis (degrees).
BANG	Rotation angle about the second rotation axis (degrees).
CANG	Rotation angle about the third rotation axis (degrees).
WA	Angular velocity component for the first axis (radian/second).
WB	Angular velocity component for the second axis (radian/second).
WC	Angular velocity component for the third axis (radian/second).





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

# \*INITIAL

---

## \*INITIAL\_VELOCITY

Purpose: Define initial nodal point translational velocities using nodal set ID's. This may also be used for sets in which some nodes have other velocities. See NSIDEX below.

### Card Format

Card 1            1            2            3            4            5            6            7            8

Variable	NSID	NSIDEX	BOXID					
Type	I	I	I					
Default	none	0.	0.					
Remark	1							

Card 2            1            2            3            4            5            6            7            8

Variable	VX	VY	VZ	VXR	VYR	VZR		
Type	F	F	F	F	F	F		
Default	0.	0.	0.	0.	0.	0.		

**Define the following card if and only if NSIDEX>0.**

                  1            2            3            4            5            6            7            8

Variable	VXE	VYE	VZE	VXRE	VYRE	VZRE		
Type	F	F	F	F	F	F		
Default	0.	0.	0.	0.	0.	0.		

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
NSID	Nodal set ID, see *SET_NODES, containing nodes for initial velocity: EQ.0: all nodes are included.
NSIDEX	Nodal set I, see *SET_NODES, containing nodes that are exempted from the imposed velocities and may have other initial velocities.
BOXID	All nodes in box which belong to NSID are initialized. Nodes outside the box are not initialized. Exempted nodes are initialized to velocities defined by VXE, VYE, and VZE below regardless of their location relative to the box.
VX	Initial velocity in x-direction
VY	Initial velocity in y-direction
VZ	Initial velocity in z-direction
VXR	Initial rotational velocity about the x-axis
VYR	Initial rotational velocity about the y-axis
VZR	Initial rotational velocity about the z-axis
VXE	Initial velocity in x-direction of exempted nodes
VYE	Initial velocity in y-direction of exempted nodes
VZE	Initial velocity in z-direction of exempted nodes
VXRE	Initial rotational velocity in x-direction of exempted nodes
VYRE	Initial rotational velocity in y-direction of exempted nodes
VZRE	Initial rotational velocity in z-direction of exempted nodes

**Remarks:**

1. This generation input must not be used with \*INITIAL\_VELOCITY\_GENERATION keyword.
2. If a node is initialized on more than one input card set, then the last set input will determine its velocity. However, if the nodal velocity is also specified on a \*INITIAL\_VELOCITY\_NODE card, then the velocity specification on this card will be used.

# \*INITIAL

---

## \*INITIAL\_VELOCITY\_NODE

Purpose: Define initial nodal point velocities for a node.

### Card Format

	1	2	3	4	5	6	7	8
Variable	NID	VX	VY	VZ	VXR	VYR	VZR	
Type	I	F	F	F	F	F	F	
Default	none	0.	0.	0.	0.	0.	0.	

<u>VARIABLE</u>	<u>DESCRIPTION</u>
NID	Node ID
VX	Initial translational velocity in x-direction
VY	Initial translational velocity in y-direction
VZ	Initial translational velocity in z-direction
VXR	Initial rotational velocity about the x-axis
VYR	Initial rotational velocity about the y-axis
VZR	Initial rotational velocity about the z-axis

See remark on \*INITIAL\_VELOCITY card.

**\*INITIAL\_VELOCITY\_GENERATION**

Purpose: Define initial velocities for rotating and translating bodies.

**Card Format**

Card 1            1            2            3            4            5            6            7            8

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

Card 2            1            2            3            4            5            6            7            8

Variable	XC	YC	ZC	NX	NY	NZ	PHASE	
Type	F	F	F	F	F	F	I	
Default	0.	0.	0.	0.	0.	0.	0	

**VARIABLE****DESCRIPTION**

ID	Part ID, part set ID, or node set ID. If zero STYP is ignored and all velocities are set.
STYP	Set type: EQ.1: part set ID, see *SET_PART, EQ.2: part ID, see *PART, EQ.3: node set ID, see *SET_NODE.
OMEGA	Angular velocity about rotational axis.
VX	Initial translational velocity in global x-direction.
VY	Initial translational velocity in global y-direction.
VZ	Initial translational velocity in global z-direction. .

# \*INITIAL

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
XC	x-coordinate on rotational axis.
YC	y-coordinate on rotational axis.
ZC	z-coordinate on rotational axis.
NX	x-direction cosine.
NY	y-direction cosine.
NZ	z-direction cosine.
PHASE	Flag specifying phase of the analysis the velocities apply to: EQ.0. Velocities applied immediately, EQ.1. Velocities applied after dynamic relaxation.

## **Remarks:**

1. This generation input must not be used with \*INITIAL\_VELOCITY or \*INITIAL\_VELOCITY\_NODE options.
2. The velocities are initialized in the order the \*INITIAL\_VELOCITY\_GENERATION input is defined. Later input via the \*INITIAL\_VELOCITY\_GENERATION keyword may overwrite the velocities previously set.
3. For rigid bodies, initial velocities given in \*PART\_INERTIA will overwrite generated initial velocities.

**\*INITIAL\_VOID\_OPTION**

Available options are:

**PART  
SET**

Purpose: Define initial voided part set ID's or part numbers. Void materials cannot be created during the calculation. Fluid elements which are evacuated, e.g., by a projectile moving through the fluid, during the calculation are approximated as fluid elements with very low densities. The constitutive properties of fluid materials used as voids must be identical to those of the materials which will fill the voided elements during the calculation. Mixing of two fluids with different properties is not permitted with this option.

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	PSID/PID							
Type	I							
Default	none							
Remark	1							

**VARIABLE**

**DESCRIPTION**

PSID/PID

Part set ID or part ID, see also \*SET\_PART:

**Remark:**

This void option and multiple materials per element, see \*ALE\_MULTI-MATERIAL\_GROUP are incompatible and cannot be used together in the same run.

**\*INITIAL**

---



# **\*INTEGRATION**

## **\*INTEGRATION\_BEAM**

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

### **Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	IRID	NIP	RA	ICST				
Type	I	I	F	I				
Default	none	0	0.0	0				

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

                  1            2            3            4            5            6            7            8

Variable	W	TF	D	TW	SREF	TREF		
Type	F	F	F	F	F	F		
Default	none	none	none	none	1.0	0.0		

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

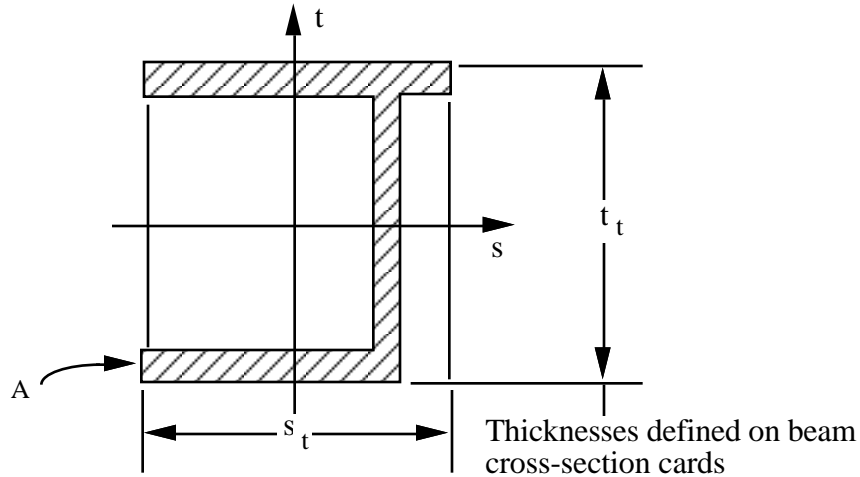
                  1            2            3            4            5            6            7            8

Variable	S	T	WF					
Type	F	F	F					

# \*INTEGRATION

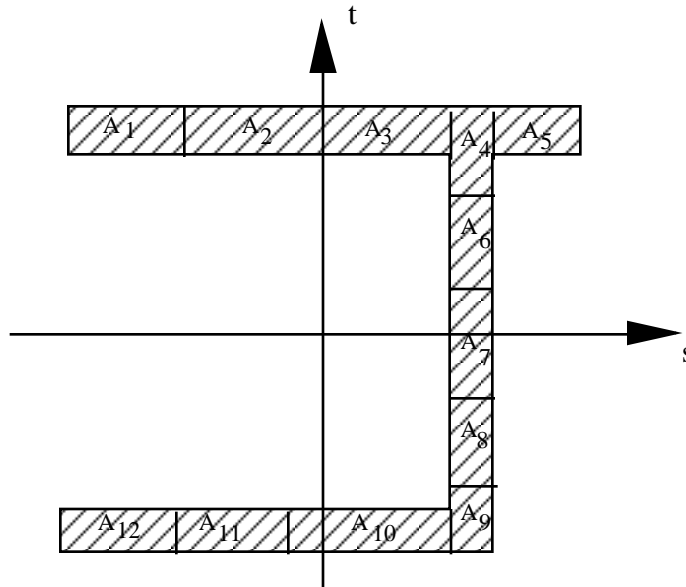
---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
IRID	Integration rule ID. IRID refers to IRID on *SECTION_BEAM card.
NIP	Number of integration points, see also ICST.
RA	Relative area of cross section, i.e., the actual cross-sectional area divided by the area defined by the product of the specified thickness in the s direction and the thickness in the t direction. See also ICST below and Figure 17.1.
ICST	Standard cross section type, ICST. If this type is nonzero then NIP and the relative area above should be input as zero. See the discussion following the input description Figures 17.3a and 17.3b. EQ.1: W-section, EQ.2: C-section, EQ.3: Angle section, EQ.4: T-section, EQ.5: Rectangular tubing, EQ.6: Z-section,. EQ.7: Trapezoidal section
W	w, flange width
TF	t <sub>f</sub> , flange thickness
D	d, depth
TW	t <sub>w</sub> , web thickness
SREF	s <sub>ref</sub> , location of reference surface normal to s, for the Hughes-Liu beam only. This option is only useful if the beam is connected to a shell or another beam on its outer surface, see also *SECTION_BEAM.
TREF	t <sub>ref</sub> , location of reference surface normal to t, for the Hughes-Liu beam only. This option is only useful if the beam is connected to a shell or another beam on its outer surface, see also *SECTION_BEAM.
S	Normalized s coordinate of integration point, $-1 \leq s \leq 1$ .
T	Normalized t coordinate of integration point, $-1 \leq t \leq 1$ .
WF	Weighting factor, $A_{ri}$ , i.e., the area associated with the integration point divided by actual cross sectional area $A_{ri} = A_i/A$ , see Figure 17.2.



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

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



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

**Remarks:**

The input for standard beam section types is defined below. In Figures 17.3a and 17.3b, the dimensions are shown on the left and the location of the integration points are shown on the right. If a quantity is not defined in the sketch, then it should be set to zero in the input. The input quantities include:

# \*INTEGRATION

$w$  = flange width

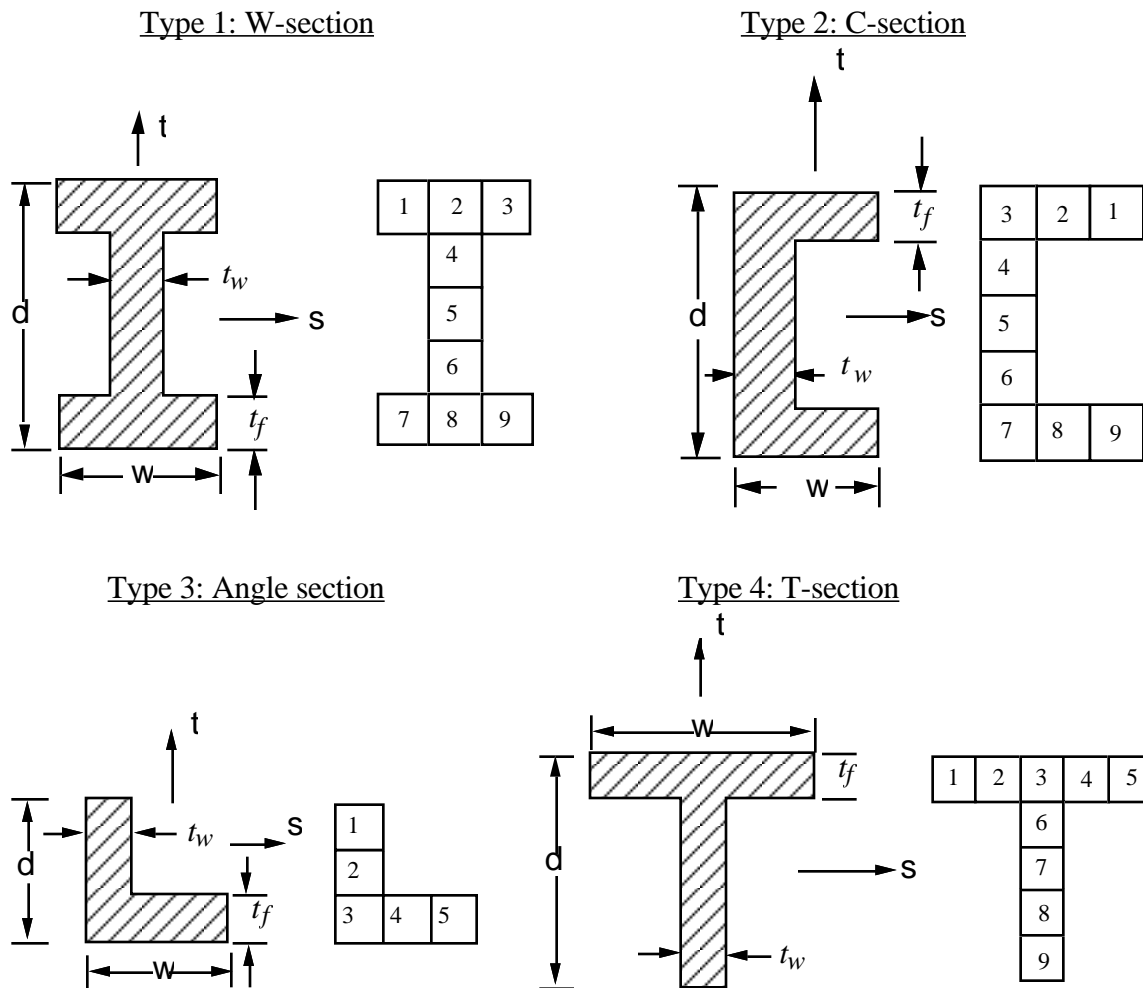
$t_f$  = flange thickness

$d$  = depth

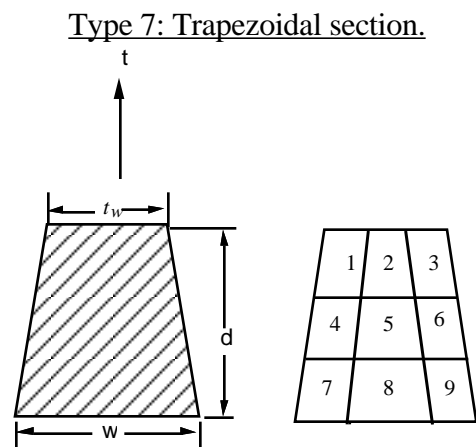
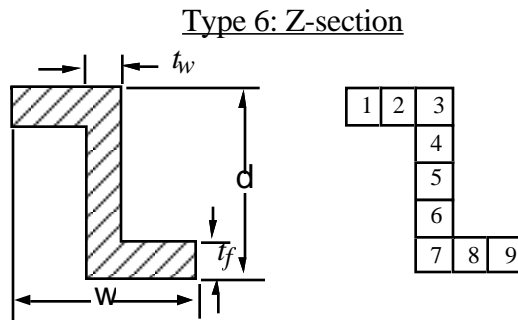
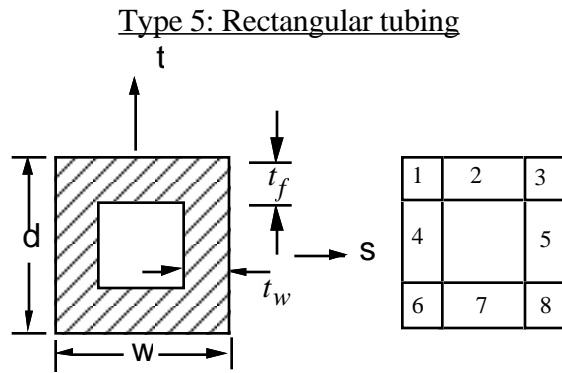
$t_w$  = web thickness

$s_{ref}$  = location of reference surface normal to  $s$ , Hughes-Liu beam only

$t_{ref}$  = location of reference surface normal to  $t$ , Hughes-Liu beam only



**Figure 17.3a.** Standard beam cross sections.



**Figure 17.3b.** Standard beam cross sections.

# \*INTEGRATION

---

## \*INTEGRATION\_SHELL

Purpose: Define user defined through the thickness integration rules for the shell element. This option applies to three dimensional shell elements with three or four nodes (ELEMENT\_SHELL types 1-11 and 16) and to the eight nodel thick shell (ELEMENT\_TSHELL).

### Card Format

Card 1            1            2            3            4            5            6            7            8

Variable	IRID	NIP	ESOP					
Type	I	I	I					

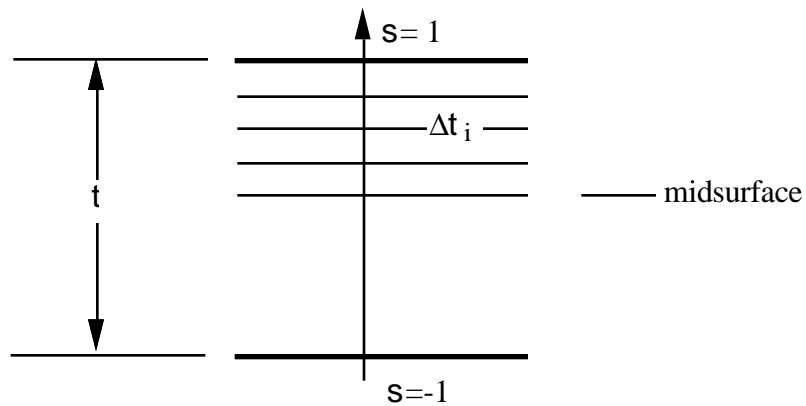
Define NIP cards below if ESOP = 0.

1            2            3            4            5            6            7            8

Variable	S	WF	PID					
Type	F	F	I					

<u>VARIABLE</u>	<u>DESCRIPTION</u>
IRID	Integration rule ID (IRID refers to IRID on *SECTION_SHELL card).
NIP	Number of integration points
ESOP	Equal spacing of integration points option: EQ.0: integration points are defined below, EQ.1: integration points are equally spaced through thickness such that the shell is subdivided into NIP layers of equal thickness.
S	Coordinate of integration point in range -1 to 1.
WF	Weighting factor. This is typically the thickness associated with the integration point divided by actual shell thickness, i.e., the weighting factor for the $i$ th integration point = $\frac{\Delta t_i}{t}$ as seen in Figure 17.4.

VARIABLE	DESCRIPTION
PID	Optional part ID if different from the ID specified on the element card. The material type is not allowed to change, see *PART. THE DENSITY FOR THE ELEMENT IS ALWAYS TAKEN FROM THE DATA FOR THE PART ID SPECIFIED ON THE ELEMENT CARD.



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

# **\*INTEGRATION**

---



# **\*INTERFACE**

## **\*INTERFACE\_COMPONENT\_OPTION**

Options include:

- NODE**
- SEGMENT**

Purpose: Define an interface for linking calculations. This card applies to the first analysis for storing interfaces in the file specified by Z=isf1 on the execution command line. The output interval used to write data to the interface file is controlled by OPIFS on \*CONTROL\_OUTPUT.

This capability allows the definition of interfaces that isolate critical components. A database is created that records the motion of the interfaces. In later calculations the isolated components can be reanalyzed with arbitrarily refined meshes with the motion of their boundaries specified by the database created by this input. The interfaces defined here become the masters in the tied interface options.

Each definition consists of a set of cards that define the interface. Interfaces may consist of a set of four node segments for moving interfaces of solid elements, a line of nodes for treating interfaces of shells, or a single node for treating beam and spring elements.

### **Card Format**

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

<u>VARIABLE</u>	<u>DESCRIPTION</u>
SID	Set ID, see *SET_NODE or *SET_SEGMENT.

# **\*INTERFACE**

---

## **\*INTERFACE\_LINKING\_DISCRETE\_NODE\_OPTION**

Options include:

**NODE**  
**SET**

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

### **Card Format**

	1	2	3	4	5	6	7	8
Variable	NID/NSID	IFID						
Type	I	I						

<u>VARIABLE</u>	<u>DESCRIPTION</u>
NID	Node ID or Node set ID to be moved by interface file, see *NODE or *SET_NODE.
IFID	Interface ID in interface file.

# \*INTERFACE

---

## \*INTERFACE\_LINKING\_SEGMENT

Purpose: Define an interface for linking segments to an interface file for the second analysis using L=isf2 on the execution command line. This applies segments on shell and solid elements.

### Card Format

	1	2	3	4	5	6	7	8
Variable	SSID	IFID						
Type	I	I						

---

#### VARIABLE

#### DESCRIPTION

SSID	Segment set to be moved by interface file.
IFID	Interface ID in interface file.

# \*INTERFACE

---

## \*INTERFACE\_LINKING\_EDGE

Purpose: Define an interface for linking a series of nodes in shell structure to an interface file for the second analysis using L=isf2 on the execution command line. This link applies segments on shell elements only.

### Card Format

1            2            3            4            5            6            7            8

Variable	NSID	IFID						
Type	I	I						

---

#### VARIABLE

#### DESCRIPTION

---

NSID            Node set ID to be moved by interface file.

IFID            Interface ID in interface file.

# **\*INTERFACE**

---

## **\*INTERFACE\_JOY**

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

### **Card Format**

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

---

#### **VARIABLE**

#### **DESCRIPTION**

---

SID

Nodal set ID, see \*SET\_NODE.

# \*INTERFACE

---

## \*INTERFACE\_SPRINGBACK\_OPTION1\_OPTION2

Options included for *OPTION1* are:

**NIKE3D**  
**DYNA3D**  
**NASTRAN**  
**SEAMLESS**

and for *OPTION2*:

**THICKNESS**  
**NOTHICKNESS**

See the remarks below.

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

### Card Format

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

<u>VARIABLE</u>	<u>DESCRIPTION</u>
PSID	Part set ID for springback, see *SET_PART.

# \*INTERFACE

Define a list of nodal points that are constrained for the springback. This section is terminated by an “\*” indicating the next input section.

## Card Format

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

### VARIABLE

### DESCRIPTION

NID	Node ID, see *NODE.
TC	Tranlational Constraint: EQ.0: no constraints, EQ.1: constrained x displacement, EQ.2: constrained y displacement, EQ.3: constrained z displacement, EQ.4: constrained x and y displacements, EQ.5: constrained y and z displacements, EQ.6: constrained z and x displacements.
RC	Rotational constraint: EQ.0: no constraints, EQ.1: constrained x rotation, EQ.2: constrained y rotation, EQ.3: constrained z rotation, EQ.4: constrained x and y rotations, EQ.5: constrained y and z rotations, EQ.6: constrained z and x rotations, EQ.7: constrained x, y, and z rotations.

# **\*INTERFACE**

---

## **Remarks:**

1. The default is NIKE3D with the THICKNESS option for each element and the NOTHICKNESS option is not available for NIKE3D. The file name for the NIKE3D option is "nikin". The adaptive constraint option is not available for this option.
2. The NOTHICKNESS option is available for DYNA3D and NASTRAN in which case the shell element thickness is not an output. The file name for LS-DYNA is "dynain" and for NASTRAN is "nastin." The \*CONTROL\_ADAPTIVITY is available for LS-DYNA.
3. Trimming is available for the adaptive mesh but it requires some steps. To trim an adaptive mesh use the following procedure:
  - (1) Generate the file, "dynain" using the keyword \*INTERFACE\_SPRINGBACK\_DYNA3D.
  - (2) Prepare a new input deck including the file "dynain."
  - (3) Add the keyword \*ELEMENT\_TRIM to this new deck.
  - (4) Add the keyword \*DEFINE\_CURVE\_TRIM to this new deck.
  - (5) Run this new input deck with i=input\_file\_name. The adaptive constraints are eliminated by remeshing and the trimming is performed.
  - (6) In case this new trimmed mesh is needed, run a zero termination time job and output the file generated via the keyword, \*INTERFACE\_SPRINGBACK\_DYNA3D.

## **Remarks for Seamless Springback:**

Seamless springback avoids the use of LS-NIKE3D for static springback analysis. Instead, LS-DYNA automatically and seamlessly switches from explicit dynamic to implicit static mode at the end of a forming simulation, and continues to run the static springback analysis. Seamless springback can be activated in the original LS-DYNA input deck, or later using a small restart input deck. In this way, the user can decide to continue a previous forming analysis by restarting to add the implicit springback phase. (Another alternative approach to springback simulation is to use the keyword \*INTERFACE\_SPRINGBACK\_DYNA3D to generate a "dynain" file after forming, and then perform a second simulation running LS-DYNA in fully implicit mode for springback. See Appendix M for a description of how to run an implicit analysis using LS-DYNA.

The implicit springback phase begins when the forming simulation termination time ENDTIM is reached, as specified with the keyword \*CONTROL\_TERMINATION. Since the springback phase is static, its termination time can be chosen arbitrarily (unless material rate



effects are included). The default choice is  $2.0*ENDTIM$ , and can be changed using the `*CONTROL_IMPLICIT_GENERAL` keyword.

Since the springback analysis is a static simulation, a minimum number of essential boundary conditions or Single Point Constraints (SPC's) are required to prohibit rigid body motion of the part. These boundary conditions can be added for the springback phase using the input option on the `*INTERFACE_SPRINGBACK_SEAMLESS` keyword above.

Several new `*CONTROL_IMPLICIT` keywords have been added to control the implicit springback phase. These keywords can also be added to a restart input deck. Generally, default settings can be used, so these keywords need not be included in the input deck.

To obtain accurate springback solutions, a nonlinear springback analysis must be performed. In many simulations, this iterative equilibrium search will converge without difficulty. If the springback simulation is particularly difficult, either due to nonlinear deformation, nonlinear material response, or numerical precision errors, a multi-step springback simulation will be automatically invoked. In this approach, the springback deformation is divided into several smaller, more manageable steps.

Two specialized features in LS-DYNA are used to perform multi-step springback analyses. The addition and gradual removal of artificial springs is performed by the artificial stabilization feature. Simultaneously, the automatic time step control is used to guide the solution to the termination time as quickly as possible, and to persistently retry steps where the equilibrium search has failed. By default, both of these features are active during a seamless springback simulation. However, the default method attempts to solve the springback problem in a single step. If this is successful, the solution will terminate normally. If the single step springback analysis fails to converge, the step size will be reduced, and artificial stabilization will become active. Defaults for these features can be changed using the `*CONTROL_IMPLICIT_GENERAL`, `*CONTROL_IMPLICIT_AUTO` and `*CONTROL_IMPLICIT_STABILIZATION` keywords.

# **\*INTERFACE**

---

# **\*LOAD**

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

- \*LOAD\_BEAM\_OPTION**
- \*LOAD\_BODY\_OPTION**
- \*LOAD\_BODY\_GENERALIZED**
- \*LOAD\_BRODE**
- \*LOAD\_DENSITY\_DEPTH**
- \*LOAD\_HEAT\_GENERATION\_OPTION**
- \*LOAD\_MASK**
- \*LOAD\_NODE\_OPTION**
- \*LOAD\_RIGID\_BODY**
- \*LOAD\_SEGMENT**
- \*LOAD\_SEGMENT\_SET**
- \*LOAD\_SHELL\_OPTION**
- \*LOAD\_SSA**
- \*LOAD\_SUPERPLASTIC\_FORMING**
- \*LOAD\_THERMAL\_OPTION**

# \*LOAD

---

## \*LOAD\_BEAM\_OPTION

Options include:

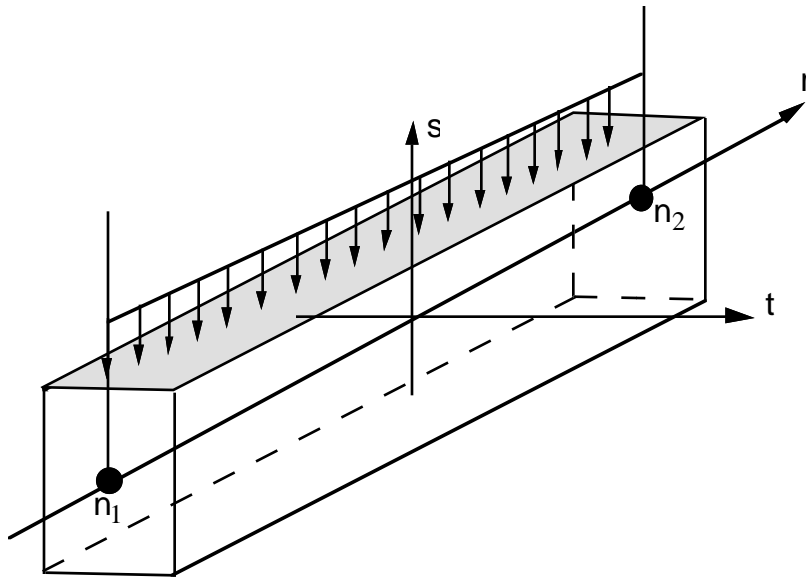
### ELEMENT SET

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

### Card Format

	1	2	3	4	5	6	7	8
Variable	EID/ESID	DAL	LCID	SF				
Type	I	I	I	F				
Default	none	none	none	1.				
Remarks								

<u>VARIABLE</u>	<u>DESCRIPTION</u>
EID/ESID	Beam ID (EID) or beam set ID (ESID), see *ELEMENT_BEAM or *SET_BEAM.
DAL	Direction of applied load: EQ.1: along r-axis of beam, EQ.2: along s-axis of beam, EQ.3: along t-axis of beam.
LCID	Load curve ID, see *DEFINE_CURVE.
SF	Load curve scale factor. This is for a simple modification of the function values of the load curve.



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

# \*LOAD

---

## \*LOAD\_BODY\_OPTION

Options include for base accelerations:

**X**

**Y**

**Z**

for angular velocities:

**RX**

**RY**

**RZ**

and to specify a part set:

**PARTS**

Purpose: Define body force loads due to a prescribed base acceleration or angular velocity using global axes directions. This data applies to all nodes in the complete problem unless a part subset is specified via the \*LOAD\_BODY\_PARTS keyword. If a part subset is defined then all nodal points belonging to the subset will have body forces applied. The parts specified via the \*LOAD\_BODY\_PARTS keyword apply to the options X, Y, Z, RX, RY, and RZ above, i.e., different part sets may not apply to different options. Only one part set is expected. **Note: This option applies nodal forces, i.e., it cannot be used to prescribe translational or rotational motion.** Two keyword definitions are needed to apply body loads on a subset of parts: \*LOAD\_BODY\_X and \*LOAD\_BODY\_PARTS.

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

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

**Card Format for option: PARTS.**

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

---

**VARIABLE**

---

**DESCRIPTION**

LCID	Load curve ID, see *DEFINE_CURVE.
SF	Load curve scale factor
LCIDDR	Load curve ID for dynamic relaxation phase (optional). This is only needed if dynamic relaxation is defined and a different load curve to LCID is required during the dynamic relaxation phase. Note if LCID is set to zero then no body load will be applied during dynamic relaxation regardless of the value LCIDDR is set to. See *CONTROL_DYNAMIC_RELAXATION
XC	X-center of rotation, define for angular velocities.
YC	Y-center of rotation, define for angular velocities.
ZC	Z-center of rotation, define for angular velocities.
PSID	Part set ID.

**Remarks:**

1. Translational base accelerations allow body forces loads to be imposed on a structure. Conceptually, base acceleration may be thought of as accelerating the coordinate system in the direction specified, and, thus, the inertial loads acting on the model are of opposite sign. For example, if a cylinder were fixed to the y-z plan and extended in the positive x-direction, then a positive x-direction base acceleration would tend to shorten the cylinder, i.e., create forces acting in the negative x-direction.
2. Base accelerations are frequently used to impose gravitational loads during dynamic relaxation to initialize the stresses and displacements. During the analysis, in this latter case, the body forces loads are held constant to simulate gravitational loads. When imposing loads during dynamic relaxation, it is recommended that the load curve slowly ramp up to avoid the excitation of a high frequency response.
3. Body force loads due to the angular velocity about an axis are calculated with respect to the deformed configuration and act radially outward from the axis of rotation. Torsional effects

## \*LOAD

---

which arise from changes in angular velocity are neglected with this option. The angular velocity is assumed to have the units of radians per unit time.

4. The body force density is given at a point P of the body by:

$$b = \rho(\omega \times \omega \times r)$$

where  $\rho$  is the mass density,  $\omega$  is the angular velocity vector, and  $r$  is a position vector from the origin to point P. Although the angular velocity may vary with time, the effects of angular acceleration are not included.

5. Angular velocities are useful for studying transient deformation of spinning three-dimensional objects. Typical applications have included stress initialization during dynamic relaxation where the initial rotational velocities are assigned at the completion of the initialization, and this option ceases to be active.



```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$ *LOAD_BODY_Z
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$ Add gravity such that it acts in the negative Z-direction.
$ Use units of mm/ms2. Since gravity is constant, the load
$ curve is set as a constant equal to 1. If the simulation
$ is to exceed 1000 ms, then the load curve needs to be
$ extended.
$
$$$ Note: Positive body load acts in the negative direction.
$
*LOAD_BODY_Z
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$   lcid      sf   lciddr      xc       yc       zc
$       5   0.00981
$
$
$ *DEFINE_CURVE
$   lcid   sidr   scla   sclo   offa   offo
$       5
$
$       abscissa          ordinate
$           0.00          1.000
$        1000.00         1.000
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

# \*LOAD

---

## \*LOAD\_BODY\_GENERALIZED

Purpose: Define body force loads due to a prescribed base acceleration or a prescribed angular velocity over a subset of the complete problem. The subset is defined by using nodes.

### Card Format

Card 1            1            2            3            4            5            6            7            8

Variable	N1	N2	LCID	DRLCID	XC	YC	ZC	
Type	I	I	I	I	F	F	F	
Default	none	none	none	0	0.	0.	0.	
Remarks								

Card 2            1            2            3            4            5            6            7            8

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

---

### VARIABLE

### DESCRIPTION

N1            Beginning node ID for body force load.

N2            Ending node ID for body force load.

LCID            Load curve ID, see \*DEFINE\_CURVE.

DRLCID            Load curve ID for dynamic relaxation phase. Only necessary if dynamic relaxation is defined. See \*CONTROL\_DYNAMIC\_RELAXATION.

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
XC	X-center of rotation. Define only for angular velocity.
YC	Y-center of rotation. Define only for angular velocity.
ZC	Z-center of rotation. Define only for angular velocity.
AX	Scale factor for acceleration in x-direction
AY	Scale factor for acceleration in y-direction
AZ	Scale factor for acceleration in z-direction
OMX	Scale factor for x-angular velocity
OMY	Scale factor for y-angular velocity
OMZ	Scale factor for z-angular velocity

**Remarks:**

1. Translational base accelerations allow body forces loads to be imposed on a structure. Conceptually, base acceleration may be thought of as accelerating the coordinate system in the direction specified, and, thus, the inertial loads acting on the model are of opposite sign. For example, if a cylinder were fixed to the y-z plane and extended in the positive x-direction, then a positive x-direction base acceleration would tend to shorten the cylinder, i.e., create forces acting in the negative x-direction.
2. Base accelerations are frequently used to impose gravitational loads during dynamic relaxation to initialize the stresses and displacements. During the analysis, in this latter case, the body forces loads are held constant to simulate gravitational loads. When imposing loads during dynamic relaxation, it is recommended that the load curve slowly ramp up to avoid the excitation of a high frequency response.
3. Body force loads due to the angular velocity about an axis are calculated with respect to the deformed configuration and act radially outward from the axis of rotation. Torsional effects which arise from changes in angular velocity are neglected with this option. The angular velocity is assumed to have the units of radians per unit time.
4. The body force density is given at a point P of the body by:

$$b = \rho(\omega \times \omega \times r)$$

where  $\rho$  is the mass density,  $\omega$  is the angular velocity vector, and  $r$  is a position vector from the origin to point P. Although the angular velocity may vary with time, the effects of angular acceleration are included.

5. Angular velocities are useful for studying transient deformation of spinning three-dimensional objects. Typical applications have included stress initialization during dynamic relaxation where the initial rotational velocities are assigned at the completion of the initialization, and this option ceases to be active.

# \*LOAD

---

## \*LOAD\_BRODE

Purpose: Define Brode function for application of pressure loads due to explosion, see Brode [1970], also see \*LOAD\_SEGMENT, \*LOAD\_SEGMENT\_SET, or LOAD\_SHELL.

### Card Format

Card 1            1            2            3            4            5            6            7            8

Variable	YLD	BHT	XBO	YBO	ZBO	TBO	TALC	SFLC
Type	F	F	F	F	F	F	I	I
Default	0.0	0.0	0.0	0.0	0.0	0.0	0	0
Remarks							1	1

Card 2            1            2            3            4            5            6            7            8

Variable	CFL	CFT	CFP					
Type	F	F	F					
Default	0.0	0.0	0.0					

---

### VARIABLE

### DESCRIPTION

YLD            Yield (Kt, equivalent tons of TNT).  
BHT            Height of burst.  
XBO            x-coordinates of Brode origin.  
YBO            y-coordinates of Brode origin.  
ZBO            z-coordinates of Brode origin.  
TBO            Time offset of Brode origin.

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
TALC	Load curve number giving time of arrival versus range relative to Brode origin (space, time), see *DEFINE_CURVE and remark below.
SFLC	Load curve number giving yield scaling versus scaled time (time relative to Brode origin divided by $[\text{yield}^{**1/3}]$ )origin (space, time), see *DEFINE_CURVE and remark below.
CFL	Conversion factor - kft to LS-DYNA length units.
CFT	Conversion factor - milliseconds to LS-DYNA time units.
CFP	Conversion factor - psi to LS-DYNA pressure units.

**Remark:**

1. If these curves are defined a variable yield is assumed. Both load curves must be specified for the variable yield option. If this option is used, the shock time of arrival is found from the time of arrival curve. The yield used in the Brode formulas is computed by taking the value from the yield scaling curve at the current time/ $[\text{yield}^{**1/3}]$  and multiplying that value by yield.

# \*LOAD

---

## \*LOAD\_DENSITY\_DEPTH

Purpose: Define density versus depth for gravity loading. This option has been occasionally used for analyzing underground and submerged structures where the gravitational preload is important. The purpose of this option is to initialize the hydrostatic pressure field at the integration points in the element.

This card should be only defined once in the input deck.

### Card Format

	1	2	3	4	5	6	7	8
Variable	PSID	GC	DIR	LCID				
Type	I	F	I	I				
Default	0	0.0	1	none				
Remarks	1,2			3				

---

#### VARIABLE

#### DESCRIPTION

PSID	Part set ID, see *SET_PART. If a PSID of zero is defined then all parts are initialized.
GC	Gravitational acceleration value.
DIR	Direction of loading: EQ.1: global x, EQ.2: global y, EQ.3: global z.
LCID	Load curve ID defining density versus depth, see *DEFINE_CURVE.

#### **Remarks:**

1. Density versus depth curves are used to initialize hydrostatic pressure due to gravity acting on an overburden material. The hydrostatic pressure acting at a material point at depth,  $d$ , is given by:

$$p = - \int_d^{d_{surface}} \rho(z) g dz$$

where  $p$  is pressure,  $d_{surface}$ , is the depth of the surface of the material to be initialized (usually zero),  $\rho(z)$  is the mass density at depth  $z$ , and  $g$  is the acceleration of gravity. This integral is evaluated for each integration point. Depth may be measured along any of the global coordinate axes, and the sign convention of the global coordinate system should be respected. The sign convention of gravity also follows that of the global coordinate system. For example, if the positive  $z$  axis points "up", then gravitational acceleration should be input as a negative number.

2. For this option there is a limit of 12 parts that can be defined by PSID, unless all parts are initialized.
3. Depth is the ordinate of the curve and is input as a descending x, y, or z coordinate value. Density is the abscissa of the curve and must vary (increase) with depth, i.e., an infinite slope is not allowed.

# \*LOAD

---

## \*LOAD\_HEAT\_GENERATION\_OPTION

Available options are:

### SET SOLID

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

### Card Format

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

### VARIABLE

### DESCRIPTION

SID	Solid element set ID or solid element ID, see *SET_SOLID or *ELEMENT_SOLID, respectively.
LCID	Load curve ID for volumetric heat generation rate, $\dot{q}'''$ : GT.0: function versus time, EQ.0: use multiplier value CMULT only, LT.0: function versus temperature.
CMULT	Curve multiplier for $\dot{q}'''$ . Depending on the definition of LCID this value is either used for scaling or for constant heat generation.



**\*LOAD\_MASK**

Purpose: Apply a distributed pressure load over a three-dimensional shell part. The pressure is applied to a subset of elements that are within a fixed global box and lie either outside or inside of a closed curve in space which is projected onto the surface.

**Card Format**

	1	2	3	4	5	6	7	8
Variable	PID	LCID	VID1	OFF	BOXID	LCIDM	VID2	INOUT
Type	I	I	F	F	I	I	I	I
Default	none	none	1.	0.	0	0	none	0
Remarks	1		2					

	1	2	3	4	5	6	7	8
Variable	ICYCLE							
Type	I							
Default	200							
Remarks								

---

**VARIABLE**

---

**DESCRIPTION**

PID

Part ID (PID). This part must consist of 3D shell elements. To use this option with solid element the surface of the solid elements must be covered with null shells. See \*MAT\_NULL.

LCID

Curve ID defining the pressure time history, see \*DEFINE\_CURVE.

# \*LOAD

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
VID1	Vector ID normal to the surface on which the applied pressure acts. Positive pressure acts in a direction that is in the opposite direction. This vector may be used if the surface on which the pressure acts is relatively flat. If zero, the pressure load depends on the orientation of the shell elements as shown in Figure 19.3.
OFF	Pressure loads will be discontinued if $ VID1 \cdot n_{shell}  < OFF$ where $n_{shell}$ is the normal vector to the shell element.
BOXID	Only elements inside the box with part ID, SSID, are considered. If no ID is given all elements of part ID, SSID, are included. When the active list of elements are updated, elements outside the box will no longer have pressure applied, i.e., the current configuration is always used.
LCIDM	Curve ID defining the mask. This curve gives (x,y) pairs of points in a local coordinate system defined by the vector ID, VID2. Generally, the curve should form a closed loop, i.e., the first point is identical to the last point, and the curve should be flagged as a DATTYP=1 curve in the *DEFINE_CURVE section. If no curve ID is given, all elements of part ID, PID, are included with the exception of those deleted by the box. The mask works like the trimming option, i.e., see DEFINE_CURVE_TRIM and Figure 10.4.
VID2	Vector ID used to project the masking curve onto the surface of part ID, PID. The origin of this vector determines the origin of the local system that the coordinates of the PID are transformed into prior to determining the pressure distribution in the local system. This curve must be defined if LCIDM is nonzero.
INOUT	If 0, elements whose center falls inside the projected curve are considered. If 1, elements whose center falls outside the projected curve are considered.
ICYCLE	Number of time steps between updating the list of active elements (default=200). The list update can be quite expensive and should be done at a reasonable interval. The default is not be appropriate for all problems.

## Remarks:

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

**\*LOAD\_NODE\_OPTION**

Options include:

**POINT  
SET**

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

**Card Format**

	1	2	3	4	5	6	7	8
Variable	NODE/NSID	DOF	LCID	SF	CID	M1	M2	M3
Type	I	I	I	F	I	I	I	I
Default	none	none	none	1.	0	0	0	0
Remarks					1	2		

**VARIABLE****DESCRIPTION**

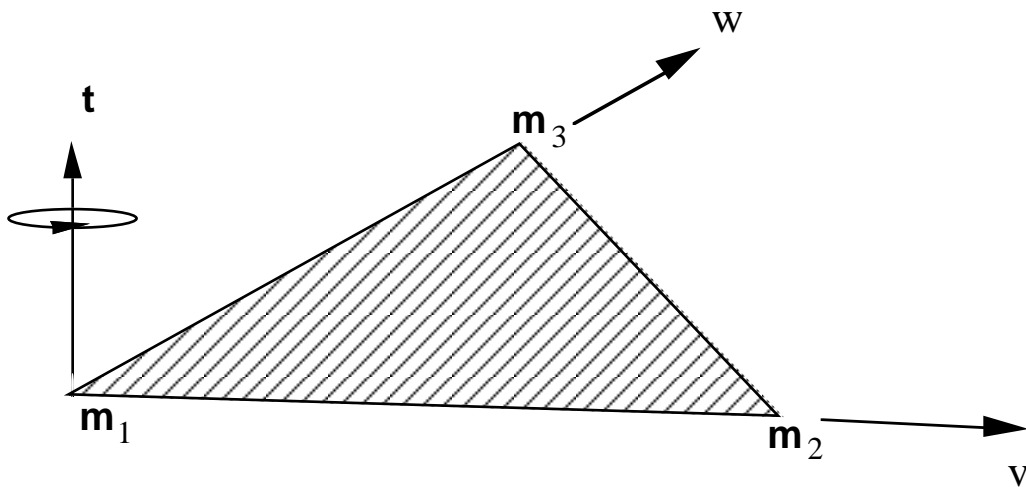
NODE/NSID	Node ID or nodal set ID (NSID), see *SET_NODE_OPTION.
DOF	Applicable degrees-of-freedom: EQ.1: x-direction of load action, EQ.2: y-direction of load action, EQ.3: z-direction of load action, EQ.4: follower force, see remark 2 on next page, EQ.5: moment about the x-axis, EQ.6: moment about the y-axis, EQ.7: moment about the z-axis. EQ.8: follower moment
LCID	Load curve ID, see *DEFINE_CURVE.
SF	Load curve scale factor.
CID	Coordinate system ID (optional), see remark 1 on next page.

# \*LOAD

VARIABLE	DESCRIPTION
M1	Node 1 ID. Only necessary if DOF.EQ.4 or 8, see remark 2 below.
M2	Node 2 ID. Only necessary if DOF.EQ.4 or 8, see remark 2 below.
M3	Node 3 ID. Only necessary if DOF.EQ.4 or 8, see remark 2 below.

## Remarks:

- 1 The global coordinate system is the default. The local coordinate system ID's are defined in the \*DEFINE\_COORDINATE\_SYSTEM section.
- 2 Nodes  $M_1$ ,  $M_2$ ,  $M_3$  must be defined for a follower force. A positive follower force acts normal to the plane defined by these nodes, and a positive follower moment puts a counterclockwise torque about the  $t$ -axis. These actions are depicted in Figure 19.2.
- 3 For shell formulations 14 and 15, the axisymmetric solid elements with area and volume weighting, respectively, the specified nodal load is per unit length (type 14) and per radian (type 15).



**Figure 19.2.** Follower force and moment acting on a plane defined by nodes  $m_1$ ,  $m_2$ , and  $m_3$ . In this case, the load is applied to node  $m_1$ ; i.e.,  $m=m_1$ . A positive force acts in the positive  $t$  direction, and a positive moment puts a counterclockwise torque about the normal vector. The positive  $t$ -direction is found by the cross product  $t = v \times w$  where  $v$  and  $w$  are vectors as shown.

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$ *LOAD_NODE_SET
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$ A cantilever beam (made from shells) is loaded on the two end nodes
$ (nodes 21 & 22). The load is applied in the y-direction (dof=2).
$ Load curve number 1 defines the load, but is scaled by sf=0.5 in the
$ *LOAD_NODE_SET definition.
$
*LOAD_NODE_SET
$
$.>...1.>...2.>...3.>...4.>...5.>...6.>...7.>...8
$   nsid      dof      lcid      sf      cid      m1      m2      m3
$   14        2        1        0.5
$
$
$
*SET_NODE_LIST
$   sid
$   14
$
$   nid1      nid2      nid3      nid4      nid5      nid6      nid7      nid8
$   21        22
$
$
*DEFINE_CURVE
$   lcid      sidr      scla      sclo      offa      offo
$   1
$
$   abscissa      ordinate
$   0.0            0.0
$   10.0           100.0
$   20.0           0.0
$
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

# \*LOAD

---

## \*LOAD\_RIGID\_BODY

Purpose: Apply a concentrated nodal force to a rigid body. The force is applied at the center of mass or a moment is applied around a global axis. As an option, local axes can be defined for force or moment directions.

### Card Format

	1	2	3	4	5	6	7	8
Variable	PID	DOF	LCID	SF	CID	M1	M2	M3
Type	I	I	I	F	I	I	I	I
Default	none	none	none	1.	0	0	0	0
Remark					1	2		

<u>VARIABLE</u>	<u>DESCRIPTION</u>
PID	Part ID of the rigid body, see *PART_OPTION.
DOF	Applicable degrees-of-freedom: EQ.1: x-direction of load action, EQ.2: y-direction of load action, EQ.3: z-direction of load action, EQ.4: follower force, see remark 2 on next page, EQ.5: moment about the x-axis, EQ.6: moment about the y-axis, EQ.7: moment about the z-axis. EQ.8: follower moment, see remark 2.
LCID	Load curve ID, see *DEFINE_CURVE. GT.0: force as a function of time, LT.0: force as a function of the absolute value of the rigid body displacement.
SF	Load curve scale factor
CID	Coordinate system ID
M1	Node 1 ID. Only necessary if DOF.EQ.4 or 8, see remark 2 on next page.

VARIABLE	DESCRIPTION
M2	Node 2 ID. Only necessary if DOF.EQ.4 or 8, see remark 2.
M3	Node 3 ID. Only necessary if DOF.EQ.4 or 8, see remark 2.

**Remarks:**

- 1 The global coordinate system is the default. The local coordinate system ID's are defined in the \*DEFINE\_COORDINATE\_SYSTEM section. This local axis is fixed in inertial space, i.e., it does not move with the rigid body.
2. Nodes M<sub>1</sub>, M<sub>2</sub>, M<sub>3</sub> must be defined for a follower force or moment. The follower force acts normal to the plane defined by these nodes as depicted in Figure 19.2. The positive t-direction is found by the cross product  $t = v \times w$  where  $v$  and  $w$  are vectors as shown. The follower force is applied at the center of mass. A positive follower moment puts a counterclockwise torque about the t-axis.

```

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

```

# \*LOAD

---

## \*LOAD\_SEGMENT

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

### Card Format

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

---

### VARIABLE

### DESCRIPTION

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

### Remarks:

- 1 If LCID is input as -1, then the Brode function is used to determine the pressure for the segments, see \*LOAD\_BRODE.
- 2 The load curve multipliers may be used to increase or decrease the pressure. The time value is not scaled.
- 3 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.

- 4. Triangular segments are defined by repeating the third node.

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

# \*LOAD

---

## \*LOAD\_SEGMENT\_SET

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

### Card Format

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

---

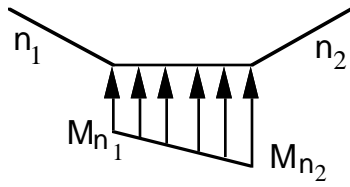
### VARIABLE

### DESCRIPTION

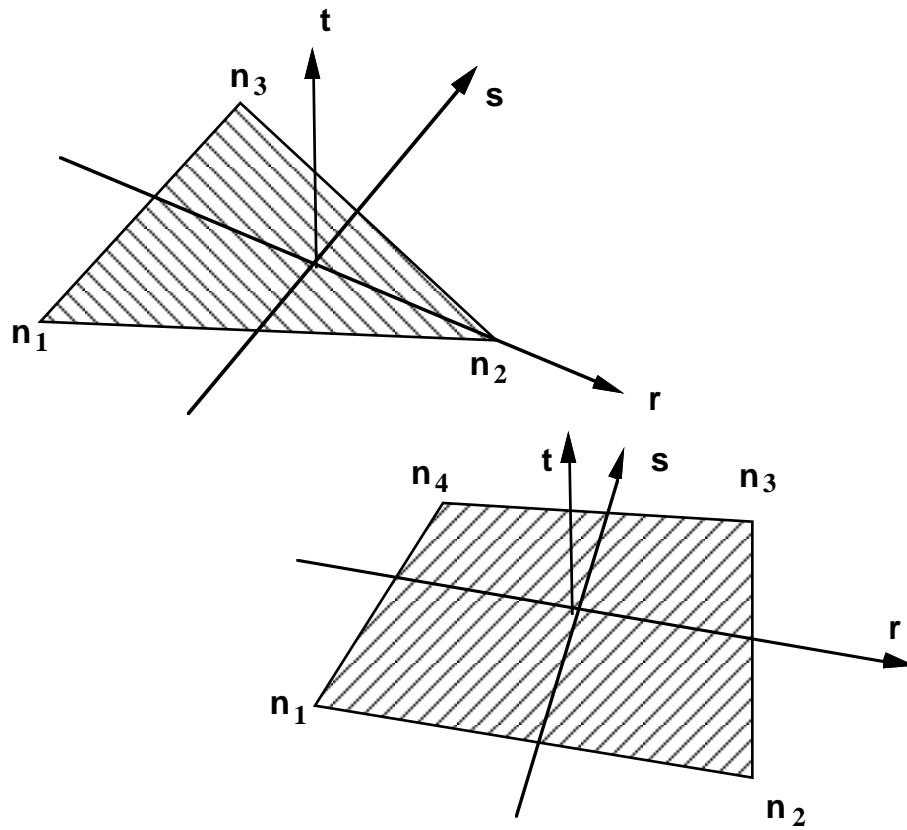
SSID	Segment set ID, see *SET_SEGMENT.
LCID	Load curve ID, see *DEFINE_CURVE.
SF	Load curve scale factor
AT	Arrival time for pressure or birth time of pressure.

### Remarks:

1. If LCID is input as -1, then the Brode function is used to determine pressure for the segment set, also see \*LOAD\_BRODE.
2. The load curve multipliers may be used to increase or decrease the pressure. The time value is not scaled.
3. The activation time, AT, is the time during the solution that the pressure begins to act. Until this time, the pressure is ignored. The function value of the load curves will be evaluated at the offset time given by the difference of the solution time and AT i.e., (solution time-AT). Relative displacements that occur prior to reaching AT are ignored. Only relative displacements that occur after AT are prescribed.



2-Dimensional Definition for axisymmetric, plane stress, and plane strain geometries



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

# \*LOAD

---

## \*LOAD\_SHELL\_OPTION

Options include:

### ELEMENT SET

Purpose: Apply the distributed pressure load over one shell element or shell element set. The numbering of the shell nodal connectivities must follow the right hand rule with positive pressure acting in the negative t-direction. See Figure 19.3. This option applies to the three-dimensional shell elements only.

### Card Format

	1	2	3	4	5	6	7	8
Variable	EID/ESID	LCID	SF	AT				
Type	I	I	F	F				
Default	none	none	1.	0.				
Remarks	1	1	2					

---

#### VARIABLE

#### DESCRIPTION

EID/ESID	Shell ID (SID) or shell set ID (SSID), see *ELEMENT_SHELL or *SET_SHELL.
LCID	Load curve ID, see *DEFINE_CURVE.
SF	Load curve scale factor
AT	Arrival time for pressure or birth time of pressure.

### Remarks:

- 1 If LCID is input as -1, then the Brode function is used to determine the pressure for the segments, see also \*LOAD\_BRODE.

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

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$ *LOAD_SHELL_ELEMENT
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$ From a sheet metal forming example. A blank is hit by a punch, a holder is
$ used to hold the blank on its sides. All shells on the holder are given a
$ pressure boundary condition to clamp down on the blank. The pressure
$ follows load curve 3, but is scaled by -1 so that it applies the load in the
$ correct direction. The load starts at zero, but quickly rises to 5 MPa
$ after 0.001 sec. (Units of this model are in: ton, mm, s, N, MPa, N-mm)
$
*LOAD_SHELL_ELEMENT
$
$...>....1...>....2...>....3...>....4...>....5...>....6...>....7...>....8
$   eid      lcid      sf      at
$   30001     3 -1.00E+00    0.0
$   30002     3 -1.00E+00    0.0
$   30003     3 -1.00E+00    0.0
$   30004     3 -1.00E+00    0.0
$   30005     3 -1.00E+00    0.0
$   30006     3 -1.00E+00    0.0
$   30007     3 -1.00E+00    0.0
$
$ Note: Just a subset of all the shell elements of the holder is shown above,
$       in practice this list contained 448 shell element id's.
$
$
$
*DEFINE_CURVE
$   lcid      sidr      scla      sclo      offa      offo
$           3
$
$       abscissa      ordinate
$           0.000           0.0
$           0.001           5.0
$           0.150           5.0
$
$
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

# \*LOAD

---

## \*LOAD\_SSA

Purpose: The Sub-Sea Analysis capability allows a simple way of loading the structure to account for the effects of the primary explosion and the subsequent bubble oscillations.

### Define one card.

Card 1            1            2            3            4            5            6            7            8

Variable	VS	DS	REFL	ZB	ZSURF	FPSID	PSID	
Type	F	F	F	F	F	I	I	
Default	none	none	0.	0.	0.	0	0	

Define two cards for each explosive charge. This input is terminated by the next “\*” keyword card.

Card 1            1            2            3            4            5            6            7            8

Variable	A	ALPHA	GAMMA	KTHETA	KAPPA			
Type	F	F	F	F	F			
Default	none	none	none	none	none			

Card 2            1            2            3            4            5            6            7            8

Variable	XS	YS	ZS	W	TDELY	RAD	CZ	
Type	F	F	F	F	F	F	F	
Default	none	none	none	none	none	none	none	

---

<b>VARIABLE</b>	<b>DESCRIPTION</b>
VS	Sound speed in fluid
DS	Density of fluid
REFL	Consider reflections from sea floor. EQ.0: off EQ.1: on
ZB	Z coordinate of sea floor if REFL=1, otherwise, not used.
ZSURF	Z coordinate of sea surface
FPSID	Part set ID of parts subject to flood control. Use the *PART_SET_COLUMN option where the parameters A1 and A2 must be defined as follows:  Parameter A1: Flooding status: EQ.1.0: Fluid on both sides. EQ.2.0: Fluid outside, air inside. EQ.3.0: Air outside, fluid inside. EQ.4.0: Material or part is ignored.  Parameter A2: Tubular outer diameter of beam elements. For shell elements this input must be greater than zero for loading.
PSID	Part IDs of parts defining the wet surface. The elements defining these parts must have their outward normals pointing into the fluid. See Figure 19.4. EQ.0: all parts are included. GT.0: define NPIDS part ID's below.
A	Shock pressure parameter
ALPHA	$\alpha$ , shock pressure parameter
GAMMA	$\gamma$ , time constant parameter
KTHETA	$K_\theta$ , time constant parameter
KAPPA	$\kappa$ , ratio of specific heat capacities
XS	X coordinate of charge
YS	Y coordinate of charge
ZS	Z coordinate of charge
W	Weight of charge

# \*LOAD

---

VARIABLE	DESCRIPTION
TDELY	Time delay before charge detonates
RAD	Charge radius
CZ	Water depth

## Remarks:

The pressure history of the primary shockwave at a point in space through which a detonation wave passes is given as:

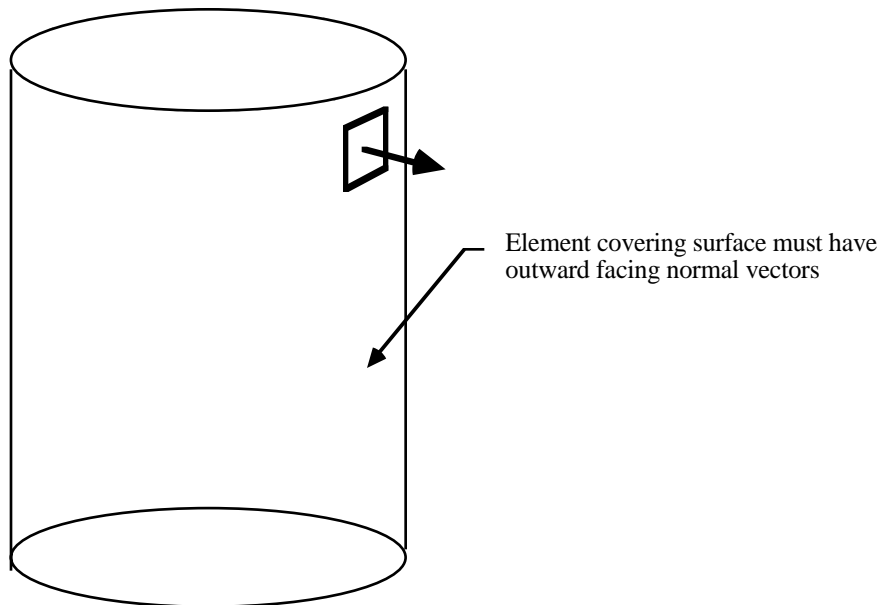
$$P(t) = P_m e^{-\frac{t}{\theta}}$$

where  $P_m$  and the time constant  $\theta$  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  $A$ ,  $\alpha$ ,  $\gamma$ , and  $K_\theta$  are constants for the explosive being used.



**Figure 19.4.** The shell elements interacting with the fluid must be numbered such that their outward normal vector points into the fluid media.



**\*LOAD\_SUPERPLASTIC\_FORMING**

Purpose: Perform superplastic forming (SPF) analyses. This option can be applied to both solid and shell elements. The pressure loading controlled by the load curve ID given below is scaled to maintain a constant maximum strain rate.

This option must be used with material model 64, \*MAT\_RATE\_SENSITIVE\_POWERLAW\_PLASTICITY, for strain rate sensitive, powerlaw plasticity. For the output of data, see \*DATABASE\_SUPERPLASTIC\_FORMING. Mass scaling is recommended in SPF applications.

**Card Format**

	1	2	3	4	5	6	7	8
Variable	LCP1	CSP1	NCP1	LCP2	CSP2	NCP2		
Type	I	I	F	I	I	F		
Default	none	none	none.	none	none	none		
Remarks				1	1	1		

	1	2	3	4	5	6	7	8
Variable	ERATE	SCMIN	SCMAX	NCYL				
Type	F	F	F	I				
Default	none	none	none.	0				
Remarks				2				

# \*LOAD

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
LCP1	Load curve number for Phase I pressure loading, see *DEFINE_CURVE.
CSP1	Contact surface number to determine completion of Phase 1.
NCP1	Percent of nodes in contact to terminate Phase I, see *CONTACT_OPTION.
LCP2	Load curve number for Phase II pressure loading (reverse), see *DEFINE_CURVE.
CSP2	Contact surface to determine completion of Phase II, see *CONTACT_OPTION.
NCP2	Percent of nodes in contact to terminate Phase II.
ERATE	Desired strain rate. This is the time derivative of the logarithmic strain.
SCMIN	Minimum allowable value for load curve scale factor. To maintain a constant strain rate the pressure curve is scaled. In the case of a snap through buckling the pressure may be removed completely. By putting a value here the pressure will continue to act but at a value given by this scale factor multiplying the pressure curve.
SCMAX	Maximum allowable value for load curve scale factor. Generally, it is a good idea to put a value here to keep the pressure from going to unreasonable values after full contact has been attained. When full contact is achieved the strain rates will approach zero and pressure will go to infinity unless it is limited or the calculation terminates.
NCYL	Number of cycles for monotonic pressure after reversal.

## **Remarks:**

1. Optionally, a second phase can be defined. In this second phase a unique set of pressure segments must be defined whose pressure is controlled by load curve 2. During the first phase, the pressure segments of load curve 2 are inactive, and, likewise, during the second phase the pressure segments of the first phase are inactive. When shell elements are used the complete set of pressure segments can be repeated in the input with a sign reversal used on the load curve. When solid elements are used the pressure segments for each phase will, in general, be unique.
2. This is an ad hoc parameter which should probably not be used.
3. The output files named: “pressure”, “curve1”, and “curve2”, may be plotted by LS-TAURUS in PHS3 using the SUPERPL command. The file “curve2” is created only if the second phase is active. See \*DATABASE\_SUPERPLASTIC\_FORMING.
4. The constraint method contact, \*CONTACT\_CONSTRAINT\_NODES\_TO\_SURFACE, is recommended for superplastic forming simulations since the penalty methods are not as reliable when mass scaling is applied. Generally, in superplastic simulations mass scaling is used to enable the calculation to be carried out in real time.

## **\*LOAD\_THERMAL\_OPTION**

Options include:

**CONSTANT**  
**CONSTANT\_NODE**  
**LOAD\_CURVE**  
**TOPAZ**  
**VARIABLE**  
**VARIABLE\_NODE**

Purpose: To define nodal temperatures that thermally load the structure. Nodal temperatures defined by the \*LOAD\_THERMAL\_OPTION method are all applied in a structural only analysis. They are ignored in a thermal only or coupled thermal/structural analysis, see \*CONTROL\_THERMAL\_OPTION.

All the \*LOAD\_THERMAL options cannot be used in conjunction with each other. Only those of the same thermal load type, as defined below in column 2, may be used together.

*LOAD_THERMAL_CONSTANT	- Thermal load type 1
*LOAD_THERMAL_CONSTANT_NODE	- Thermal load type 1
*LOAD_THERMAL_LOAD_CURVE	- Thermal load type 2
*LOAD_THERMAL_TOPAZ	- Thermal load type 3
*LOAD_THERMAL_VARIABLE	- Thermal load type 4
*LOAD_THERMAL_VARIABLE_NODE	- Thermal load type 4

# \*LOAD

---

## \*LOAD\_THERMAL\_CONSTANT

Purpose: Define nodal sets giving the temperature that remains constant for the duration of the calculation. The reference temperature state is assumed to be a null state with this option. A nodal temperature state, read in above and held constant throughout the analysis, dynamically loads the structure. Thus, the temperature defined can also be seen as a relative temperature to a surrounding or initial temperature.

### Card Format

Card 1            1            2            3            4            5            6            7            8

Variable	NSID	NSIDEX	BOXID					
Type	I	I	I					
Default	none	0.	0.					

Card 2            1            2            3            4            5            6            7            8

Variable	T	TE						
Type	F	F						
Default	0.	0.						

VARIABLE	DESCRIPTION
NSID	Nodal set ID containing nodes for initial temperature (see *SET_NODES): EQ.0: all nodes are included:
NSIDEX	Nodal set ID containing nodes that are exempted from the imposed temperature (optional).
BOXID	All nodes in box which belong to NSID are initialized. Others are excluded (optional).
T	Temperature
TE	Temperature of exempted nodes (optional)

**\*LOAD\_THERMAL\_CONSTANT\_NODE**

Purpose: Define nodal temperature that remains constant for the duration of the calculation. The reference temperature state is assumed to be a null state with this option. A nodal temperature state, read in above and held constant throughout the analysis, dynamically loads the structure. Thus, the temperature defined can also be seen as a relative temperature to a surrounding or initial temperature.

**Card Format**

	1	2	3	4	5	6	7	8
Variable	NID	T						
Type	I	F						
Default	none	0.						

**VARIABLE****DESCRIPTION**

NID	Node ID
T	Temperature

# \*LOAD

---

## \*LOAD\_THERMAL\_LOAD\_CURVE

Purpose: Nodal temperatures will be uniform throughout the model and will vary according to a load curve. It is assumed that the temperatures refer to a null state at the beginning and are thus relative temperatures. They dynamically load the structure.

### Card Format

1            2            3            4            5            6            7            8

Variable	LCID							
Type	I							
Default	none							

---

#### VARIABLE

#### DESCRIPTION

LCID

Load curve ID, see \*DEFINE\_CURVE to define temperature versus time.

## **\*LOAD\_THERMAL\_TOPAZ**

Purpose: Nodal temperatures will be read in from the TOPAZ3D database. This file is defined in the EXECUTION SYNTAX, see INTRODUCTION.

# \*LOAD

---

## \*LOAD\_THERMAL\_VARIABLE

Purpose: Define nodal sets giving the temperature that is variable in the duration of the calculation. The reference temperature state is assumed to be a null state with this option. A nodal temperature state, read in above and varied according to the load curve, dynamically loads the structure. Thus, the defined temperatures are relative temperatures to an initial reference temperature.

### Card Format

Card 1            1            2            3            4            5            6            7            8

Variable	NSID	NSIDEX	BOXID					
Type	I	I	I					
Default	none	0.	0.					

Card 2            1            2            3            4            5            6            7            8

Variable	TS	TB	LCID	TSE	TBE	LCIDE		
Type	F	F	I	F	F	I		
Default	0.	0.	none	0.	0.	none		
Remark	1	1	1	1	1			

<u>VARIABLE</u>	<u>DESCRIPTION</u>
NSID	Nodal set ID containing nodes (see *SET_NODE_OPTION): EQ.0: all nodes are included.
NSIDEX	Nodal set ID containing nodes that are exempted (optional), see *SET_NODE_OPTION.
BOXID	All nodes in box which belong to NSID are initialized. Others are excluded.
TS	Scaled temperature.



<u>VARIABLE</u>	<u>DESCRIPTION</u>
TB	Base temperature.
LCID	Load curve ID that multiplies the scaled temperature, see *DEFINE_CURVE.
TSE	Scaled temperature of the exempted nodes (optional).
TBE	Base temperature of the exempted nodes (optional).
LCIDE	Load curve ID that multiplies the scaled temperature of the exempted nodes (optional), see *DEFINE_CURVE.

**Remark:**

1. The temperature is defined as

$$T = T_{\text{base}} + T_{\text{scale}} f(t)$$

where  $f(t)$  is the current value of the load curve,  $T_{\text{scale}}$ , is the scaled temperature, and,  $T_{\text{base}}$ , is the base temperature.

# \*LOAD

---

## \*LOAD\_THERMAL\_VARIABLE\_NODE

Purpose: Define nodal temperature that are variable during the calculation. The reference temperature state is assumed to be a null state with this option. A nodal temperature state read in and varied according to the load curve dynamically loads the structure. Thus, the defined temperatures are relative temperatures to an initial reference temperature.

### Card Format

	1	2	3	4	5	6	7	8
Variable	NID	TS	TB	LCID				
Type	I	F	F	I				
Default	none	0.	0.	none				

---

#### VARIABLE

#### DESCRIPTION

NID	Node ID
TS	Scaled temperature
TB	Base temperature
LCID	Load curve ID that multiplies the scaled temperature, see *DEFINE_CURVE.

### Remarks:

The temperature is defined as

$$T = T_{\text{base}} + T_{\text{scale}} f(t)$$

where

$f(t)$  is the current value of the load curve

$T_{\text{scale}}$  is the scaled temperature

$T_{\text{base}}$  is the base temperature

# **\*MAT**

LS-DYNA has historically referenced materials by type identifiers. Below these identifiers are given with the corresponding keyword name. The numbers in brackets identify the element formulations for which the material model is implemented:

	<b>0</b>	- Solids,
	<b>1H</b>	- Hughes-Liu beam,
	<b>1B</b>	- Belytschko resultant beam,
	<b>1I</b>	- Belytschko integrated solid and tubular beams,
	<b>1T</b>	- Truss,
	<b>1D</b>	- Discrete beam,
	<b>1SW</b>	- Spotweld beam,
	<b>2</b>	- Shells,
	<b>3</b>	- Thick shells.
	<b>4</b>	- Special airbag element.
<b>TYPE 1:</b>	<b>*MAT_ELASTIC</b>	<b>[0,1H, 1B, 1I, 1T,2,3]</b>
<b>TYPE 2:</b>	<b>*MAT_ORTHOTROPIC_ELASTIC</b>	<b>[0,2,3]</b>
<b>TYPE 3:</b>	<b>*MAT_PLASTIC_KINEMATIC</b>	<b>[0,1H, 1I, 1T,2,3]</b>
<b>TYPE 4:</b>	<b>*MAT_ELASTIC_PLASTIC_THERMAL</b>	<b>[0,2,3]</b>
<b>TYPE 5:</b>	<b>*MAT_SOIL_AND_FOAM</b>	<b>[0]</b>
<b>TYPE 6:</b>	<b>*MAT_VISCOELASTIC</b>	<b>[0,1H]</b>
<b>TYPE 7:</b>	<b>*MAT_BLATZ-KO_RUBBER</b>	<b>[0,2]</b>
<b>TYPE 8:</b>	<b>*MAT_HIGH_EXPLOSIVE_BURN</b>	<b>[0]</b>
<b>TYPE 9:</b>	<b>*MAT_NULL</b>	<b>[0,1,2]</b>
<b>TYPE 10:</b>	<b>*MAT_ELASTIC_PLASTIC_HYDRO</b>	<b>[0]</b>
<b>TYPE 11:</b>	<b>*MAT_STEINBERG</b>	<b>[0]</b>
<b>TYPE 11:</b>	<b>*MAT_STEINBERG_LUND</b>	<b>[0]</b>
<b>TYPE 12:</b>	<b>*MAT_ISOTROPIC_ELASTIC_PLASTIC</b>	<b>[0,2,3]</b>
<b>TYPE 13:</b>	<b>*MAT_ISOTROPIC_ELASTIC_FAILURE</b>	<b>[0]</b>
<b>TYPE 14:</b>	<b>*MAT_SOIL_AND_FOAM_FAILURE</b>	<b>[0]</b>
<b>TYPE 15:</b>	<b>*MAT_JOHNSON_COOK</b>	<b>[0,2]</b>
<b>TYPE 16:</b>	<b>*MAT_PSEUDO_TENSOR</b>	<b>[0]</b>
<b>TYPE 17:</b>	<b>*MAT_ORIENTED_CRACK</b>	<b>[0]</b>
<b>TYPE 18:</b>	<b>*MAT_POWER_LAW_PLASTICITY</b>	<b>[0,1H,2,3]</b>
<b>TYPE 19:</b>	<b>*MAT_STRAIN_RATE_DEPENDENT_PLASTICITY</b>	<b>[0,2,3]</b>
<b>TYPE 20:</b>	<b>*MAT_RIGID</b>	<b>[0,1H,1B,1T,2,3]</b>
<b>TYPE 21:</b>	<b>*MAT_ORTHOTROPIC_THERMAL</b>	<b>[0,2,3]</b>
<b>TYPE 22:</b>	<b>*MAT_COMPOSITE_DAMAGE</b>	<b>[0,2,3]</b>

# \*MAT

---

TYPE 23: \*MAT\_TEMPERATURE\_DEPENDENT\_ORTHOTROPIC [0,2,3]  
TYPE 24: \*MAT\_PIECEWISE\_LINEAR\_PLASTICITY [0,1H,2,3]  
TYPE 25: \*MAT\_GEOLOGIC\_CAP\_MODEL [0]  
TYPE 26: \*MAT\_HONEYCOMB [0]  
TYPE 27: \*MAT\_MOONEY-RIVLIN\_RUBBER [0,2]  
TYPE 28: \*MAT\_RESULTANT\_PLASTICITY [1B,2]  
TYPE 29: \*MAT\_FORCE\_LIMITED [1B]  
TYPE 30: \*MAT\_CLOSED\_FORM\_SHELL\_PLASTICITY [2,3]  
TYPE 31: \*MAT\_FRAZER-NASH\_RUBBER [0]  
TYPE 32: \*MAT\_LAMINATED\_GLASS [2,3]  
TYPE 33: \*MAT\_BARLAT\_ANISOTROPIC\_PLASTICITY [0,2,3]  
TYPE 33: \*MAT\_BARLAT\_YLD96 [2,3]  
TYPE 34: \*MAT\_FABRIC [4]  
TYPE 35: \*MAT\_PLASTIC\_GREEN-NAGHDI\_RATE [0]  
TYPE 36: \*MAT\_3-PARAMETER\_BARLAT [2]  
TYPE 37: \*MAT\_TRANSVERSELY\_ANISOTROPIC\_ELASTIC\_PLASTIC [2,3]  
TYPE 38: \*MAT\_BLATZ-KO\_FOAM [0,2]  
TYPE 39: \*MAT\_FLD\_TRANSVERSELY\_ANISOTROPIC [2,3]  
TYPE 40: \*MAT\_NONLINEAR\_ORTHOTROPIC [2]  
TYPE 41-50: \*MAT\_USER\_DEFINED\_MATERIALS  
TYPE 42: \*MAT\_USER\_DEFINED\_PLANAR\_ANISOTROPIC\_PLASTICITY [2,3]  
TYPE 51: \*MAT\_BAMMAN [0,2,3]  
TYPE 52: \*MAT\_BAMMAN\_DAMAGE [0,2,3]  
TYPE 53: \*MAT\_CLOSED\_CELL\_FOAM [0]  
TYPE 54-55: \*MAT\_ENHANCED\_COMPOSITE\_DAMAGE [2]  
TYPE 57: \*MAT\_LOW\_DENSITY\_FOAM [0]  
TYPE 58: \*MAT\_LAMINATED\_COMPOSITE\_FABRIC [2]  
TYPE 59: \*MAT\_COMPOSITE\_FAILURE [0,2]  
TYPE 60: \*MAT\_ELASTIC\_WITH\_VISCOSITY [0,2]  
TYPE 61: \*MAT\_KELVIN-MAXWELL\_VISCOELASTIC [0]  
TYPE 62: \*MAT\_VISCOUS\_FOAM [0]  
TYPE 63: \*MAT\_CRUSHABLE\_FOAM [0]  
TYPE 64: \*MAT\_RATE\_SENSITIVE\_POWERLAW\_PLASTICITY [0,2,3]  
TYPE 65: \*MAT\_MODIFIED\_ZERILLI-ARMSTRONG [0,2]

TYPE 66: \*MAT\_LINEAR\_ELASTIC\_DISCRETE\_BEAM [1D]  
TYPE 67: \*MAT\_NONLINEAR\_ELASTIC\_DISCRETE\_BEAM [1D]  
TYPE 68: \*MAT\_NONLINEAR\_PLASTIC\_DISCRETE\_BEAM [1D]  
TYPE 69: \*MAT\_SID\_DAMPER\_DISCRETE\_BEAM [1D]  
TYPE 70: \*MAT\_HYDRAULIC\_GAS\_DAMPER\_DISCRETE\_BEAM [1D]  
TYPE 71: \*MAT\_CABLE\_DISCRETE\_BEAM [1D]  
TYPE 72: \*MAT\_CONCRETE\_DAMAGE [0]  
TYPE 73: \*MAT\_LOW\_DENSITY\_VISCOUS\_FOAM [0]  
TYPE 75: \*MAT\_BILKHU/DUBOIS\_FOAM [0]  
TYPE 76: \*MAT\_GENERAL\_VISCOELASTIC [0,2]  
TYPE 77A: \*MAT\_HYPERELASTIC\_RUBBER [0]  
TYPE 77B: \*MAT\_OGDEN\_RUBBER [0]  
TYPE 78: \*MAT\_SOIL\_CONCRETE [0]  
TYPE 79: \*MAT\_HYSTERETIC\_SOIL [0]  
TYPE 80: \*MAT\_RAMBERG-OSGOOD[0]  
TYPE 81: \*MAT\_PLASTICITY\_WITH\_DAMAGE [2,3]  
TYPE 83: \*MAT\_FU\_CHANG\_FOAM [0]  
TYPE 86: \*MAT\_ORTHOTROPIC\_VISCOELASTIC [2]  
TYPE 87: \*MAT\_CELLULAR\_RUBBER [0]  
TYPE 88: \*MAT\_MTS [0,2]  
TYPE 89: \*MAT\_PLASTICITY\_POLYMER [2]  
TYPE 90: \*MAT\_ACOUSTIC [0]  
TYPE 96: \*MAT\_BRITTLE\_DAMAGE [0]  
TYPE 100: \*MAT\_SPOTWELD [1SW]  
TYPE 103: \*MAT\_ANISOTROPIC\_VISCOPLASTIC [0,2]  
TYPE 104: \*MAT\_DAMAGE\_1 [0,2]  
TYPE 105: \*MAT\_DAMAGE\_2 [0,2]  
TYPE 123: \*MAT\_MODIFIED\_PIECEWISE\_LINEAR\_PLASTICITY [2,3]  
TYPE 124: \*MAT\_PLASTICITY\_COMPRESSION\_TENSION [0]  
TYPE 126: \*MAT\_MODIFIED\_HONEYCOMB [0]  
TYPE 127: \*MAT\_ARRUDA\_BOYCE [0]  
TYPE 128: \*MAT\_HEART\_TISSUE [0]  
TYPE 129: \*MAT\_LUNG\_TISSUE [0]  
TYPE 130: \*MAT\_SPECIAL\_ORTHOTROPIC [2]

# **\*MAT**

---

For the discrete springs and dampers eight materials are available:

- \*MAT\_SPRING\_ELASTIC**
- \*MAT\_DAMPER\_VISCOUS**
- \*MAT\_SPRING\_ELASTOPLASTIC**
- \*MAT\_SPRING\_NONLINEAR\_ELASTIC**
- \*MAT\_DAMPER\_NONLINEAR\_VISCOUS**
- \*MAT\_SPRING\_GENERAL\_NONLINEAR**
- \*MAT\_SPRING\_MAXWELL**
- \*MAT\_SPRING\_INELASTIC**
- \*MAT\_SPRING\_MUSCLE**

For the seatbelts one material is available:

- \*MAT\_SEATBELT**

For thermal materials in a coupled structural/thermal or thermal only analysis six materials are available. These materials are related to the structural material via the \*PART card. Thermal materials are defined only for solid and shell elements.

- \*MAT\_THERMAL\_ISOTROPIC**
- \*MAT\_THERMAL\_ORTHOTROPIC**
- \*MAT\_THERMAL\_ISOTROPIC\_TD**
- \*MAT\_THERMAL\_ORTHOTROPIC\_TD**
- \*MAT\_THERMAL\_ISOTROPIC\_PHASE\_CHANGE**
- \*MAT\_THERMAL\_ISOTROPIC\_TD\_LC**

In the table below, a list of the available material models and the applicable element types are given. Some materials include strain rate sensitivity, failure, equations of state, and thermal effects and this is also noted. General applicability of the materials to certain kinds of behavior is suggested in the last column.

Material number	Material Title	Bricks	Beams	Thin Shells	Thick Shells	Strain-Rate Effects	Failure	Equation-of-State	Thermal Effects	Notes:
										Gn Cm Cr Fl Fm Gl Hy Mt Pl Rb Sl
1	Elastic	Y	Y	Y	Y					Gn, Fl
2	Orthotropic Elastic (Anisotropic - solids)	Y		Y	Y					Cm, Mt
3	Plastic Kinematic/Isotropic	Y	Y	Y	Y	Y	Y			Cm, Mt, Pl
4	Elastic Plastic Thermal	Y	Y	Y	Y				Y	Mt, Pl
5	Soil and Foam	Y								Fm, Sl
6	Linear Viscoelastic	Y	Y	Y		Y				Rb
7	Blatz-Ko Rubber	Y		Y						Rb, Polyurethane
8	High Explosive Burn	Y							Y	Hy
9	Null Material	Y					Y	Y	Y	Fl, Hy
10	Elastic Plastic Hydro(dynamic)	Y					Y	Y		Hy, Mt
11	Steinberg: Temp. Dependent Elastoplastic	Y				Y	Y	Y	Y	Hy, Mt
12	Isotropic Elastic Plastic	Y		Y	Y					Mt
13	Isotropic Elastic Plastic with Failure	Y					Y			Mt
14	Soil and Foam with Failure	Y					Y			Fm, Sl
15	Johnson/Cook Plasticity Model	Y		Y		Y	Y	Y	Y	Hy, Mt
16	Pseudo TENSOR Geological Model	Y				Y	Y	Y		Sl
17	Oriented Crack (Elastoplastic with Fracture)	Y					Y	Y		Hy, Mt, Pl
18	Power Law Plasticity (Isotropic)	Y	Y	Y	Y	Y				Mt, Pl
19	Strain Rate Dependent Plasticity	Y		Y	Y	Y	Y			Mt, Pl
20	Rigid	Y	Y	Y	Y					

# \*MAT

Material number	Material Title								Notes: Gn General Cm Composites Cr Ceramics Fl Fluids Fm Foam Gl Glass Hy Hydro-dyn Mt Metal Pl Plastic Rb Rubber Sl Soil/Conc
		Bricks	Beams	Thin Shells	Thick Shells	Strain-Rate Effects	Failure	Equation-of-State	
21	Orthotropic Thermal (Elastic)	Y	Y	Y			Y	Gn	
22	Composite Damage	Y	Y	Y		Y		Cm	
23	Temperature Dependent Orthotropic	Y	Y	Y			Y	Cm	
24	Piecewise Linear Plasticity (Isotropic)	Y	Y	Y	Y	Y		Mt, Pl	
25	Inviscid Two Invariant Geologic Cap	Y						Sl	
26	Honeycomb	Y			Y	Y		Cm, Fm, Sl	
27	Mooney-Rivlin Rubber	Y	Y					Rb	
28	Resultant Plasticity		Y	Y				Mt	
29	Force Limited Resultant Formulation		Y						
30	Closed Form Update Shell Plasticity			Y	Y			Mt	
31	Frazer-Nash Rubber	Y						Rb	
32	Laminated Glass (Composite)			Y	Y	Y		Cm, Gl	
33	Barlet Anisotropic Plasticity	Y	Y	Y				Cr, Mt	
34	Fabric		Y						
35	Plastic Green-Naghdi Rate	Y			Y			Mt	
36	3-Parameter Barlat Plasticity			Y				Mt	
37	Transversely Anisotropic Elastic Plastic			Y	Y			Mt	
38	Blatz-Ko Foam	Y	Y					Fm, Pl	
39	FLD Transversely Anisotropic			Y	Y			Mt	
40	Nonlinear Orthotropic			Y		Y	Y	Cm	
41-50	User Defined Materials	Y	Y	Y	Y	Y	Y	Gn	



Material number	Material Title	Bricks	Beams	Thin Shells	Thick Shells	Strain-Rate Effects	Failure	Equation-of-State	Thermal Effects	Notes:
										Gn General Cm Composites Cr Ceramics Fl Fluids Fm Foam G1 Glass Hy Hydro-dyn Mt Metal Pl Plastic Rb Rubber S1 Soil/Conc
51	Bamman (Temp /Rate Dependent Plasticity)	Y	Y	Y		Y		Y		Gn
52	Bamman Damage	Y	Y	Y		Y	Y	Y		Mt
53	Closed Cell Foam (Low Density Polyurethane)	Y								Fm
54	Composite Damage with Chang Failure			Y			Y			Cm
55	Composite Damage with Tsai-Wu Failure			Y			Y			Cm
56										
57	Low Density Urethane Foam	Y				Y	Y			Fm
58										
59	Composite Failure (Plasticity Based)	Y	Y				Y			Cm, Cr
60	Elastic with Viscosity (Viscous Glass)	Y	Y			Y		Y		G1
61	Kelvin-Maxwell Viscoelastic	Y				Y				Fm
62	Viscous Foam (Crash Dummy Foam)	Y				Y				Fm
63	Isotropic Crushable Foam	Y				Y				Fm
64	Rate Sensitive Powerlaw Plasticity	Y	Y	Y		Y				Mt
65	Zerilli-Armstrong (Rate/Temp Plasticity)	Y	Y			Y		Y	Y	Mt
66	Linear Elastic Discrete Beam		Y			Y				
67	Nonlinear Elastic Discrete Beam		Y			Y				
68	Nonlinear Plastic Discrete Beam		Y			Y	Y			
69	SID Damper Discrete Beam		Y			Y				
70	Hydraulic Gas Damper Discrete Beam		Y			Y				

**\*MAT**

Material number	Material Title	Bricks	Beams	Thin Shells	Thick Shells	Strain-Rate Effects	Failure	Equation-of-State	Thermal Effects	Notes:
										Gn General Cm Composites Cr Ceramics Fl Fluids Fm Foam Gl Glass Hy Hydro-dyn Mt Metal Pl Plastic Rb Rubber Sl Soil/Conc
71	Cable Discrete Beam (Elastic)		Y							
72	Concrete Damage	Y				Y	Y	Y		Sl
73	Low Density Viscous Foam	Y				Y	Y			Fm
74										
75	Bilkhu/Dubois Foam (Isotropic)	Y				Y				Fm
76	General Viscoelastic (Maxwell Model)	Y				Y				Rb
77	Hyperelastic and Ogden Rubber	Y								Rb
78	Soil Concrete	Y					Y			Sl
79	Hysteretic Soil (Elasto-Perfectly Plastic)	Y					Y			Sl
80										
81	Plasticity with Damage (Elasto-Plastic)			Y	Y	Y	Y			Mt, Pl
82										
83	Fu Chang Foam	Y				Y	Y			Fm
84										
85										
86	Orthotropic Viscoelastic			Y		Y				Rb
87	Cellular Rubber	Y				Y				Rb
88	MTS	Y		Y		Y		Y		Mt
89	Plasticity/Polymer			Y						
90	Acoustic	Y								Fl
96	Brittle Damage	Y				Y	Y			

Material number	Material Title					Strain-Rate Effects	Failure	Equation-of-State	Thermal Effects	Notes: Gn General Cm Composites Cr Ceramics Fl Fluids Fm Foam Gl Glass Hy Hydro-dyn Mt Metal Pl Plastic Rb Rubber Sl Soil/Conc
		Bricks	Beams	Thin Shells	Thick Shells					
DS1	Spring Elastic (Linear)		Y							
DS2	Damper Viscous (Linear)		Y		Y					
DS3	Spring Elastoplastic (Isotropic)		Y							
DS4	Spring Nonlinear Elastic		Y		Y					
DS5	Damper Nonlinear Viscous		Y		Y					
DS6	Spring General Nonlinear		Y							
DS7	Spring Maxwell (Three Parameter Viscoelastic)		Y		Y					
DS8	Spring Inelastic (Tension or Compression)		Y							
SB1	Seatbelt									
TH1	Thermal Isotropic	Y	Y					Y		
TH2	Thermal Orthotropic	Y	Y					Y		
TH3	Thermal Isotropic (Temp. Dependent)	Y	Y					Y		
TH4	Thermal Orthotropic (Temp. Dependent)	Y	Y					Y		
5										
6										
7										
8										
TH9	Thermal Isotropic (Phase Change)	Y	Y					Y		
TH10	Thermal Isotropic (Temp Dep - Load Curve)	Y	Y					Y		

**\*MAT\_ADD\_EROSION**

Many of the constitutive models in LS-DYNA do not allow failure and erosion. The ADD\_EROSION option provides a way of including failure in these models although the option can also be applied to constitutive models with other failure/erosion criterion. Each of the criterion defined here are applied independently, and once any one of them is satisfied, the element is deleted from the calculation. NOTE: THIS OPTION CURRENTLY APPLIES TO THE 2D AND 3D SOLID ELEMENTS WITH ONE POINT INTEGRATION.

**Define the following two cards:**

Card 1            1            2            3            4            5            6            7            8

Variable	MID	EXCL						
Type	I	F						
Default	none	none						

Card 2

Variable	PFAIL	SIGP1	SIGVM	EPSP1	EPSSH	SIGTH	IMPULSE	
Type	F	F	F	F	F	F	F	
Default	none	none	none	none	none	none	none	

---

<b>VARIABLE</b>	<b>DESCRIPTION</b>
-----------------	--------------------

---

MID            Material identification for which this erosion definition applies

EXCL            The exclusion number. 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.

VARIABLE	DESCRIPTION
PFAIL	Pressure at failure, $P_{\min}$ .
SIGP1	Principal stress at failure, $\sigma_{\max}$ .
SIGVM	Equivalent stress at failure, $\bar{\sigma}_{\max}$ .
EPSP1	Principal strain at failure, $\varepsilon_{\max}$ .
EPSSH	Shear strain at failure, $\gamma_{\max}$ .
SIGTH	Threshold stress, $\sigma_0$ .
IMPULSE	Stress impulse for failure, $K_f$ .

The criteria for failure are:

1.  $P \leq P_{\min}$  where  $P$  is the pressure (positive in compression), and  $P_{\min}$  is the pressure at failure.
2.  $\sigma_1 \geq \sigma_{\max}$ , where  $\sigma_1$  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  $\sigma'_{ij}$  are the deviatoric stress components, and  $\bar{\sigma}_{\max}$  is the equivalent stress at failure.
4.  $\varepsilon_1 \geq \varepsilon_{\max}$ , where  $\varepsilon_1$  is the maximum principal strain, and  $\varepsilon_{\max}$  is the principal strain at failure.
5.  $\gamma_1 \geq \gamma_{\max}$ , where  $\gamma_1$  is the shear strain, and  $\gamma_{\max}$  is the shear strain at failure.
6. The Tuler-Butcher criterion,

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

where  $\sigma_1$  is the maximum principal stress,  $\sigma_0$  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.

These failure models apply only to solid elements with one point integration in 2 and 3 dimensions.

**\*MAT\_ELASTIC\_{OPTION}**

This is Material Type 1. This is an isotropic elastic material and is available for beam, shell, and solid elements in LS-DYNA. A specialization of this material allows the modeling of fluids.

Options include:

**<BLANK>**

**FLUID**

such that the keyword cards appear:

**\*MAT\_ELASTIC**

**\*MAT\_ELASTIC\_FLUID**

The fluid option is valid for solid elements only.

**Define the following card for all options:**

**Card Format**

	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	DA	DB	K	
Type	I	F	F	F	F	F	F	
Default	none	none	none	none	0.0	0.0	0.0	

Define the following extra card for the FLUID option:

**Card Format**

	1	2	3	4	5	6	7	8
Variable	VC	CP						
Type	F	F						
Default	none	1.0E+20						

**VARIABLE**

**DESCRIPTION**

MID	Material identification. A unique number has to be chosen.
RO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.
DA	Axial damping factor (used for Belytschko-Schwer beam, type 1, only).
DB	Bending damping factor (used for Belytschko-Schwer beam, type 1, only).
K	Bulk Modulus (define for fluid option only).
VC	Tensor viscosity coefficient, values between .1 and .5 should be okay.
CP	Cavitation pressure (default = 1.0e+20).

**Remarks:**

The axial and bending damping factors are used to damp down numerical noise. The update of the force resultants,  $F_i$ , and moment resultants,  $M_i$ , includes the damping factors:

$$F_i^{n+1} = F_i^n + \left(1 + \frac{DA}{\Delta t}\right) \Delta F_i^{n+\frac{1}{2}}$$

$$M_i^{n+1} = M_i^n + \left(1 + \frac{DB}{\Delta t}\right) \Delta M_i^{n+\frac{1}{2}}$$

For the fluid option the bulk modulus (K) has to be defined as Young's modulus, and Poisson's ratio are ignored. With the fluid option fluid-like behavior is obtained where the bulk modulus, K, and pressure rate, p, are given by:

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

$$\dot{p} = -K \dot{\epsilon}_{ii}$$

and the shear modulus is set to zero. A tensor viscosity is used which acts only the deviatoric stresses,  $S_{ij}^{n+1}$ , given in terms of the damping coefficient as:

$$S_{ij}^{n+1} = VC \cdot \Delta L \cdot a \cdot \rho \dot{\epsilon}_{ij}'$$

where  $p$ , is a characteristic element length,  $a$  is the fluid bulk sound speed,  $\rho$  is the fluid density, and  $\dot{\epsilon}_{ij}'$  is the deviatoric strain rate.



\*MAT\_OPTION TROPIC\_ELASTIC

This is Material Type 2. This material is valid for modeling the elastic-orthotropic behavior of solids, shells, and thick shells. An anisotropic option is available for solid elements.

Options include:

**ORTHO**  
**ANISO**

such that the keyword cards appear:

\*MAT\_ORTHOTROPIC\_ELASTIC (4 cards follow)

\*MAT\_ANISOTROPIC\_ELASTIC (5 cards follow)

**Card Format of Cards 1 and 2 for the ORTHO option.**

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	EA	EB	EC	PRBA	PRCA	PRCB
Type	I	F	F	F	F	F	F	F

Card 2

Variable	GAB	GBC	GCA	AOPT				
Type	F	F	F	F				

**Card Format of Cards 1, 2, and 3 for the ANISO option.**

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	C11	C12	C22	C13	C23	C33
Type	I	F	F	F	F	F	F	F

Card 2

Variable	C14	C24	C34	C44	C15	C25	C35	C45
Type	F	F	F	F	F	F	F	F

Card 3

Variable	C55	C16	C26	C36	C46	C56	C66	AOPT
Type	F	F	F	F	F	F	F	F

**Card Format of Cards 3/4 and 4/5 for the ORTHO/ANISO options.**

Card 3/4

Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card 4/5

Variable	V1	V2	V3	D1	D2	D3	BETA	REF
Type	F	F	F	F	F	F	F	F



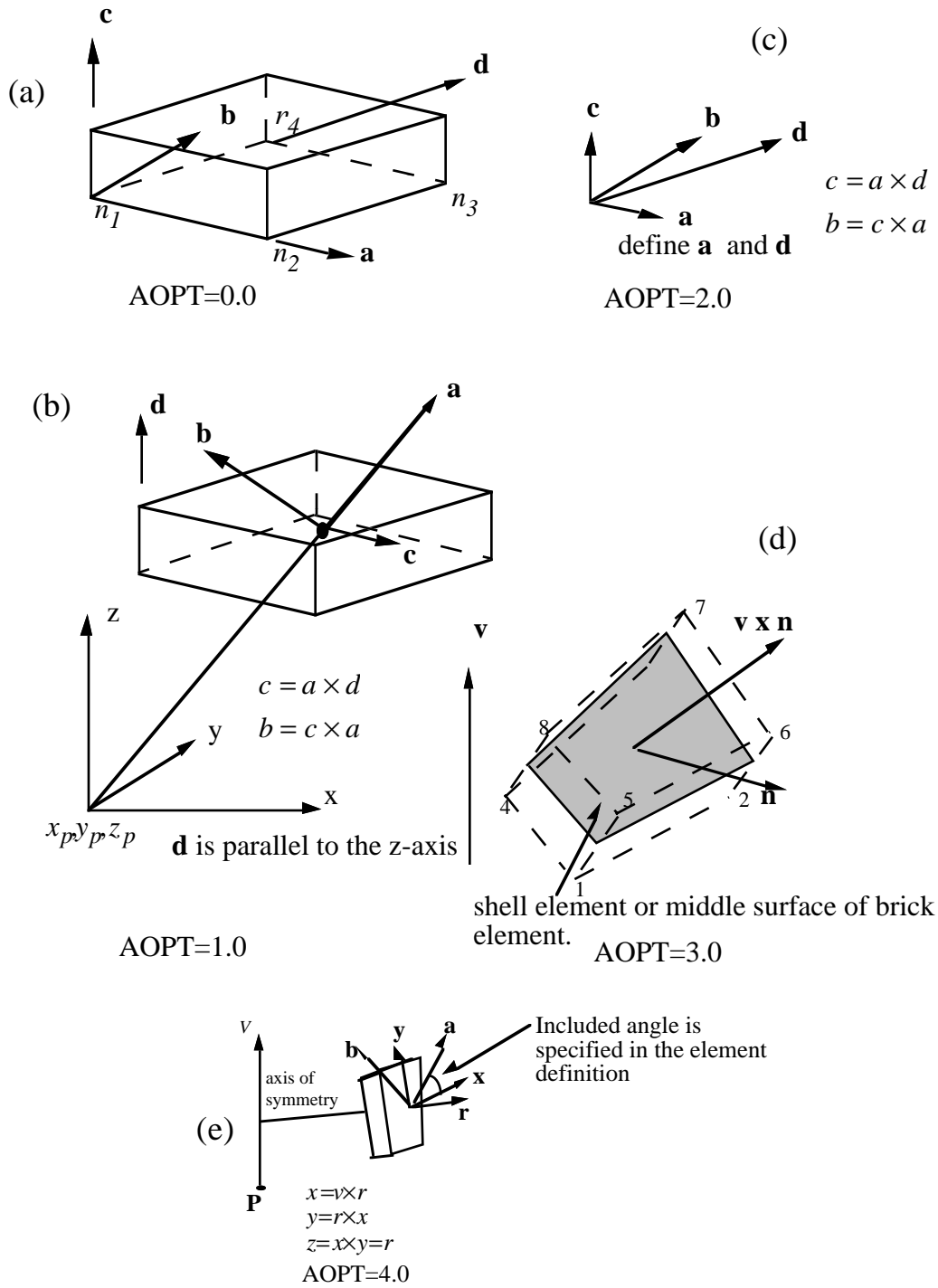
<u>VARIABLE</u>	<u>DESCRIPTION</u>
	EQ. 0.0: locally orthotropic with material axes determined by element nodes as shown in Figure 20.1. Nodes 1, 2, and 4 of an element are identical to the Nodes used for the definition of a coordinate system as by *DEFINE_COORDINATE_NODES.
	EQ. 1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center; this is the a-direction.
	EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR.
	EQ. 3.0: applicable to shell elements only. This option determines locally orthotropic material axes by offsetting the material axes by an angle to be specified from a line in the plane of the shell determined by taking the cross product of the vector <b>v</b> defined below with the shell normal vector.
	EQ. 4.0: locally orthotropic in cylindrical coordinate system with the material axes determined by a vector <b>v</b> and an originating point, P. They define the axis of symmetry.
XP,YP,ZP	x <sub>p</sub> y <sub>p</sub> z <sub>p</sub> , define coordinates of point <b>p</b> for AOPT = 1 and 4.
A1,A2,A3	a <sub>1</sub> a <sub>2</sub> a <sub>3</sub> , define components of vector <b>a</b> for AOPT = 2.
D1,D2,D3	d <sub>1</sub> d <sub>2</sub> d <sub>3</sub> , define components of vector <b>d</b> for AOPT = 2.
V1,V2,V3	v <sub>1</sub> v <sub>2</sub> v <sub>3</sub> , define components of vector <b>v</b> for AOPT = 3 and 4.
BETA	Material angle in degrees for AOPT = 3, which may be overridden on the element card, see *ELEMENT_SHELL.
REF	Use reference geometry to initialize the stress tensor. The reference geometriy is defined by the keyword:*INITIAL_FOAM_REFERENCE_GEOMETRY. This option is currently restricted to 8-noded solid elements with one point integration. EQ.0.0: off, EQ.1.0: on.

**Remarks:**

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

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

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



**Figure 20.1.** Options for determining principal material axes: (a)  $AOPT = 0.0$ , (b)  $AOPT = 1.0$ , (c)  $AOPT = 2.0$ , (d)  $AOPT = 3.0$ , and (e)  $AOPT=4.0$  for brick elements.

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

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

\*MAT\_PLASTIC\_KINEMATIC

This is Material Type 3. This model is suited to model isotropic and kinematic hardening plasticity with the option of including rate effects. It is a very cost effective model and is available for beam (Hughes-Liu), shell, and solid elements.

Card Format

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	E	PR	SIGY	ETAN	BETA	
Type	I	F	F	F	F	F	F	
Default	none	none	none	none	none	0.0	0.0	

Card 2

Variable	SRC	SRP	FS	VP				
Type	F	F	F	F				
Default	not used	not used	not used	0.0				

VARIABLE

DESCRIPTION

MID	Material identification. A unique number has to be chosen.
RO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.
SIGY	Yield stress.
ETAN	Tangent modulus, see Figure 20.2.
BETA	Hardening parameter, $0 < \beta' < 1$ . See comments below.

---

<b>VARIABLE</b>	<b>DESCRIPTION</b>
SRC	Strain rate parameter, C, for Cowper Symonds strain rate model, see below. If zero, rate effects are not considered.
SRP	Strain rate parameter, P, for Cowper Symonds strain rate model, see below. If zero, rate effects are not considered.
FS	Failure strain for eroding elements.
VP	Formulation for rate effects: EQ.0.0: Scale yield stress (default), EQ.1.0: Viscoplastic formulation

**Remarks:**

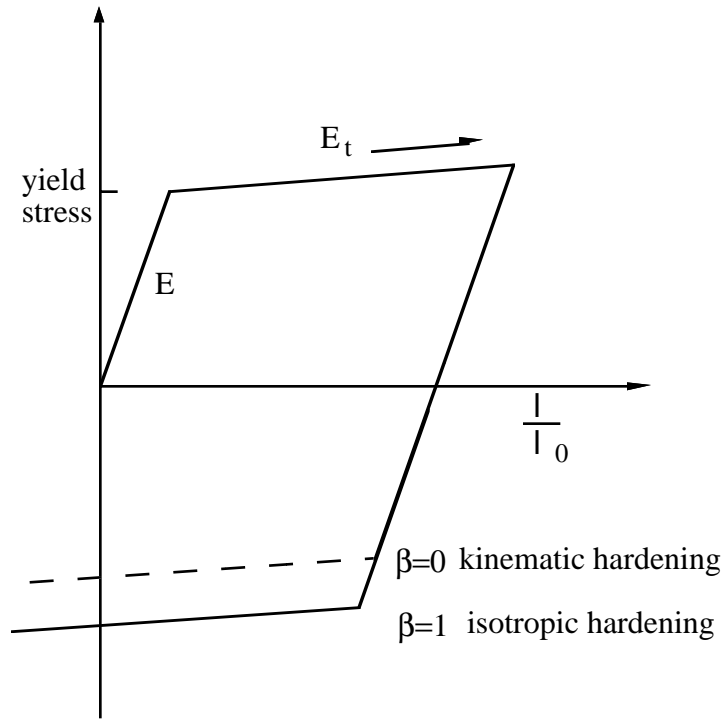
Strain rate is accounted for using the Cowper and Symonds model which scales the yield stress with the factor

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

where  $\dot{\epsilon}$  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 in results can be dramatic. To ignore strain rate effects set both SRC and SRP to zero.

Kinematic, isotropic, or a combination of kinematic and isotropic hardening may be specified by varying  $\beta'$  between 0 and 1. For  $\beta'$  equal to 0 and 1, respectively, kinematic and isotropic hardening are obtained as shown in Figure 20.2. For isotropic hardening,  $\beta' = 1$ , Material Model 12, \*MAT\_ISOTROPIC\_ELASTIC\_PLASTIC, requires less storage and is more efficient. Whenever possible, Material 12 is recommended for solid elements, but for shell elements it is less accurate and thus material 12 is not recommended in this case.





**Figure 20.2.** Elastic-plastic behavior with kinematic and isotropic hardening where  $l_0$  and  $l$  are undeformed and deformed lengths of uniaxial tension specimen.  $E_t$  is the slope of the bilinear stress strain curve.

**\*MAT\_ELASTIC\_PLASTIC\_THERMAL**

This is Material Type 4. Temperature dependent material coefficients can be defined. A maximum of eight temperatures with the corresponding data can be defined. A minimum of two points is needed. When this material type is used it is necessary to define nodal temperatures by activating a coupled analysis or by using another option to define the temperatures such as \*LOAD\_THERMAL\_LOAD\_CURVE, or \*LOAD\_THERMAL\_VARIABLE.

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO						
Type	I	F						

Card 2

Variable	T1	T2	T3	T4	T5	T6	T7	T8
Type	F	F	F	F	F	F	F	F

Card 3

Variable	E1	E2	E3	E4	E5	E6	E7	E8
Type	F	F	F	F	F	F	F	F

Card 4

Variable	PR1	PR2	PR3	PR4	PR5	PR6	PR7	PR8
Type	F	F	F	F	F	F	F	F

Card Format (no defaults are assumed)

Card 5

Variable	ALPHA1	ALPHA2	ALPHA3	ALPHA4	ALPHA5	ALPHA6	ALPHA7	ALPHA8
Type	F	F	F	F	F	F	F	F

Card 6            1            2            3            4            5            6            7            8

Variable	SIGY1	SIGY2	SIGY3	SIGY4	SIGY5	SIGY6	SIGY7	SIGY8
Type	F	F	F	F	F	F	F	F

Card 7

Variable	ETAN1	ETAN2	ETAN3	ETAN4	ETAN5	ETAN6	ETAN7	ETAN8
Type	F	F	F	F	F	F	F	F

VARIABLE

DESCRIPTION

- MID            Material identification. A unique number must be chosen.
- RO            Mass density.
- TI            Temperatures. The minimum is 2, the maximum is 8.
- EI            Corresponding Young's moduli at temperature TI.
- PRI           Corresponding Poisson's ratios.
- ALPHA1      Corresponding coefficients of thermal expansion.
- SIGY1       Corresponding yield stresses.
- ETAN1       Corresponding plastic hardening moduli.

**Remarks:**

At least two temperatures and their corresponding material properties must be defined. The analysis will be terminated if a material temperature falls outside the range defined in the input. If a thermoelastic material is considered, do not define SIGY and ETAN. The coefficient of thermal expansion is defined as the instantaneous value. Thus, the thermal strain rate becomes:

$$\dot{\epsilon}_{ij}^T = \alpha \dot{T} \delta_{ij}$$

\*MAT\_SOIL\_AND\_FOAM

This is Material Type 5. This is a very simple model and works in some ways like a fluid. It should be used only in situations when soils and foams are confined within a structure or when geometric boundaries are present.

Card Format

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	G	BULK	A0	A1	A2	PC
Type	I	F	F	F	F	F	F	F

Card 2

Variable	VCR	REF						
Type	F	F						

Card 3

Variable	EPS1	EPS2	EPS3	EPS4	EPS5	EPS6	EPS7	EPS8
Type	F	F	F	F	F	F	F	F

Card 4

Variable	EPS9	EPS10						
Type	F	F						

Card 5

Variable	P1	P2	P3	P4	P5	P6	P7	P8
Type	F	F	F	F	F	F	F	F

Card 6

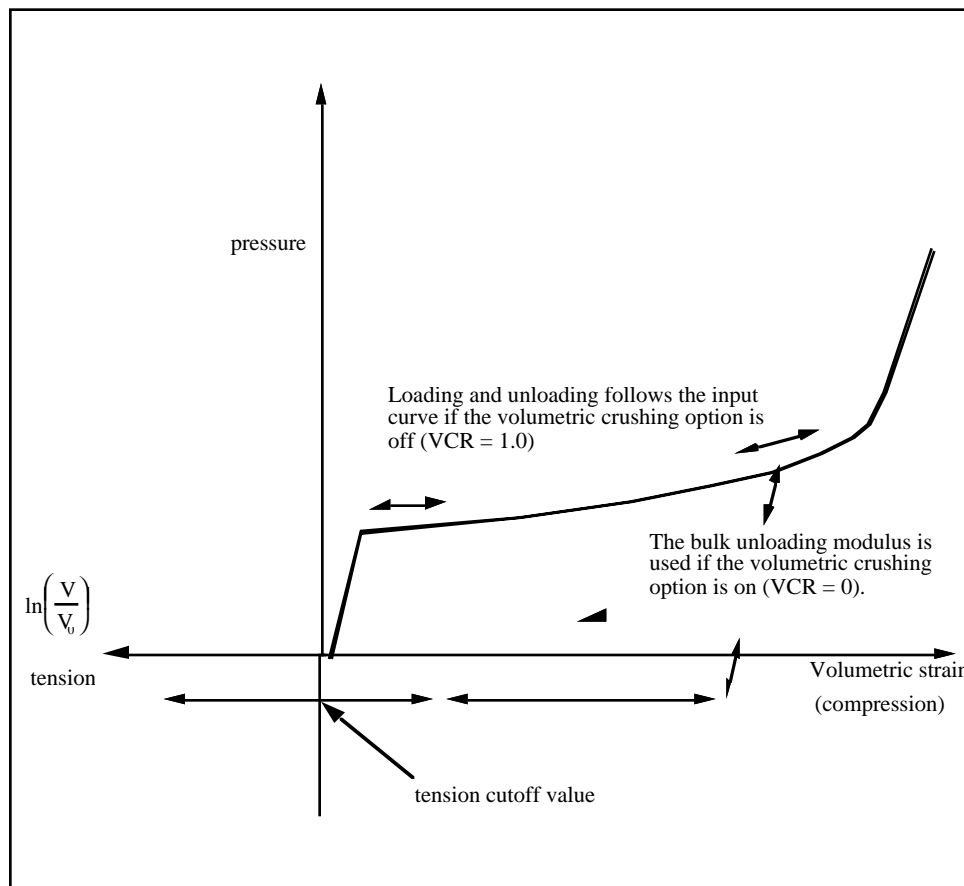
Variable	P9	P10						
Type	F	F						

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
G	Shear modulus.
K	Bulk modulus for unloading used for VCR=0.0.
A0	Yield function constant for plastic yield function below.
A1	Yield function constant for plastic yield function below.
A2	Yield function constant for plastic yield function below.
PC	Pressure cutoff for tensile fracture.
VCR	Volumetric crushing option: EQ.0.0: on, EQ.1.0: loading and unloading paths are the same.
REF	Use reference geometry to initialize the pressure. The reference geometry is defined by the keyword: *INITIAL_FOAM_REFERENCE_GEOMETRY. This option does not initialize the deviatoric stress state. EQ.0.0: off, EQ.1.0: on.

VARIABLE	DESCRIPTION
EPS1,.....	Volumetric strain values (natural logarithmic values), see comments below. A maximum of 10 values are allowed and a minimum of 2 values are necessary. The tabulated values must completely cover the expected values in the analysis. If the first value is not for a volumetric strain value of zero then the point (0.0,0.0) will be automatically generated and upto a further nine additional values may be defined.
P1, P2,..PN	Pressures corresponding to volumetric strain values.

**Remarks:**

Pressure is positive in compression. Volumetric strain is given by the natural log of the relative volume and is negative in compression. Relative volume is ratio of the current volume to the initial volume at the start of the calculation. The tabulated data should be given in order of increasing compression. If the pressure drops below the cutoff value specified, it is reset to that value. For a detailed description we refer to Kreig [1972].



**Figure 20.3.** 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,  $\phi$ , 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:

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

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

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

There is no strain hardening on this surface.

To eliminate the pressure dependence of the yield strength, set:

$$a_1 = a_2 = 0 \quad a_0 = \frac{1}{3} \sigma_y^2 .$$

This approach is useful when a von Mises type elastic-plastic model is desired for use with the tabulated volumetric data



\*MAT\_VISCOELASTIC

This is Material Type 6. This model allows the modeling of viscoelastic behavior for beams (Hughes-Liu), shells, and solids. Also see \*MAT\_GENERAL\_VISCOELASTIC for a more general formulation.

Card Format

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	BULK	G0	GI	BETA		
Type	I	F	F	F	F	F		

VARIABLE

DESCRIPTION

- MID            Material identification. A unique number has to be chosen.
- RO            Mass density
- BULK         Elastic bulk modulus.
- G0            Short-time shear modulus, see equations below.
- GI            Long-time (infinite) shear modulus,  $G_{\infty}$ .
- BETA         Decay constant.

**Remarks:**

The shear relaxation behavior is described by [Hermann and Peterson, 1968]:

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

A Jaumann rate formulation is used

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

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

**\*MAT\_BLATZ-KO\_RUBBER**

This is Material Type 7. This one parameter material allows the modeling of nearly incompressible continuum rubber. The Poisson’s ratio is fixed to 0.463.

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	G	REF				
Type	I	F	F	F				

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
G	Shear modulus.
REF	Use reference geometry to initialize the stress tensor. The reference geometry is defined by the keyword:*INITIAL_FOAM_REFERENCE_GEOMETRY. This option is currently restricted to 8-noded solid elements with one point integration. EQ.0.0: off, EQ.1.0: on.

**Remarks:**

The second Piola-Kirchhoff stress is computed as

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

where  $V$  is the relative volume defined as being the ratio of the current volume to the initial volume,  $C_{ij}$  is the right Cauchy-Green strain tensor, and  $\nu$  is Poisson’s ratio, which is set to .463 internally. This stress measure is transformed to the Cauchy stress,  $\sigma_{ij}$ , according to the relationship

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

where  $F_{ij}$  is the deformation gradient tensor. Also see Blatz and Ko [1962].

\*MAT\_HIGH\_EXPLOSIVE\_BURN

This is Material Type 8. It allows the modeling of the detonation of a high explosive. In addition an equation of state must be defined. See Wilkins [1969] and Giroux [1972].

Card Format

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	D	PCJ	BETA	K	G	SIGY
Type	I	F	F	F	F	F	F	F

VARIABLE

DESCRIPTION

MID	Material identification. A unique number has to be chosen.
RO	Mass density.
D	Detonation velocity.
PCJ	Chapman-Jouget pressure.
BETA	Beta burn flag, BETA (see comments below): EQ.0.0: beta + programmed burn, EQ.1.0: beta burn only, EQ.2.0: programmed burn only.
K	Bulk modulus (BETA=2.0 only).
G	Shear modulus (BETA=2.0 only).
SIGY	$\sigma_y$ , yield stress (BETA=2.0 only).

**Remarks:**

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

$$p = F p_{eos}(V, E)$$

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

$$F_1 = \begin{cases} \frac{2(t - t_l)DA_{e_{\max}}}{3v_e} & \text{if } t > t_l \\ 0 & \text{if } t \leq t_l \end{cases}$$

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

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

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

$$F = F_2$$

and  $F_1$  is not computed.

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

As an option, the high explosive material can behave as an elastic perfectly-plastic solid prior to detonation. In this case we update the stress tensor, to an elastic trial stress,  $*s_{ij}^{n+1}$ ,

$$*s_{ij}^{n+1} = s_{ij}^n + s_{ip}\Omega_{pj} + s_{jp}\Omega_{pi} + 2G\dot{\epsilon}'_{ij} dt$$

where  $G$  is the shear modulus, and  $\dot{\epsilon}'_{ij}$  is the deviatoric strain rate. The von Mises yield condition is given by:

$$\phi = J_2 - \frac{\sigma_y^2}{3}$$

where the second stress invariant,  $J_2$ , is defined in terms of the deviatoric stress components as

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

and the yield stress is  $\sigma_y$ . If yielding has occurred, i.e.,  $\phi > 0$ , the deviatoric trial stress is scaled to obtain the final deviatoric stress at time n+1:

$$s_{ij}^{n+1} = \frac{\sigma_y}{\sqrt{3J_2}} * s_{ij}^{n+1}$$

If  $\phi \leq 0$ , then

$$s_{ij}^{n+1} = *s_{ij}^{n+1}$$

Before detonation pressure is given by the expression

$$p^{n+1} = K \left( \frac{1}{V^{n+1}} - 1 \right)$$

where K is the bulk modulus. Once the explosive material detonates:

$$s_{ij}^{n+1} = 0$$

and the material behaves like a gas.

**\*MAT\_NULL**

This is Material Type 9. This material allows equations of state to be considered without computing deviatoric stresses. Optionally, a viscosity can be defined. Also, erosion in tension and compression is possible.

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

Beams and shells that use this material type are completely bypassed in the element processing; however, the mass of the null shell elements is computed and added to the nodal points which define the connectivity, 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.

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	PC	MU	TEROD	CEROD	YM	PR
Type	I	F	F	F	F	F	F	F
Defaults	none	none	0.0	0.0	0.0	0.0	0.0	0.0

**VARIABLE****DESCRIPTION**

MID	Material identification. A unique number has to be chosen.
RO	Mass density
PC	Pressure cutoff ( $\leq 0.0$ ).
MU	Viscosity coefficient $\mu$ (optional).
TEROD	Relative volume. $\frac{V}{V_0}$ , for erosion in tension. Typically, use values greater than unity. If zero, erosion in tension is inactive.

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
CEROD	Relative volume, $\frac{V}{V_0}$ , for erosion in compression. Typically, use values less than unity. If zero, erosion in compression is inactive.
YM	Young's modulus (used for null beams and shells only)
PR	Poisson's ratio (used for null beams and shells only)

**Remarks:**

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

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

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

# \*MAT

\*MAT\_ELASTIC\_PLASTIC\_HYDRO\_{OPTION}

---

\*MAT\_ELASTIC\_PLASTIC\_HYDRO\_{OPTION}

Available options are:

<BLANK>

SPALL

This is Material Type 10. This material allows the modeling of an elastic-plastic hydrodynamic material.

## Card Format

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	G	SIGY	EH	PC	FS	
Type	I	F	F	F	F	F	F	
Default	none	none	none	0.0	0.0	$-\infty$	0.0	

**Define this card if and only if the SPALL option is specified.**

Optional

Variable	A1	A2	SPALL					
Type	F	F	F					

Card 2

Variable	EPS1	EPS2	EPS3	EPS4	EPS5	EPS6	EPS7	EPS8
Type	F	F	F	F	F	F	F	F



Card 3            1            2            3            4            5            6            7            8

Variable	EPS9	EPS10	EPS11	EPS12	EPS13	EPS14	EPS15	EPS16
Type	F	F	F	F	F	F	F	F

Card 4

Variable	ES1	ES2	ES3	ES4	ES5	ES6	ES7	ES8
Type	F	F	F	F	F	F	F	F

Card 5

Variable	ES9	ES10	ES11	ES12	ES13	ES14	ES15	ES16
Type	F	F	F	F	F	F	F	F

**VARIABLE**

**DESCRIPTION**

- MID            Material identification. A unique number has to be chosen.
- RO            Mass density.
- G             Shear modulus.
- SIGY         Yield stress, see comment below.
- EH            Plastic hardening modulus, see definition below.
- PC            Pressure cutoff ( $\leq 0.0$ ). If zero, a cutoff of  $-\infty$  is assumed.
- FS            Failure strain for erosion.
- A1            Linear pressure hardening coefficient.
- A2            Quadratic pressure hardening coefficient.

VARIABLE	DESCRIPTION
SPALL	Spall type: EQ.0.0: default set to "1.0", EQ.1.0: $p \geq p_{min}$ , EQ.2.0: if $\sigma_{max} \geq -p$ min element spalls and tension, $p < 0$ , is never allowed, EQ.3.0: $p < -p_{min}$ element spalls and tension, $p < 0$ , is never allowed.
EPS	Effective plastic strain (True). Define up to 16 values. Care must be taken that the full range of strains expected in the analysis is covered. Linear extrapolation is used if the strain values exceed the maximum input value.
ES	Effective stress. Define up to 16 values.

**Remarks:**

If ES and EPS are undefined, the yield stress and plastic hardening modulus are taken from SIGY and EH. In this case, the bilinear stress-strain curve shown in Figure 20.2. is obtained with hardening parameter,  $\beta = 1$ . The yield strength is calculated as

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

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

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

and  $p$  is the pressure taken as positive in compression.

If ES and EPS are specified, a curve like that shown in Figure 20.4 may be defined. Effective stress is defined in terms of the deviatoric stress tensor,  $s_{ij}$ , as:

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

and effective plastic strain by:

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

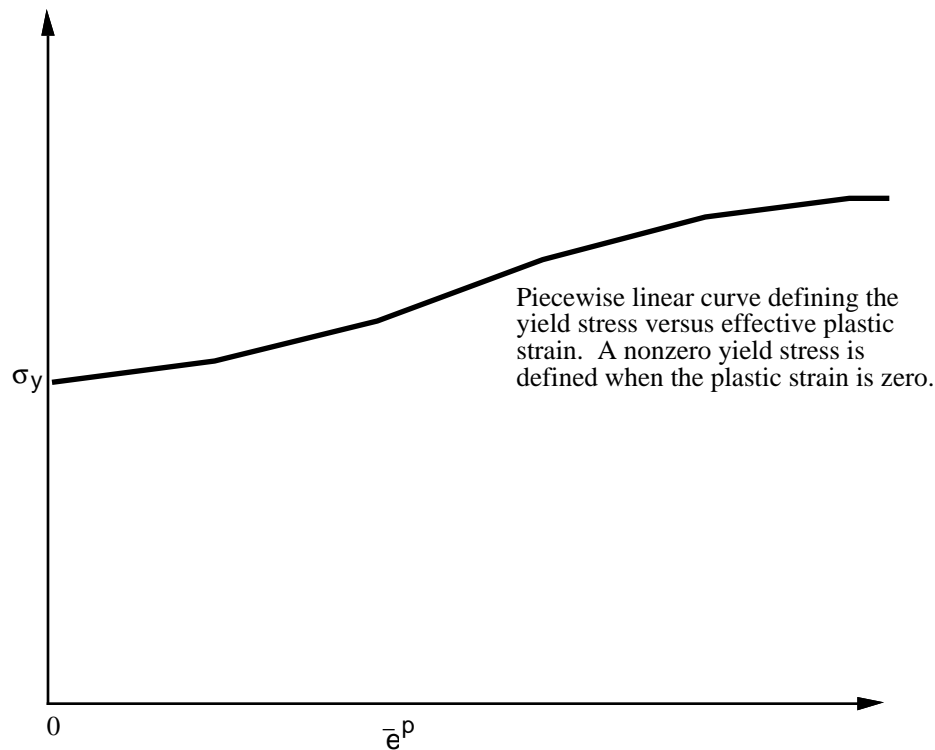
where  $t$  denotes time and  $D_{ij}^p$  is the plastic component of the rate of deformation tensor. In this case the plastic hardening modulus on Card 1 is ignored and the yield stress is given as

$$\sigma_y = f(\bar{\epsilon}^p),$$

where the value for  $f(\bar{\epsilon}^p)$  is found by interpolation from the data curve.

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

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



**Figure 20.4.** Effective stress versus effective plastic strain curve.

**\*MAT\_STEINBERG**

This is Material Type 11. This material is available for modeling materials deforming at very high strain rates ( $>10^5$ ) and can be used with solid elements. The yield strength is a function of temperature and pressure. An equation of state determines the pressure.

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	G0	SIGO	BETA	N	GAMA	SIGM
Type	I	F	F	F	F	F	F	F

Card 2

Variable	B	BP	H	F	A	TMO	GAMO	SA
Type	F	F	F	F	F	F	F	F

Card 3

Variable	PC	SPALL	RP	FLAG	MMN	MMX	ECO	EC1
Type	F	F	F	F	F	F	F	F

Card 4

Variable	EC2	EC3	EC4	EC5	EC6	EC7	EC8	EC9
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
G0	Basic shear modulus.
SIGO	$\sigma_0$ , see defining equations.
BETA	$\beta$ , see defining equations.
N	n, see defining equations.
GAMA	$\gamma_i$ , initial plastic strain, see defining equations.
SIGM	$\sigma_m$ , see defining equations.
B	b, see defining equations.
BP	b', see defining equations.
H	h, see defining equations.
F	f, see defining equations.
A	Atomic weight (if = 0.0, R' must be defined).
TMO	$T_{mo}$ , see defining equations.
GAMO	$\gamma_0$ , see defining equations.
SA	a, see defining equations.
PC	$p_{cut}$ or $-\sigma_f$ (default=-1.e+30)
SPALL	Spall type: EQ. 0.0: default set to "2.0", EQ. 1.0: $p \geq p_{cut}$ , EQ. 2.0: if $\sigma_{max} \geq -p_{cut}$ element spalls and tension, $p < 0$ , is never allowed, EQ. 3.0: $p < -p_{cut}$ element spalls and tension, $p < 0$ , is never allowed.
RP	R'. If R' $\neq$ 0.0, A is not defined.
FLAG	Set to 1.0 for $\mu$ coefficients for the cold compression energy fit. Default is $\eta$ .

VARIABLE	DESCRIPTION
MMN	$\mu_{\min}$ or $\eta_{\min}$ . Optional $\mu$ or $\eta$ minimum value.
MMX	$\mu_{\max}$ or $\eta_{\max}$ . Optional $\mu$ or $\eta$ maximum value.
EC0,...EC9	Cold compression energy coefficients (optional).

**Remarks:**

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

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

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

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

$$E_c(x) = \int_0^x p dx - \frac{900 R' \exp(ax)}{(1-x)^{2(\gamma_0 - a - 1/2)}},$$

$$x = 1 - V,$$

and  $E_m$  is the melting energy:

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

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

$$T_m(x) = \frac{T_{m0} \exp(2ax)}{V^{2(\gamma_0 - a - 1/2)}}$$

and the melting temperature at  $\rho = \rho_0$ ,  $T_{m0}$ .

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  $\sigma_y$  is given by:

$$\sigma_y = \sigma'_0 \left[ 1 + b'pV^{1/3} - h \left( \frac{E_i - E_c}{3R'} - 300 \right) \right] e^{-fE_i/E_m - E_i}$$

if  $E_m$  exceeds  $E_i$ . Here,  $\sigma_0'$  is given by:

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

where  $\gamma_i$  is the initial plastic strain. Whenever  $\sigma_0'$  exceeds  $\sigma_m$ ,  $\sigma_0'$  is set equal to  $\sigma_m$ . After the materials melts,  $\sigma_y$  and  $G$  are set to one half their initial value.

If the coefficients EC0,...,EC9 are not defined above, LS-DYNA will fit the cold compression energy to a ten term polynomial expansion either as a function of  $\mu$  or  $\eta$  depending on the input variable, FLAG, as:

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

$$E_c(\mu^i) = \sum_{i=0}^9 EC_i \mu^i$$

where  $EC_i$  is the  $i$ th coefficient and:

$$\eta = \frac{\rho}{\rho_o}$$

$$\mu = \frac{\rho}{\rho_o} - 1$$

A linear least squares method is used to perform the fit.

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

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

**\*MAT\_STEINBERG\_LUND**

This is Material Type 11. This material is a modification of the Steinberg model above to include the rate model of Steinberg and Lund [1989]. An equation of state determines the pressure.

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	G0	SIGO	BETA	N	GAMA	SIGM
Type	I	F	F	F	F	F	F	F

Card 2

Variable	B	BP	H	F	A	TMO	GAMO	SA
Type	F	F	F	F	F	F	F	F

Card 3

Variable	PC	SPALL	RP	FLAG	MMN	MMX	ECO	EC1
Type	F	F	F	F	F	F	F	F

Card 4

Variable	EC2	EC3	EC4	EC5	EC6	EC7	EC8	EC9
Type	F	F	F	F	F	F	F	F



Card 5

Variable	UK	C1	C2	YP	YA	YM		
Type	F	F	F	F	F	F		

VARIABLEDESCRIPTION

MID	Material identification. A unique number has to be chosen.
RO	Mass density.
G0	Basic shear modulus.
SIGO	$\sigma_o$ , see defining equations.
BETA	$\beta$ , see defining equations.
N	n, see defining equations.
GAMA	$\gamma_i$ , initial plastic strain, see defining equations.
SIGM	$\sigma_m$ , see defining equations.
B	b, see defining equations.
BP	$b'$ , see defining equations.
H	h, see defining equations.
F	f, see defining equations.
A	Atomic weight (if = 0.0, $R'$ must be defined).
TMO	$T_{mo}$ , see defining equations.
GAMO	$\gamma_o$ , see defining equations.
SA	a, see defining equations.
PC	$p_{cut}$ or $-\sigma_f$ (default=-1.e+30)

<b>VARIABLE</b>	<b>DESCRIPTION</b>
SPALL	Spall type: EQ. 0.0: default set to "2.0", EQ. 1.0: $p \geq p_{min}$ , EQ. 2.0: if $\sigma_{max} \geq -p_{min}$ element spalls and tension, $p < 0$ , is never allowed, EQ. 3.0: $p < -p_{min}$ element spalls and tension, $p < 0$ , is never allowed.
RP	$R'$ . If $R' \neq 0.0$ , A is not defined.
FLAG	Set to 1.0 for $\mu$ coefficients for the cold compression energy fit. Default is $\eta$ .
MMN	$\mu_{min}$ or $\eta_{min}$ . Optional $\mu$ or $\eta$ minimum value.
MMX	$\mu_{max}$ or $\eta_{max}$ . Optional $\mu$ or $\eta$ maximum value.
EC0,...EC9	Cold compression energy coefficients (optional).
UK	Activation energy for rate dependent model.
C1	Exponent prefactor in rate dependent model.
C2	Coefficient of drag term in rate dependent model.0
YP	Peierls stress for rate dependent model.
YA	Athermal yield stress for rate dependent model.
YMAX	Work hardening maximum for rate model.

**Remarks:**

This model is similar in theory to the \*MAT\_STEINBERG above but with the addition of rate effects. When rate effects are included, the yield stress is given by:

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

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

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

and the second is on the thermal part:

$$Y_T \leq Y_p$$

\*MAT\_ISOTROPIC\_ELASTIC\_PLASTIC

This is Material Type 12. This is a very low cost isotropic plasticity model for three dimensional solids. In the plane stress implementation for shell elements, a one-step radial return approach is used to scale the Cauchy stress tensor to if the state of stress exceeds the yield surface. This approach to plasticity leads to inaccurate shell thickness updates and stresses after yielding. This is the only model in LS-DYNA for plane stress that does not default to an iterative approach.

Card Format

	1	2	3	4	5	6	7	8
Variable	MID	RO	G	SIGY	ETAN	BULK		
Type	I	F	F	F	F	F		

VARIABLE

DESCRIPTION

MID	Material identification. A unique number has to be chosen.
RO	Mass density.
G	Shear modulus.
SIGY	Yield stress.
ETAN	Plastic hardening modulus.
BULK	Bulk modulus, K.

**Remarks:**

Here the pressure is integrated in time

$$\dot{p} = -K \dot{\epsilon}_{ii}$$

where  $\dot{\epsilon}_{ii}$  is the volumetric strain rate.

**\*MAT\_ISOTROPIC\_ELASTIC\_FAILURE**

This is Material Type 13. This is a non-iterative plasticity with simple plastic strain failure model.

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	G	SIGY	ETAN	BULK		
Type	I	F	F	F	F	F		
Default	none	none	none	none	0.0	none		

Card 2

Variable	EPF	PRF	REM	TREM				
Type	F	F	F	F				
Default	none	0.0	0.0	0.0				

**VARIABLE****DESCRIPTION**

MID	Material identification. A unique number has to be chosen.
RO	Mass density.
G	Shear modulus.
SIGY	Yield stress.
ETAN	Plastic hardening modulus.
BULK	Bulk modulus.
EPF	Plastic failure strain.
PRF	Failure pressure ( $\leq 0.0$ ).

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
REM	Element erosion option: EQ.0.0: failed element eroded after failure, NE.0.0: element is kept, no removal except by $\Delta t$ below.
TREM	$\Delta t$ for element removal: EQ.0.0: $\Delta t$ is not considered (default), GT.0.0: element eroded if element time step size falls below $\Delta t$ .

**Remarks:**

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

The element erosion option based on  $\Delta t$  must be used cautiously with the contact options. Nodes to surface contact is recommended with all nodes of the eroded brick elements included in the node list. As the elements are eroded the mass remains and continues to interact with the master surface.

**\*MAT\_SOIL\_AND\_FOAM\_FAILURE**

This is Material Type 14. The input for this model is the same as for \*MATERIAL\_SOIL\_AND\_FOAM (Type 5); however, when the pressure reaches the failure pressure, the element loses its ability to carry tension. It should be used only in situations when soils and foams are confined within a structure or when geometric boundaries are present.

\*MAT\_JOHNSON\_COOK

This is Material Type 15. The Johnson/Cook strain and temperature sensitive plasticity is sometimes used for problems where the strain rates vary over a large range and adiabatic temperature increases due to plastic heating cause material softening.

Card Format

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	G	E	PR	DTF	VP	
Type	I	F	F	F	F	F	F	
Default	none	none	none	none	none	0.0	0.0	

Card 2

Variable	A	B	N	C	M	TM	TR	EPSO
Type	F	F	F	F	F	F	F	F
Default	none	0.0	0.0	0.0	none	none	none	none

Card 3

Variable	CP	PC	SPALL	IT	D1	D2	D3	D4
Type	F	F	F	F	F	F	F	F
Default	none	0.0	2.0	0.0	0.0	0.0	0.0	0.0

Card 4

Variable	D5							
Type	F							
Default	0.0							

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MID	Material identification. A unique number has to be chosen.
RO	Mass density
G	Shear modulus
E	Young's Modulus (shell elements only)
PR	Poisson's ratio (shell elements only)
DTF	Minimum time step size for automatic element deletion (shell elements)
VP	Formulation for rate effects: EQ.0.0: Scale yield stress (default), EQ.1.0: Viscoplastic formulation.
A	See equations below.
B	See equations below.
N	See equations below.
C	See equations below.
M	See equations below.
TM	Melt temperature
TR	Room temperature
EPSO	Effective plastic strain rate. This value depends on the time units. Typically, nput 1 for units of seconds, 0.001 for units of milliseconds, 0.000001 for microseconds, etc.
CP	Specific heat



VARIABLE	DESCRIPTION
PC	Failure stress or pressure cutoff ( $p_{\min} < 0.0$ )
SPALL	Spall type: EQ. 0.0: default set to "2.0", EQ. 1.0: $p \geq p_{\min}$ , EQ. 2.0: if $\sigma_{\max} \geq -p_{\min}$ element spalls and tension, $p < 0$ , is never allowed, EQ. 3.0: $p < -p_{\min}$ element spalls and tension, $p < 0$ , is never allowed.
IT	Plastic strain iteration options: EQ. 0.0: no iterations (default), EQ. 1.0: accurate iterative solution for plastic strain. Much more expensive than default.
D1-D5	Failure parameters, see equations below.

**Remarks:**

Johnson and Cook express the flow stress as

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

where

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

$\bar{\epsilon}^p$  effective plastic strain

$\dot{\epsilon}^* = \frac{\dot{\bar{\epsilon}}^p}{\epsilon_0}$  effective plastic strain rate for  $\epsilon_0 = 1 \text{ s}^{-1}$

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

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

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

The strain at fracture is given by

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

where  $\sigma^*$  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,  $\sigma_{max}$ , exceeds the limiting value  $\sigma_p$ . 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,  $\Delta t_{max}$ . Generally,  $\Delta t_{max}$  goes down as the element becomes more distorted. To assure stability of time integration, the global LS-DYNA time step is the minimum of the  $\Delta t_{max}$  values calculated for all elements in the model. Using this option allows the selective deletion of elements whose time step  $\Delta t_{max}$  has fallen below the specified minimum time step,  $\Delta t_{crit}$ . 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.

**\*MAT\_PSEUDO\_TENSOR**

This is Material Type 16. This model has been used to analyze buried steel reinforced concrete structures subjected to impulsive loadings.

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	G	PR				
Type	I	F	F	F				
Default	none	none	none	none				

Card 2

Variable	SIGF	A0	A1	A2	A0F	A1F	B1	PER
Type	F	F	F	F	F	F	F	F
Default	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0

Card 3

Variable	ER	PRR	SIGY	ETAN	LCP	LCR		
Type	F	F	F	F	F	F		
Default	0.0	0.0	none	0.0				

Card 4

Variable	X1	X2	X3	X4	X5	X6	X7	X8
Type	F	F	F	F	F	F	F	F
Default								

Card 5

Variable	X9	X10	X11	X12	X13	X14	X15	X16
Type	F	F	F	F	F	F	F	F
Default								

Card 6            1            2            3            4            5            6            7            8

Variable	YS1	YS2	YS3	YS4	YS5	YS6	YS7	YS8
Type	F	F	F	F	F	F	F	F
Default								

Card 7

Variable	YS9	YS10	YS11	YS12	YS13	YS14	YS15	YS16
Type	F	F	F	F	F	F	F	F
Default								

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
G	Shear modulus.
PR	Poisson's ratio.
SIGF	Tensile cutoff (maximum principal stress for failure).
A0	Cohesion.
A1	Pressure hardening coefficient.
A2	Pressure hardening coefficient.
A0F	Cohesion for failed material.
A1F	Pressure hardening coefficient for failed material.
B1	Damage scaling factor.
PER	Percent reinforcement.
ER	Elastic modulus for reinforcement.
PRR	Poisson's ratio for reinforcement.
SIGY	Initial yield stress.
ETAN	Tangent modulus/plastic hardening modulus.
LCP	Load curve ID giving rate sensitivity for principal material, see *DEFINE_CURVE.
LCR	Load curve ID giving rate sensitivity for reinforcement, see *DEFINE_CURVE.
X	Effective plastic strain, damage, or pressure. See discussion below.
YS	Yield stress.

**Remarks:**

This model can be used in two major modes - a simple tabular pressure-dependent yield surface, and a potentially complex model featuring two yield versus pressure functions with the means of migrating from one curve to the other. For both modes, load curve 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  $\sigma_1$  is the maximum principal stress and  $\sigma_3$  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  $1.5 \times 10^6$  psi).

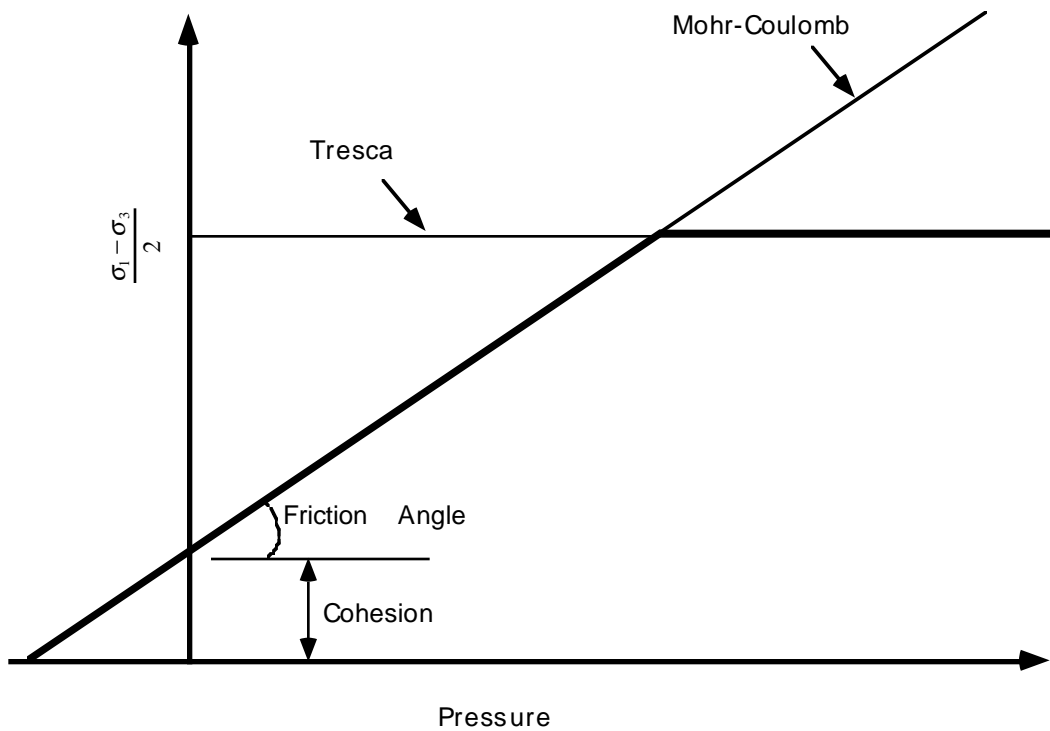


Figure 3.9. Mohr-Coulomb surface with a Tresca limit.

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  $3p$ , 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 ( $\sigma_1$ ) in the element exceeds the tensile cut-off ( $\sigma_{cut}$ ). 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  $\sigma_{cut}$  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.

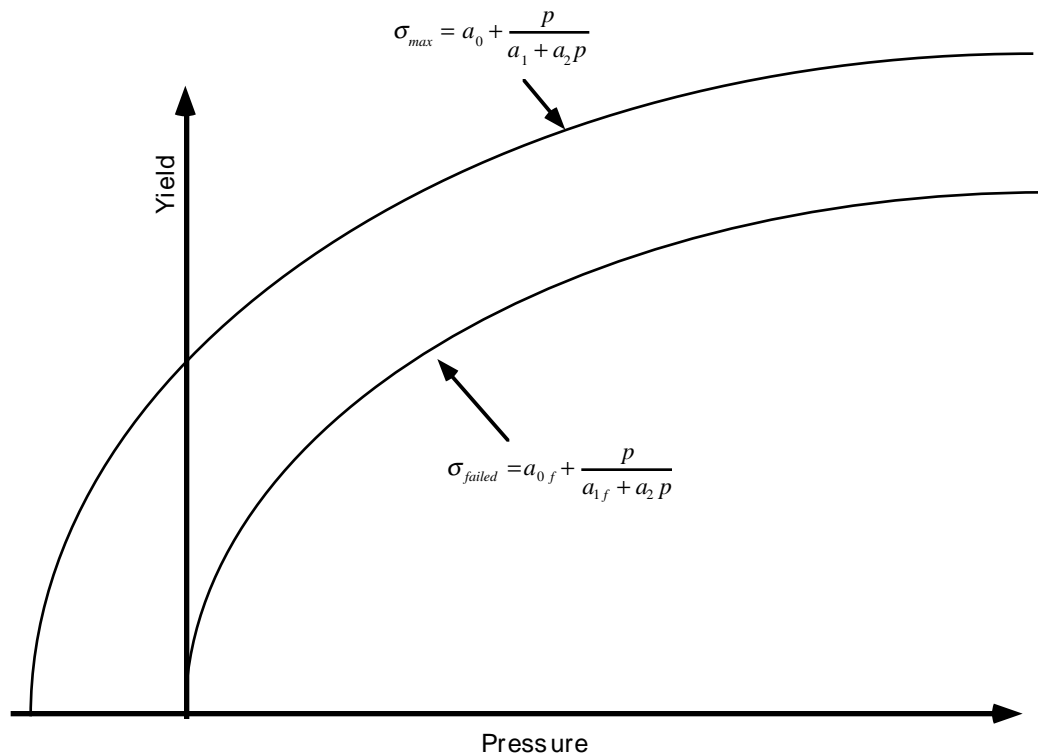


Figure 3.10. Two-curve concrete model with damage and failure.

#### 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 ( $\sigma_1$ ) in the element exceeds the tensile cut-off ( $\sigma_{cut}$ ). 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,  $\eta$ , versus effective plastic strain. LS-DYNA evaluates  $\eta$  at the current effective plastic strain and then calculated the yield stress as

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

where  $\sigma_{max}$  and  $\sigma_{failed}$  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:

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

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

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

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  $a_0$  is negative then  $\sigma_{cut}$  is assumed to be the unconfined concrete compressive strength,  $f'_c$ , and  $-a_0$  is assumed to be a conversion fraction from LS-DYNA pressure units to psi. In this case the parameter values generated internally are



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

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

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

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

$$a_{0f} = 0$$

$$a_{1f} = 0.385$$

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

### Mixture model

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

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

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

### A Suggestion

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

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

and a damage table of:

Card 4:	0.0 5.17E-04	8.62E-06 6.38E-04	2.15E-05 7.98E-04	3.14E-05	3.95E-04
Card 5:	9.67E-04 4.00E-03	1.41E-03 4.79E-03	1.97E-03 0.909	2.59E-03	3.27E-03
Card 6:	0.309 0.790	0.543 0.630	0.840 0.469	0.975	1.000
Card 7:	0.383 0.086	0.247 0.056	0.173 0.0	0.136	0.114

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

\*MAT\_ORIENTED\_CRACK

This is Material Type 17. This material may be used to model brittle materials which fail due to large tensile stresses.

Card Format

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	E	PR	SIGY	ETAN	FS	PRF
Type	I	F	F	F	F	F	F	F
Default	none	none	none	none	none	0.0	none	0.0

VARIABLE

DESCRIPTION

- MID            Material identification. A unique number has to be chosen.
- RO            Mass density.
- E            Young's modulus.
- PR            Poisson's ratio.
- SIGY           Yield stress.
- ETAN           Plastic hardening modulus.
- FS            Fracture stress.
- PRF           Failure or cutoff pressure(≤ 0.0).

**Remarks:**

This is an isotropic elastic-plastic material which includes a failure model with an oriented crack. The von Mises yield condition is given by:

$$\phi = J_2 - \frac{\sigma_y^2}{3}$$

where the second stress invariant,  $J_2$ , is defined in terms of the deviatoric stress components as

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

and the yield stress,  $\sigma_y$ , is a function of the effective plastic strain,  $\epsilon_{eff}^p$ , and the plastic hardening modulus,  $E_p$ :

$$\sigma_y = \sigma_0 + E_p \epsilon_{eff}^p$$

The effective plastic strain is defined as:

$$\epsilon_{eff}^p = \int_0^t d\epsilon_{eff}^p$$

where 
$$d\epsilon_{eff}^p = \sqrt{\frac{2}{3}} d\epsilon_{ij}^p d\epsilon_{ij}^p$$

and the plastic tangent modulus is defined in terms of the input tangent modulus,  $E_t$ , as

$$E_p = \frac{EE_t}{E - E_t}$$

Pressure in this model is found from evaluating an equation of state. A pressure cutoff can be defined such that the pressure is not allowed to fall below the cutoff value.

The oriented crack fracture model is based on a maximum principal stress criterion. When the maximum principal stress exceeds the fracture stress,  $\sigma_f$ , the element fails on a plane perpendicular to the direction of the maximum principal stress. The normal stress and the two shear stresses on that plane are then reduced to zero. This stress reduction is done according to a delay function that reduces the stresses gradually to zero over a small number of time steps. This delay function procedure is used to reduce the ringing that may otherwise be introduced into the system by the sudden fracture.

After a tensile fracture, the element will not support tensile stress on the fracture plane, but in compression will support both normal and shear stresses. The orientation of this fracture surface is tracked throughout the deformation, and is updated to properly model finite deformation effects. If the maximum principal stress subsequently exceeds the fracture stress in another direction, the element fails isotropically. In this case the element completely loses its ability to support any shear stress or hydrostatic tension, and only compressive hydrostatic stress states are possible. Thus, once isotropic failure has occurred, the material behaves like a fluid.

This model is applicable to elastic or elastoplastic materials under significant tensile or shear loading when fracture is expected. Potential applications include brittle materials such as ceramics as well as porous materials such as concrete in cases where pressure hardening effects are not significant.

\*MAT\_POWER\_LAW\_PLASTICITY

This is Material Type 18. This is an isotropic plasticity model with rate effects which uses a power law hardening rule.

Card Format

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	E	PR	K	N	SRC	SRP
Type	I	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	0.0	0.0

Card 2            1            2            3            4            5            6            7            8

Variable	SIGY	VP						
Type	F	F						
Default	0.0	0.0						

VARIABLE

DESCRIPTION

- MID            Material identification. A unique number has to be chosen.
- RO            Mass density.
- E            Young's modulus.
- PR            Poisson's ratio.
- K            Strength coefficient.
- N            Hardening exponent.
- SRC           Strain rate parameter, C, if zero, rate effects are ignored.
- SRP           Strain rate parameter, P, if zero, rate effects are ignored.

<b>VARIABLE</b>	<b>DESCRIPTION</b>
SIGY	Optional input parameter for defining the initial yield stress, $\sigma_y$ . Generally, this parameter is not necessary and the strain to yield is calculated as described below. LT.0.02: $\epsilon_{yp} = SIGY$ GE.0.02: See below.
VP	Formulation for rate effects: EQ.0.0: Scale yield stress (default), EQ.1.0: Viscoplastic formulation.

**Remarks:**

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

$$\sigma_y = k \epsilon^n = k (\epsilon_{yp} + \bar{\epsilon}^p)^n$$

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

$$\sigma = E \epsilon$$

$$\sigma = k \epsilon^n$$

which gives the elastic strain at yield as:

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

If SIGY yield is nonzero and greater than 0.02 then:

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

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

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

where  $\dot{\epsilon}$  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 in results can be dramatic.

\*MAT\_STRAIN\_RATE\_DEPENDENT\_PLASTICITY

This is Material Type 19. A strain rate dependent material can be defined. For an alternative, see Material Type 24. Required is a curve for the yield stress versus the effective strain rate. Optionally, Young’s modulus and the tangent modulus can also be defined versus the effective strain rate. Also, optional failure of the material can be defined either by defining a von Mises stress at failure as a function of the effective strain rate (valid for solids/shells/thick shells) or by defining a minimum time step size (only for shells).

Card Format

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	E	PR	VP			
Type	I	F	F	F	F			
Default	none	none	none	none	0.0			

Card 2

Variable	LC1	ETAN	LC2	LC3	LC4	TDEL	RDEF	
Type	F	F	F	F	F	F	F	
Default	none	0.0	0.0	0.0	0.0	0.0	0.0	

VARIABLE

DESCRIPTION

- MID            Material identification. A unique number has to be chosen.
- RO            Mass density.
- E             Young’s modulus.
- PR            Poisson’s ratio.

<b>VARIABLE</b>	<b>DESCRIPTION</b>
VP	Formulation for rate effects: EQ.0.0: Scale yield stress (default), EQ.1.0: Viscoplastic formulation.
LC1	Load curve ID defining the yield stress $\sigma_0$ as a function of the effective strain rate.
ETAN	Plastic hardening modulus $E_t$
LC2	Load curve ID defining Young's modulus as a function of the effective strain rate (optional).
LC3	Load curve ID defining tangent modulus as a function of the effective strain rate (optional).
LC4	Load curve ID defining von Mises stress at failure as a function of the effective strain rate (optional).
TDEL	Minimum time step size for automatic element deletion. Use for shells only.
RDEF	Redefinition of failure curve: EQ.1.0: Effective plastic strain, EQ.2.0: Maximum principal stress.

**Remarks:**

In this model, a load curve is used to describe the yield strength  $\sigma_0$  as a function of effective strain rate  $\dot{\bar{\epsilon}}$  where

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

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

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

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

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

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



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

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

In addition to the above failure criterion, this material model also supports a shell element deletion criterion based on the maximum stable time step size for the element,  $\Delta t_{max}$ . Generally,  $\Delta t_{max}$  goes down as the element becomes more distorted. To assure stability of time integration, the global LS-DYNA time step is the minimum of the  $\Delta t_{max}$  values calculated for all elements in the model. Using this option allows the selective deletion of elements whose time step  $\Delta t_{max}$  has fallen below the specified minimum time step,  $\Delta t_{crit}$ . 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 in results can be dramatic.

**\*MAT\_RIGID**

This is Material 20. Parts made from this material are considered to belong to a rigid body (for each part ID). Also, the coupling of a rigid body with MADYMO and CAL3D can be defined via this material. Alternatively, a VDA surface can be attached as surface to model the geometry, e.g, for the tooling in metalforming applications. Also, global and local constraints on the mass center can be optionally defined. Optionally, a local consideration for output and user-defined airbag sensors can be chosen.

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	E	PR	N	COUPLE	M	ALIAS
Type	I	F	F	F	F	F	F	C
Default	none	none	none	none	0	0	0	blank

Card 2

Variable	CMO	CON1	CON2					
Type	F	F	F					
Default	0	0	0					

**Optional Card Format for output (Must be included but may be left blank).**

Card 3

Variable	LCO or A1	A2	A3	V1	V2	V3		
Type	F	F	F	F	F	F		
Default	0	0	0	0	0	0		

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MID	Material identification. A unique number has to be chosen.
RO	Mass density
E	Young's modulus. Reasonable values have to be chosen for contact analysis (choice of penalty), see remark below.
PR	Poisson's ratio. Reasonable values have to be chosen for contact analysis (choice of penalty), see remark below.
N	MADYMO3D ( <b>not CAL3D</b> ) 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 .
COUPLE	Coupling option if applicable: EQ.-1: attach VDA surface in ALIAS (defined in the eighth field) and automatically generate a mesh for viewing the surface in LS-TAURUS.  MADYMO3D/CAL3D coupling option: EQ.0: the undeformed geometry input to LS-DYNA corresponds to the local system for MADYMO/CAL3D. The finite element 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-DYNA3D.
M	MADYMO/CAL3D Coupling option flag: EQ.0: use normal LS-DYNA rigid body updates, EQ.m: this rigid body corresponds to MADYMO/CAL3D system number m. Rigid body updates are performed by MADYMO/CAL3D.
ALIAS	VDA surface alias name, see Appendix I.
CMO	Center of mass constraint option, CMO: EQ.+1.0: constraints applied in global directions,

VARIABLE	DESCRIPTION
CON1	<p>EQ. 0.0: no constraints, EQ. -1.0: constraints applied in local directions (SPC constraint).</p> <p>First constraint parameter:</p> <p><u>If CMO=+1.0, then specify global translational constraint:</u> 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.</p> <p><u>If CMO=-1.0, then specify local coordinate system ID.</u> See *DEFINE_COORDINATE_OPTION: This coordinate system is fixed in time.</p>
CON2	<p>Second constraint parameter:</p> <p><u>If CMO=+1.0, then specify global rotational constraint:</u> 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.</p> <p><u>If CMO=-1.0, then specify local (SPC) constraint:</u> EQ.000000 no constraint, EQ.100000 constrained x translation, EQ.010000 constrained y translation, EQ.001000 constrained z translation, EQ.000100 constrained x rotation, EQ.000010 constrained y rotation, EQ.000001 constrained z rotation.</p> <p>Any combination of local constraints can be achieved by adding the number 1 into the corresponding column.</p>
LCO	<p>Local coordinate system number for output. See *DEFINE_COORDINATE_OPTION.</p> <p>*****Alternative method for specifying local system below.*****</p>
A1-V3	<p>Define two vectors <b>a</b> and <b>v</b>, fixed in the rigid body which are used for output and the user defined airbag sensor subroutines. The output parameters are in the directions <b>a</b>, <b>b</b>, and <b>c</b> where the latter are given by the cross products <b>c=a×v</b> and <b>b=c×a</b>. This input is optional.</p>

**Remarks:**

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

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

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

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

Young's modulus,  $E$ , and Poisson's ratio,  $\nu$ , 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.

**\*MAT\_ORTHOTROPIC\_THERMAL**

This is Material Type 21. A linearly elastic material with orthotropic temperature dependent coefficients can be defined.

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	EA	EB	EC	PRBA	PRCA	PRCB
Type	I	F	F	F	F	F	F	F

Card 2

Variable	GAB	GBC	GCA	AA	AB	AC	AOPT	
Type	F	F	F	F	F	F	F	

Card 3

Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card 4

Variable	V1	V2	V3	D1	D2	D3	BETA	REF
Type	F	F	F	F	F	F	F	

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
EA	$E_a$ , Young's modulus in a-direction.
EB	$E_b$ , Young's modulus in b-direction.
EC	$E_c$ , Young's modulus in c-direction.
PRBA	$\nu_{ba}$ , Poisson's ratio, ba.
PRCA	$\nu_{ca}$ , Poisson's ratio, ca.
PRCB	$\nu_{cb}$ , Poisson's ratio, cb
GAB	$G_{ab}$ , Shear modulus, ab.
GBC	$G_{bc}$ , Shear modulus, bc.
GCA	$G_{ca}$ , Shear modulus, ca.
AA	$\alpha_a$ , coefficients of thermal expansion in the a-direction.
AB	$\alpha_b$ , coefficients of thermal expansion in the b-direction.
AC	$\alpha_c$ , coefficients of thermal expansion in the c-direction.
AOPT	Material axes option, see Figure 20.1: EQ. 0.0: locally orthotropic with material axes determined by element nodes as shown in Figure 20.1. Nodes 1, 2, and 4 of an element are identical to the Nodes used for the definition of a coordinate system as by *DEFINE_COORDINATE_NODES. EQ. 1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center, this is the a-direction. EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR. EQ. 3.0: applicable to shell elements only. This option determines locally orthotropic material axes by offsetting the material axes by an angle to be specified from a line in the plane of the shell determined by taking the cross product of the vector $\mathbf{v}$ defined below with the shell normal vector. EQ. 4.0: locally orthotropic in cylindrical coordinate system with the material axes determined by a vector $\mathbf{v}$ and an originating point, P. They define the axis of symmetry.
XP,YP,ZP	Coordinates of point $\mathbf{p}$ for AOPT = 1.
A1,A2,A3	Components of vector $\mathbf{a}$ for AOPT = 2.

VARIABLE	DESCRIPTION
V1,V2,V3	Components of vector <b>v</b> for AOPT = 3.
D1,D2,D3	Components of vector <b>d</b> for AOPT = 2.
BETA	Material angle in degrees for AOPT = 3, may be overridden on the element card. This angle is measured with respect to $y=v \times n$ .
REF	Use reference geometry to initialize the stress tensor. The reference geometriy is defined by the keyword: <b>*INITIAL_FOAM_REFERENCE_GEOMETRY</b> . This option is currently restricted to 8-noded solid elements with one point integration. EQ.0.0: off, EQ.1.0: on.

**Remarks:**

In the implementation for three dimensional continua a total Lagrangian formulation is used. In this approach the material law that relates second Piola-Kirchhoff stress **S** to the Green-St. Venant strain **E** is

$$S = C \cdot E = T' C_l T \cdot E$$

where **T** is the transformation matrix [Cook 1974].

$$T = \begin{bmatrix} l_1^2 & m_1^2 & n_1^2 & l_1 m_1 & m_1 n_1 & n_1 l_1 \\ l_2^2 & m_2^2 & n_2^2 & l_2 m_2 & m_2 n_2 & n_2 l_2 \\ l_3^2 & m_3^2 & n_3^2 & l_3 m_3 & m_3 n_3 & n_3 l_3 \\ 2l_1 l_2 & 2m_1 m_2 & 2n_1 n_2 & (l_1 m_2 + l_2 m_1) & (m_1 n_2 + m_2 n_1) & (n_1 l_2 + n_2 l_1) \\ 2l_2 l_3 & 2m_2 m_3 & 2n_2 n_3 & (l_2 m_3 + l_3 m_2) & (m_2 n_3 + m_3 n_2) & (n_2 l_3 + n_3 l_2) \\ 2l_3 l_1 & 2m_3 m_1 & 2n_3 n_1 & (l_3 m_1 + l_1 m_3) & (m_3 n_1 + m_1 n_3) & (n_3 l_1 + n_1 l_3) \end{bmatrix}$$

$l_i, m_i, n_i$  are the direction cosines

$$x_i' = l_i x_1 + m_i x_2 + n_i x_3 \quad \text{for } i = 1, 2, 3$$

and  $x_i'$  denotes the material axes. The constitutive matrix  $C_l$  is defined in terms of the material axes as



$$C_l^{-1} = \begin{bmatrix} \frac{1}{E_{11}} & -\frac{\nu_{21}}{E_{22}} & -\frac{\nu_{31}}{E_{33}} & 0 & 0 & 0 \\ -\frac{\nu_{12}}{E_{11}} & \frac{1}{E_{22}} & -\frac{\nu_{32}}{E_{33}} & 0 & 0 & 0 \\ -\frac{\nu_{13}}{E_{11}} & -\frac{\nu_{23}}{E_{22}} & \frac{1}{E_{33}} & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1}{G_{12}} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1}{G_{23}} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1}{G_{31}} \end{bmatrix}$$

where the subscripts denote the material axes, i.e.,

$$\nu_{ij} = \nu_{x'_i x'_j} \quad \text{and} \quad E_{ii} = E_{x'_i}$$

Since  $C_l$  is symmetric

$$\frac{\nu_{12}}{E_{11}} = \frac{\nu_{21}}{E_{22}}, \text{ etc.}$$

The vector of Green-St. Venant strain components is

$$E' = [E_{11}, E_{22}, E_{33}, E_{12}, E_{23}, E_{31},]$$

which include the local thermal strains which are integrated in time:

$$\epsilon_{aa}^{n+1} = \epsilon_{aa}^n + \alpha_a (T^{n+1} - T^n)$$

$$\epsilon_{bb}^{n+1} = \epsilon_{bb}^n + \alpha_b (T^{n+1} - T^n)$$

$$\epsilon_{cc}^{n+1} = \epsilon_{cc}^n + \alpha_c (T^{n+1} - T^n)$$

After computing  $S_{ij}$  we use Equation (15.32) to obtain the Cauchy stress. This model will predict realistic behavior for finite displacement and rotations as long as the strains are small.

In the implementation for shell elements, the stresses are integrated in time and are updated in the corotational coordinate system. In this procedure the local material axes are assumed to remain orthogonal in the deformed configuration. This assumption is valid if the strains remain small.

**\*MAT\_COMPOSITE\_DAMAGE**

This is Material Type 22. An orthotropic material with optional brittle failure for composites can be defined following the suggestion of [Chang and Chang 1982a,1982b]. Three failure criteria are possible, see Theoretical Manual. After bulk compression, a force can be still transmitted, however, another bulk modulus has to be given.

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	EA	EB	EC	PRBA	PRCA	PRCB
Type	I	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	none	none

Card 2

Variable	GAB	GBC	GCA	KFAIL	AOPT	MACF		
Type	F	F	F	F	F	F		
Default	none	none	none	0.0	0.0	1.0		

Card 3

Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		
Default	0.0	0.0	0.0	0.0	0.0	0.0		

Card 4

Variable	V1	V2	V3	D1	D2	D3		
Type	F	F	F	F	F	F		
Default	0.0	0.0	0.0	0.0	0.0	0.0		

Card 5

Variable	SC	XT	YT	YC	ALPH	SN	SYZ	SZX
Type	F	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	none	none

**VARIABLE**

**DESCRIPTION**

- MID Material identification. A unique number has to be chosen.
- RO Mass density
- EA  $E_a$ , Young's modulus in a-direction.
- EB  $E_b$ , Young's modulus in b-direction.
- EC  $E_c$ , Young's modulus in c-direction.
- PRBA  $\nu_{ba}$ , Poisson ratio, ba.
- PRCA  $\nu_{ca}$ , Poisson ratio, ca.
- PRCB  $\nu_{cb}$ , Poisson ratio, cb.
- GAB  $G_{ab}$ , Shear modulus, ab.
- GBC  $G_{bc}$ , Shear modulus, bc.
- GCA  $G_{ca}$ , Shear modulus, ca.
- KFAIL Bulk modulus of failed material. Necessary for compressive failure.

<b>VARIABLE</b>	<b>DESCRIPTION</b>
AOPT	Material axes option: EQ. 0.0: locally orthotropic with material axes determined by element nodes as shown in Figure 20.1. Nodes 1, 2, and 4 of an element are identical to the Nodes used for the definition of a coordinate system as by *DEFINE_COORDINATE_NODES. EQ. 1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center, this is the a-direction. EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR. EQ. 3.0: applicable to shell elements only. This option determines locally orthotropic material axes by offsetting the material axes by an angle to be specified from a line in the plane of the shell determined by taking the cross product of the vector $\mathbf{v}$ defined below with the shell normal vector.
MACF	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.
XP,YP,ZP	Coordinates of point $\mathbf{p}$ for AOPT = 1.
A1,A2,A3	Components of vector $\mathbf{a}$ for AOPT = 2.
V1,V2,V3	Components of vector $\mathbf{v}$ for AOPT = 3.
D1,D2,D3	Components of vector $\mathbf{d}$ for AOPT = 2.
SC	Shear strength, ab plane, see Theoretical Manual.
XT	Longitudinal tensile strength, a-axis, see Theoretical Manual.
YT	Transverse tensile strength, b-axis, see Theoretical Manual.
YC	Transverse compressive strength, b-axis, see Theoretical Manual.
ALPH	Shear stress parameter for the nonlinear term, see Theoretical Manual. Suggested range 0 – 0.5.
SN	Normal tensile strength ( <i>solid elements only</i> )
SYZ	Transverse shear strength ( <i>solid elements only</i> )
SZX	Transverse shear strength ( <i>solid elements only</i> )

**Remarks:**

The number of additional integration point variables for shells written to the LS-DYNA database is input by the optional \*DATABASE\_BINARY as variable NEIPS. These additional variables are tabulated below ( $ip$  = shell integration point):

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

# \*MAT

## \*MAT\_TEMPERATURE\_DEPENDENT\_ORTHOTROPIC

---

### \*MAT\_TEMPERATURE\_DEPENDENT\_ORTHOTROPIC

This is Material Type 23. An orthotropic elastic material with arbitrary temperature dependency can be defined.

#### Card Format

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	AOPT	REF				
Type	I	F	F	F				

Card 2

Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card 3

Variable	V1	V2	V3	D1	D2	D3		
Type	F	F	F	F	F	F		

**Define one set of constants on two cards for each temperature point. Up to 48 points (96 cards) can be defined. The next “\*” card terminates the input.**

Cards 1 for  
Temperature  
T<sub>i</sub>

Variable	E <sub>Ai</sub>	E <sub>Bi</sub>	E <sub>Ci</sub>	PRAB <sub>i</sub>	PRCA <sub>i</sub>	PRCB <sub>i</sub>		
Type	F	F	F	F	F	F		

Cards 2 for  
Temperature  
Ti

Variable	AAi	ABi	ACi	GABi	GCAi	GCBi	Ti	
Type	F	F	F	F	F	F	F	

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
AOPT	<p>Material axes option:</p> <p>EQ. 0.0: locally orthotropic with material axes determined by element nodes as shown in Figure 20.1. Nodes 1, 2, and 4 of an element are identical to the Nodes used for the definition of a coordinate system as by *DEFINE_COORDINATE_NODES.</p> <p>EQ. 1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center, this is the a-direction.</p> <p>EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR.</p> <p>EQ. 3.0: applicable to shell elements only. This option determines locally orthotropic material axes by offsetting the material axes by an angle to be specified from a line in the plane of the shell determined by taking the cross product of the vector <b>v</b> defined below with the shell normal vector.</p> <p>EQ. 4.0: locally orthotropic in cylindrical coordinate system with the material axes determined by a vector <b>v</b> and an originating point, P. They define the axis of symmetry.</p>
REF	<p>Use reference geometry to initialize the stress tensor. The reference geometry is defined by the keyword:*INITIAL_FOAM_REFERENCE_GEOMETRY. This option is currently restricted to 8-noded solid elements with one point integration.</p> <p>EQ.0.0: off, EQ.1.0: on.</p>
XP,YP,ZP	Coordinates of point <b>p</b> for AOPT = 1.
A1,A2,A3	Components of vector <b>a</b> for AOPT = 2.
V1,V2,V3	Components of vector <b>v</b> for AOPT = 3.
D1,D2,D3	Components of vector <b>d</b> for AOPT = 2.

VARIABLE	DESCRIPTION
EAI	$E_a$ , Young's modulus in a-direction at temperature $T_i$ .
EBI	$E_b$ , Young's modulus in b-direction at temperature $T_i$ .
ECI	$E_c$ , Young's modulus in c-direction at temperature $T_i$ .
PRBAi	$\nu_{ba}$ , Poisson's ratio ba at temperature $T_i$ .
PRCAi	$\nu_{ca}$ , Poisson's ratio ca at temperature $T_i$ .
PRCBI	$\nu_{cb}$ , Poisson's ratio cb at temperature $T_i$ .
AAi	$\alpha_a$ , coefficient of thermal expansion in a-direction at temperature $T_i$ .
ABi	$\alpha_b$ , coefficient of thermal expansion in b-direction at temperature $T_i$ .
ACi	$\alpha_c$ , coefficient of thermal expansion in c-direction at temperature $T_i$ .
GABi	$G_{ab}$ , Shear modulus ab at temperature $T_i$ .
GBCi	$G_{bc}$ , Shear modulus bc at temperature $T_i$ .
GCAi	$G_{ca}$ , Shear modulus ca at temperature $T_i$ .
Ti	ith temperature

**Remarks:**

In the implementation for three dimensional continua a total Lagrangian formulation is used. In this approach the material law that relates second Piola-Kirchhoff stress  $S$  to the Green-St. Venant strain  $E$  is

$$S = C \cdot E = T^T C_T T \cdot E$$

where  $T$  is the transformation matrix [Cook 1974].

$$T = \begin{bmatrix} l_1^2 & m_1^2 & n_1^2 & l_1 m_1 & m_1 n_1 & n_1 l_1 \\ l_2^2 & m_2^2 & n_2^2 & l_2 m_2 & m_2 n_2 & n_2 l_2 \\ l_3^2 & m_3^2 & n_3^2 & l_3 m_3 & m_3 n_3 & n_3 l_3 \\ 2l_1 l_2 & 2m_1 m_2 & 2n_1 n_2 & (l_1 m_2 + l_2 m_1) & (m_1 n_2 + m_2 n_1) & (n_1 l_2 + n_2 l_1) \\ 2l_2 l_3 & 2m_2 m_3 & 2n_2 n_3 & (l_2 m_3 + l_3 m_2) & (m_2 n_3 + m_3 n_2) & (n_2 l_3 + n_3 l_2) \\ 2l_3 l_1 & 2m_3 m_1 & 2n_3 n_1 & (l_3 m_1 + l_1 m_3) & (m_3 n_1 + m_1 n_3) & (n_3 l_1 + n_1 l_3) \end{bmatrix}$$

$l_i, m_i, n_i$  are the direction cosines



$$x'_i = l_i x_1 + m_i x_2 + n_i x_3 \quad \text{for } i = 1, 2, 3$$

and  $x'_i$  denotes the material axes. The temperature dependent constitutive matrix  $C_l$  is defined in terms of the material axes as

$$C_l^{-1} = \begin{bmatrix} \frac{1}{E_{11}(T)} & -\frac{\nu_{21}(T)}{E_{22}(T)} & -\frac{\nu_{31}(T)}{E_{33}(T)} & 0 & 0 & 0 \\ -\frac{\nu_{12}(T)}{E_{11}(T)} & \frac{1}{E_{22}(T)} & -\frac{\nu_{32}(T)}{E_{33}(T)} & 0 & 0 & 0 \\ -\frac{\nu_{13}(T)}{E_{11}(T)} & -\frac{\nu_{23}(T)}{E_{22}(T)} & \frac{1}{E_{33}(T)} & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1}{G_{12}(T)} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1}{G_{23}(T)} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1}{G_{31}(T)} \end{bmatrix}$$

where the subscripts denote the material axes, i.e.,

$$\nu_{ij} = \nu_{x'_i x'_j} \quad \text{and} \quad E_{ii} = E_{x'_i}$$

Since  $C_l$  is symmetric

$$\frac{\nu_{12}}{E_{11}} = \frac{\nu_{21}}{E_{22}}, \text{ etc.}$$

The vector of Green-St. Venant strain components is

$$E^t = [E_{11}, E_{22}, E_{33}, E_{12}, E_{23}, E_{31},]$$

which include the local thermal strains which are integrated in time:

$$\epsilon_{aa}^{n+1} = \epsilon_{aa}^n + \alpha_a (T^{n+\frac{1}{2}}) [T^{n+1} - T^n]$$

$$\epsilon_{bb}^{n+1} = \epsilon_{bb}^n + \alpha_b (T^{n+\frac{1}{2}}) [T^{n+1} - T^n]$$

$$\epsilon_{cc}^{n+1} = \epsilon_{cc}^n + \alpha_c (T^{n+\frac{1}{2}}) [T^{n+1} - T^n]$$

After computing  $S_{ij}$  we use Equation (15.32) to obtain the Cauchy stress. This model will predict realistic behavior for finite displacement and rotations as long as the strains are small.

For shell elements, the stresses are integrated in time and are updated in the corotational coordinate system. In this procedure the local material axes are assumed to remain orthogonal in the deformed configuration. This assumption is valid if the strains remain small.

**\*MAT\_PIECEWISE\_LINEAR\_PLASTICITY**

This is Material Type 24. An elasto-plastic material with an arbitrary stress versus strain curve and arbitrary strain rate dependency can be defined. See also Remark below. Also, failure based on a plastic strain or a minimum time step size can be defined. For another model with a more comprehensive failure criteria see MAT\_MODIFIED\_PIECEWISE\_LINEAR\_PLASTICITY.

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	E	PR	SIGY	ETAN	FAIL	TDEL
Type	I	F	F	F	F	F	F	F
Default	none	none	none	none	none	0.0	10.E+20	0

Card 2

Variable	C	P	LCSS	LCSR	VP			
Type	F	F	F	F	F			
Default	0	0	0	0	0			

Card 3

Variable	EPS1	EPS2	EPS3	EPS4	EPS5	EPS6	EPS7	EPS8
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

Card 4

Variable	ES1	ES2	ES3	ES4	ES5	ES6	ES7	ES8
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.
SIGY	Yield stress.
ETAN	Tangent modulus, ignored if (LCSS.GT.0) is defined.
FAIL	Failure flag. LT.0.0: User defined failure subroutine is called to determine failure EQ.0.0: Failure is not considered. This option is recommended if failure is not of interest since many calculations will be saved. GT.0.0: Plastic strain to failure. When the plastic strain reaches this value, the element is deleted from the calculation.
TDEL	Minimum time step size for automatic element deletion.
C	Strain rate parameter, C, see formula below.
P	Strain rate parameter, P, see formula below.
LCSS	Load curve ID or Table ID. Load curve ID defining effective stress versus effective plastic strain. If defined EPS1-EPS8 and ES1-ES8 are ignored. The table ID defines for each strain rate value a load curve ID giving the stress versus effective plastic strain for that rate, See Figure 20.7. The stress versus effective plastic strain curve for the lowest value of strain rate is used if the strain rate falls below the minimum value. Likewise, the stress versus effective plastic strain curve for the highest value of strain rate is used if the strain rate exceeds the maximum value. The strain rate parameters: C and P;

VARIABLE	DESCRIPTION
	the curve ID, LCSR; EPS1-EPS8 and ES1-ES8 are ignored if a Table ID is defined.
LCSR	Load curve ID defining strain rate scaling effect on yield stress.
VP	Formulation for rate effects: EQ.0.0: Scale yield stress (default), EQ.1.0: Viscoplastic formulation.
EPS1-EPS8	Effective plastic strain values (optional if SIGY is defined). At least 2 points should be defined. The first point must be zero corresponding to the initial yield stress. WARNING: If the first point is nonzero the yield stress is extrapolated to determine the initial yield. If this option is used SIGY and ETAN are ignored and may be input as zero.
ES1-ES8	Corresponding yield stress values to EPS1 - EPS8.

**Remarks:**

The stress strain behavior may be treated by a bilinear stress strain curve by defining the tangent modulus, ETAN. Alternately, a curve similar to that shown in Figure 20.4 is expected to be defined by (EPS1,ES1) - (EPS8,ES8); however, an effective stress versus effective plastic strain curve (LCSS) may be input instead if eight points are insufficient. The cost is roughly the same for either approach. The most general approach is to use the table definition (LCSS) discussed below.

Three options to account for strain rate effects are possible.

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

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

where  $\dot{\epsilon}$  is the strain rate.  $\dot{\epsilon} = \sqrt{\dot{\epsilon}_{ij} \dot{\epsilon}_{ij}}$ .

If the viscoplastic option is active, VP=1.0, and if SIGY is > 0 then the dynamic yield stress is computed from the sum of the static stress,  $\sigma_y^s(\epsilon_{eff}^p)$ , which is typically given by a load curve ID, and the initial yield stress, SIGY, multiplied by the Cowper-Symonds rate term as follows:

$$\sigma_y(\epsilon_{eff}^p, \dot{\epsilon}_{eff}^p) = \sigma_y^s(\epsilon_{eff}^p) + SIGY \cdot \left( \frac{\dot{\epsilon}_{eff}^p}{C} \right)^{1/p}$$

where the plastic strain rate is used. With this latter approach similar results can be obtained between this model and material model: \*MAT\_ANISOTROPIC\_VISCOPLASTIC. If SIGY=0, the following equation is used instead where the static stress,  $\sigma_y^s(\epsilon_{eff}^p)$ , must be defined by a load curve:

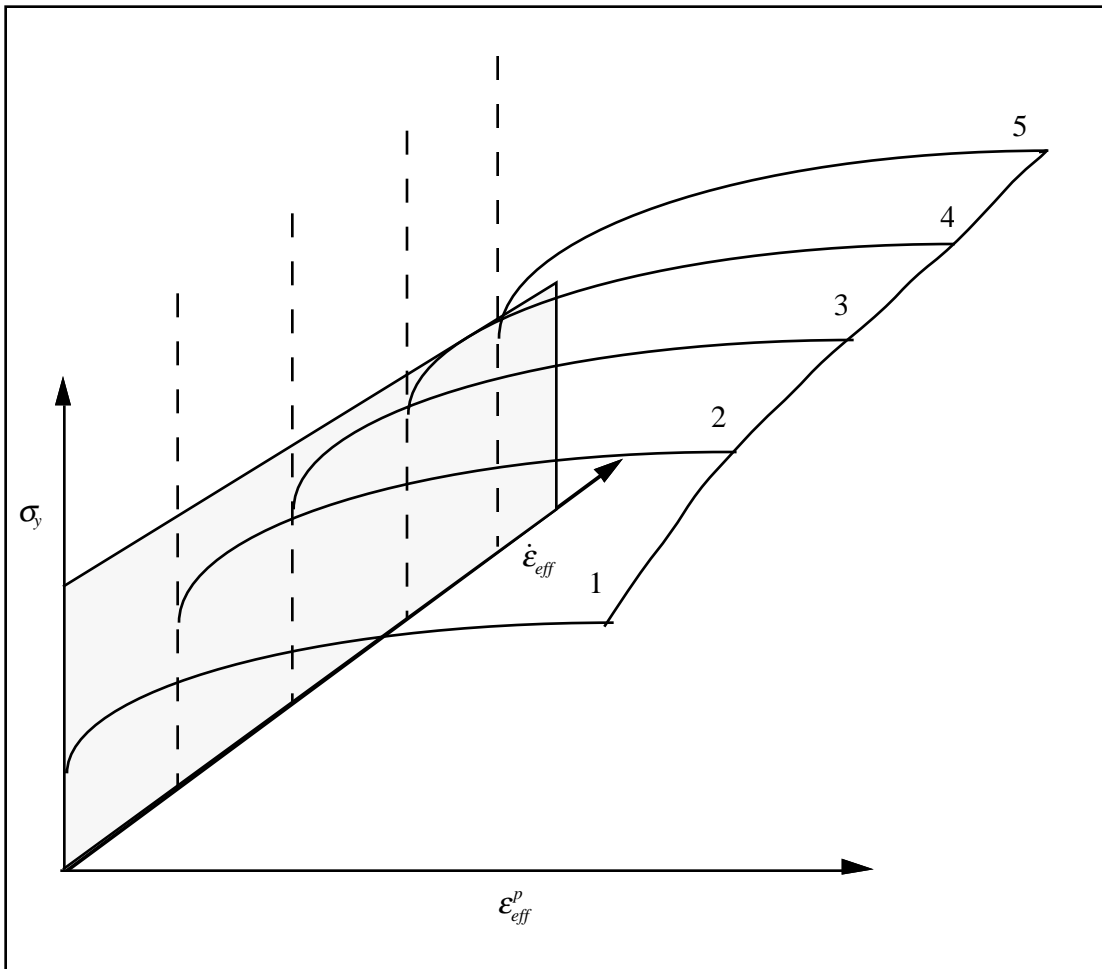
$$\sigma_y(\epsilon_{eff}^p, \dot{\epsilon}_{eff}^p) = \sigma_y^s(\epsilon_{eff}^p) \left[ 1 + \left( \frac{\dot{\epsilon}_{eff}^p}{C} \right)^{1/p} \right]$$

This latter equation is always used if the viscoplastic option is off.

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

III. If different stress versus strain curves can be provided for various strain rates, the option using the reference to a table (LCSS) can be used. Then the table input in \*DEFINE\_TABLE has to be used, see Figure 20.7.

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



**Figure 20.7.** Rate effects may be accounted for by defining a table of curves. If a table ID is specified a curve ID is given for each strain rate, see \*DEFINE\_TABLE. Intermediate values are found by interpolating between curves. Effective plastic strain versus yield stress is expected.

\*MAT\_GEOLOGIC\_CAP\_MODEL

This is Material Type 25. This an inviscid two invariant geologic cap model. This material model can be used for geomechanical problems or for materials as concrete, see references cited below.

Card Format

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	BULK	G	ALPHA	THETA	GAMMA	BETA
Type	I	F	F	F	F	F	F	F

Card 2

Variable	R	D	W	X0	C	N		
Type	F	F	F	F	F	F		

Card 3

Variable	PLOT	FTYPE	VEC	TOFF				
Type	F	F	F	F				

VARIABLE

DESCRIPTION

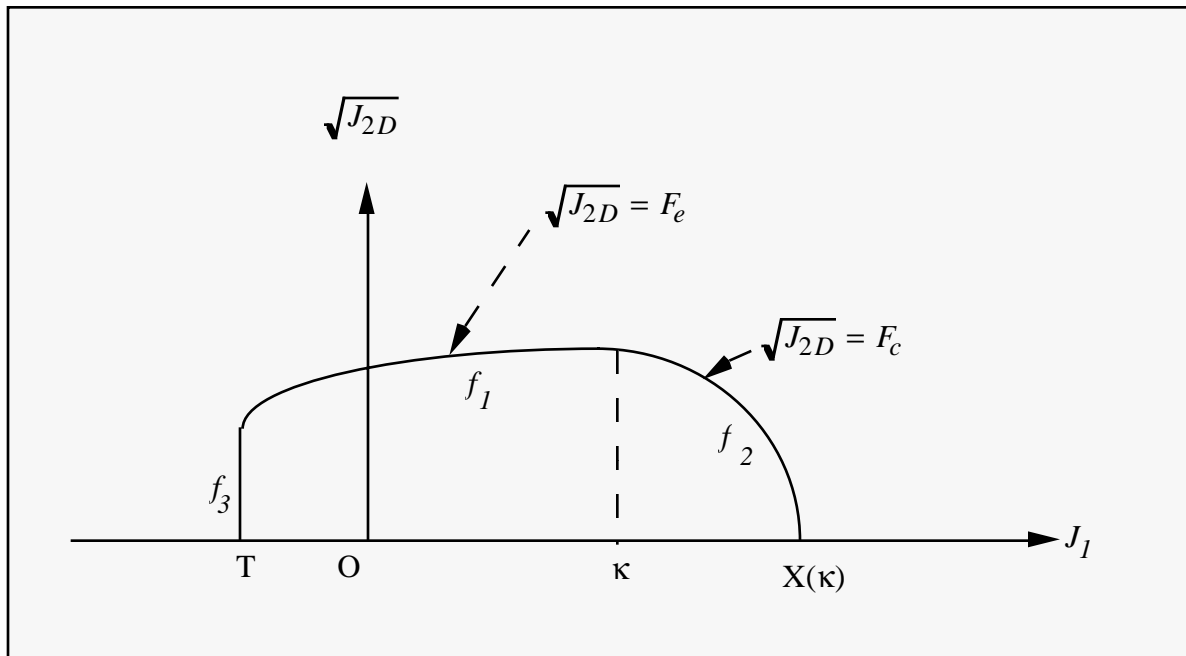
- MID            Material identification. A unique number has to be chosen.
- RO            Mass density.
- BULK        Initial bulk modulus, K.
- G            Initial Shear modulus.
- ALPHA       Failure envelope parameter,  $\alpha$ .

<b>VARIABLE</b>	<b>DESCRIPTION</b>
THETA	Failure envelope linear coefficient, $\theta$ .
GAMMA	Failure envelope exponential coefficient, $\gamma$ .
BETA	Failure envelope exponent, $\beta$ .
R	Cap, surface axis ratio.
D	Hardening law exponent.
W	Hardening law coefficient.
X0	Hardening law exponent, $X_0$ .
C	Kinematic hardening coefficient, $\bar{c}$ .
N	Kinematic hardening parameter.
PLOT	Save the following variable for plotting in TAURUS, to be labeled there as "effective plastic strain:" EQ.1: hardening parameter, $\kappa$ , EQ.2: cap $-J_1$ axis intercept, $X$ ( $\kappa$ ), EQ.3: volumetric plastic strain $\epsilon_v^p$ , EQ.4: first stress invariant, $J_1$ , EQ.5: second stress invariant, $\sqrt{J_2}$ , EQ.6: not used, EQ.7: not used, EQ.8: response mode number, EQ.9: number of iterations.
FTYPE	Formulation flag: EQ.1: soil or concrete (Cap surface may contract), EQ.2: rock (Cap doesn't contract).
VEC	Vectorization flag: EQ.0: vectorized (fixed number of iterations), EQ.1: fully iterative, If the vectorized solution is chosen, the stresses might be slightly off the yield surface; however, on vector computers a much more efficient solution is achieved.
TOFF	Tension Cut Off, $TOFF < 0$ (positive in compression).



**Remarks:**

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



**Figure 20.8.** 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 20.8. First, there is a failure envelope surface, denoted  $f_1$  in the figure. The functional form of  $f_1$  is

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

where  $F_e$  is given by

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

and  $T_{mises} \equiv |X(\kappa_n) - L(\kappa_n)|$ . 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

$$X(\kappa) = \kappa + R F_e(\kappa),$$

and  $L(\kappa)$  is defined by

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

The hardening parameter  $\kappa$  is related to the plastic volume change  $\epsilon_v^p$  through the hardening law

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

Geometrically,  $\kappa$  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:

$$\boldsymbol{\epsilon} = \boldsymbol{\epsilon}^e + \boldsymbol{\epsilon}^p,$$

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

$$\boldsymbol{\sigma} = \mathbf{C}(\boldsymbol{\epsilon} - \boldsymbol{\epsilon}^p),$$

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

The yield condition may be written

$$\begin{aligned} f_1(s) &\leq 0 \\ f_2(s, \kappa) &\leq 0 \\ f_3(s) &\leq 0 \end{aligned}$$

and the plastic consistency condition requires that

$$\begin{aligned} \dot{\lambda}_k f_k &= 0 \\ k &= 1, 2, 3 \\ \dot{\lambda}_k &\geq 0 \end{aligned}$$

where  $\lambda_k$  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  $\sqrt{J_{2D}} - J$  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 20.8). 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-Coulomb 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  $\alpha$  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,  $\alpha$ . The formulation including kinematic hardening is obtained by replacing the stress  $\sigma$  with the translated stress tensor  $\eta \equiv \sigma - \alpha$  in all of the above equation. The history tensor  $\alpha$  is assumed deviatoric, and therefore has only 5 unique components. The evolution of the back stress tensor is governed by the nonlinear hardening law

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

where  $\bar{c}$  is a constant,  $\bar{F}$  is a scalar function of  $\sigma$  and  $\alpha$  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

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

where  $N$  is a constant defining the size of the yield surface. The value of  $N$  may be interpreted as the radial distance 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  $\alpha - N$  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  $\alpha$ ,  $\beta$ ,  $\theta$ , and  $\gamma$  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 are found in (Chen and Baladi, 1985).

\*MAT\_HONEYCOMB

This is Material Type 26. The major use of this material model is for honeycomb and foam materials with real anisotropic behavior. A nonlinear elastoplastic material behavior can be defined separately for all normal and shear stresses. These are considered to be fully uncoupled. See notes below.

Card Format

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	E	PR	SIGY	VF	MU	BULK
Type	I	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	.05	0.0

Card 2

Variable	LCA	LCB	LCC	LCS	LCAB	LCBC	LCCA	LCSR
Type	F	F	F	F	F	F	F	F
Default	none	LCA	LCA	LCA	LCS	LCS	LCS	optional

Card 3            1            2            3            4            5            6            7            8

Variable	EAAU	EBBU	ECCU	GABU	GBCU	GCAU	AOPT	
Type	F	F	F	F	F	F		

Card 4

Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card 5

Variable	D1	D2	D3	TSEF	SSEF			
Type	F	F	F	F	F			

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
E	Young's modulus for compacted honeycomb material.
PR	Poisson's ratio for compacted honeycomb material.
SIGY	Yield stress for fully compacted honeycomb.
VF	Relative volume at which the honeycomb is fully compacted.
MU	$\mu$ , material viscosity coefficient. (default=.05) Recommended.
BULK	Bulk viscosity flag: EQ.0.0: bulk viscosity is not used. This is recommended. EQ.1.0: bulk viscosity is active and $\mu=0$ This will give results identical to previous versions of LS-DYNA.
LCA	Load curve ID, see *DEFINE_CURVE, for sigma-aa versus either relative volume or volumetric strain. See notes below.
LCB	Load curve ID, see *DEFINE_CURVE, for sigma-bb versus either relative volume or volumetric strain. Default LCB=LCA. See notes below.
LCC	Load curve ID, see *DEFINE_CURVE, for sigma-cc versus either relative volume or volumetric strain. Default LCC=LCA. See notes below.
LCS	Load curve ID, see *DEFINE_CURVE, for shear stress versus either relative volume or volumetric strain. Default LCS=LCA. Each component of shear stress may have its own load curve. See notes below.
LCAB	Load curve ID, see *DEFINE_CURVE, for sigma-ab versus either relative volume or volumetric strain. Default LCAB=LCS. See notes below.

<u>VARIABLE</u>	<u>DESCRIPTION</u>
LCBC	Load curve ID, see *DEFINE_CURVE, for sigma-bc versus either relative volume or volumetric strain. Default LCBC=LCS. See notes below.
LCCA	Load curve ID, see *DEFINE_CURVE, or sigma-ca versus either relative volume or volumetric strain. Default LCCA=LCS. See notes below.
LCSR	Load curve ID, see *DEFINE_CURVE, for strain-rate effects defining the scale factor versus strain rate. This is optional. The curves defined above are scaled using this curve.
EAAU	Elastic modulus $E_{aaU}$ in uncompressed configuration.
EBBU	Elastic modulus $E_{bbU}$ in uncompressed configuration.
ECCU	Elastic modulus $E_{ccU}$ in uncompressed configuration.
GABU	Shear modulus $G_{abU}$ in uncompressed configuration.
GBCU	Shear modulus $G_{bcU}$ in uncompressed configuration.
GCAU	Shear modulus $G_{caU}$ in uncompressed configuration.
AOPT	Material axes option: EQ. 0.0: locally orthotropic with material axes determined by element nodes as shown in Figure 20.1. Nodes 1, 2, and 4 of an element are identical to the Nodes used for the definition of a coordinate system as by *DEFINE_COORDINATE_NODES. EQ. 1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center, this is the a-direction. EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR. EQ. 3.0: This option determines locally orthotropic material axes by offsetting the material axes by an angle to be specified from a line in the plane of the 1-2-3-4 segment determined by taking the cross product of the vector $\mathbf{v}$ defined below with the segment normal vector.
XP YP ZP	Coordinates of point p for AOPT = 1.
A1 A2 A3	Components of vector a for AOPT = 2.
D1 D2 D3	Components of vector d for AOPT = 2.
TSEF	Tensile strain at element failure (element will erode).
SSEF	Shear strain at element failure (element will erode).

**Remarks:**

**For efficiency it is strongly recommended that the load curve ID's: LCA, LCB, LCC, LCS, LCAB, LCBC, and LCCA, contain exactly the same number of**

points with corresponding strain values on the abscissa. If this recommendation is followed the cost of the table lookup is insignificant. Conversely, the cost increases significantly if the abscissa strain values are not consistent between load curves.

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

$$E_{aa} = E_{aa0} + \beta(E - E_{aa0})$$

$$E_{bb} = E_{bb0} + \beta(E - E_{bb0})$$

$$E_{cc} = E_{cc0} + \beta(E - E_{cc0})$$

$$G_{ab} = G_{ab0} + \beta(G - G_{ab0})$$

$$G_{bc} = G_{bc0} + \beta(G - G_{bc0})$$

$$G_{ca} = G_{ca0} + \beta(G - G_{ca0})$$

where

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

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

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

The relative volume,  $V$ , is defined as the ratio of the current volume to the initial volume. Typically,  $V=1$  at the beginning of a calculation. The viscosity coefficient  $\mu$  (MU) should be set to a small number (usually .02-.10 is okay). Alternatively, the two bulk viscosity coefficients on the control cards should be set to very small numbers to prevent the development of spurious pressures that may lead to undesirable and confusing results. The latter is not recommended since spurious numerical noise may develop.

The load curves define the magnitude of the average stress as the material changes density (relative volume), see Figure 20.9. Each curve related to this model must have the same number of points and the same abscissa values. There are two ways to define these curves, **a**) as a function of relative volume ( $V$ ) or **b**) as a function of volumetric strain defined as:

$$\varepsilon_v = 1 - V$$



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

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

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

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

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

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

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

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

Each component of the updated stresses is then independently checked to ensure that they do not exceed the permissible values determined from the load curves; e.g., if

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

then

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

On Card 2  $\sigma_{ij}(V)$  is defined by LCA for the aa stress component, LCB for the bb component, LCC for the cc component, and LCS for the ab, bc, cb shear stress components. The parameter  $\lambda$  is either unity or a value taken from the load curve number, LCSR, that defines  $\lambda$  as a function of strain-rate. Strain-rate is defined here as the Euclidean norm of the deviatoric strain-rate tensor.

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

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

where the deviatoric strain increment is defined as

$$\Delta \boldsymbol{\varepsilon}_{ij}^{dev} = \Delta \boldsymbol{\varepsilon}_{ij} - \frac{1}{3} \Delta \boldsymbol{\varepsilon}_{kk} \boldsymbol{\delta}_{ij} .$$

Now a check is made to see if the yield stress for the fully compacted material is exceeded by comparing

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

the effective trial stress to the defined yield stress, SIGY. If the effective trial stress exceeds the yield stress the stress components are simply scaled back to the yield surface

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

Now the pressure is updated using the elastic bulk modulus, K

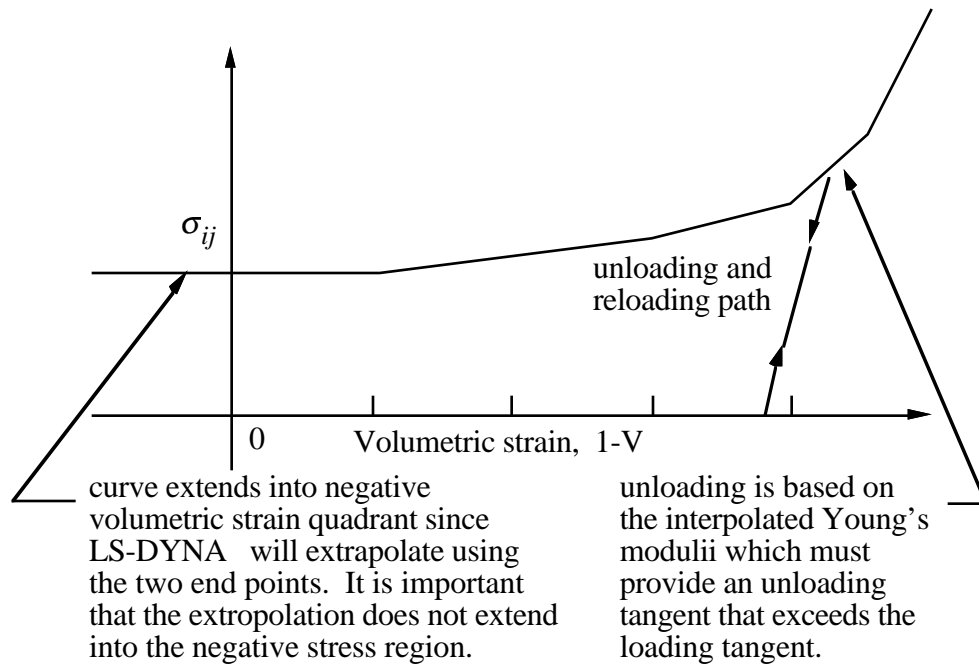
$$p^{n+1} = p^n - K \Delta \boldsymbol{\varepsilon}_{kk}^{n+1/2}$$

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

to obtain the final value for the Cauchy stress

$$\boldsymbol{\sigma}_{ij}^{n+1} = s_{ij}^{n+1} - p^{n+1} \boldsymbol{\delta}_{ij} .$$

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



**Figure 20.9.** Stress quantity versus volumetric strain. Note that the “yield stress” at a volumetric strain of zero is non-zero. In the load curve definition, see \*DEFINE\_CURVE, the “time” value is the volumetric strain and the “function” value is the yield stress.

**\*MAT\_MOONEY-RIVLIN\_RUBBER**

This is Material Type 27. A two-parametric material model for rubber can be defined.

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	PR	A	B	REF		
Type	I	F	F	F	F	F		

Card 2            1            2            3            4            5            6            7            8

Variable	SGL	SW	ST	LCID				
Type	F	F	F	F				

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
PR	Poisson's ratio (> .49 is recommended, smaller values may not work).
A	Constant, see literature and equations defined below.
B	Constant, see literature and equations defined below.
REF	Use reference geometry to initialize the stress tensor. The reference geometry is defined by the keyword: *INITIAL_FOAM_REFERENCE_GEOMETRY. This option is currently restricted to 8-noded solid elements with one point integration. EQ.0.0: off, EQ.1.0: on.

If A=B=0.0, then a least square fit is computed from tabulated uniaxial data via a load curve. The following information should be defined.

SGL            Specimen gauge length  $l_0$ , see Figure 20.10.

VARIABLE	DESCRIPTION
SW	Specimen width, see Figure 20.10.
ST	Specimen thickness, see Figure 20.10.
LCID	Load curve ID, see *DEFINE_CURVE, giving the force versus actual change ΔL in the gauge length. See also Figure 20.11 for an alternative definition.

**Remarks:**

The strain energy density function is defined as:

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

where

$$C = 0.5 A + B$$

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

$\nu$  = Poisson's ratio

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

I, II, III = invariants of right Cauchy-Green Tensor  $C$ .

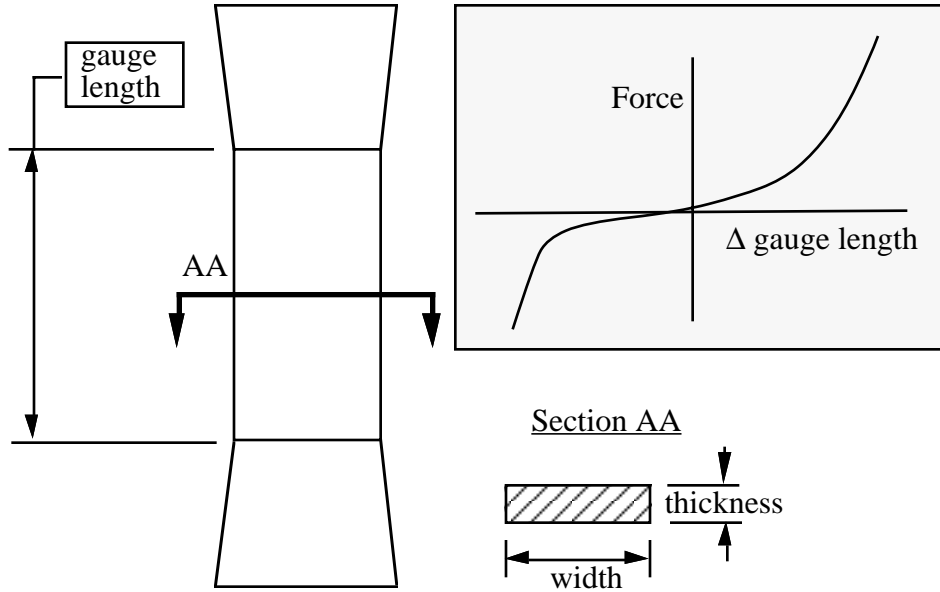
The load curve definition that provides the uniaxial data should give the change in gauge length, ΔL, versus the corresponding force. In compression both the force and the change in gauge length must be specified as negative values. In tension the force and change in gauge length should be input as positive values. The principal stretch ratio in the uniaxial direction, λ<sub>1</sub>, is then given by

$$\lambda_1 = \frac{L_0 + \Delta L}{L_0}$$

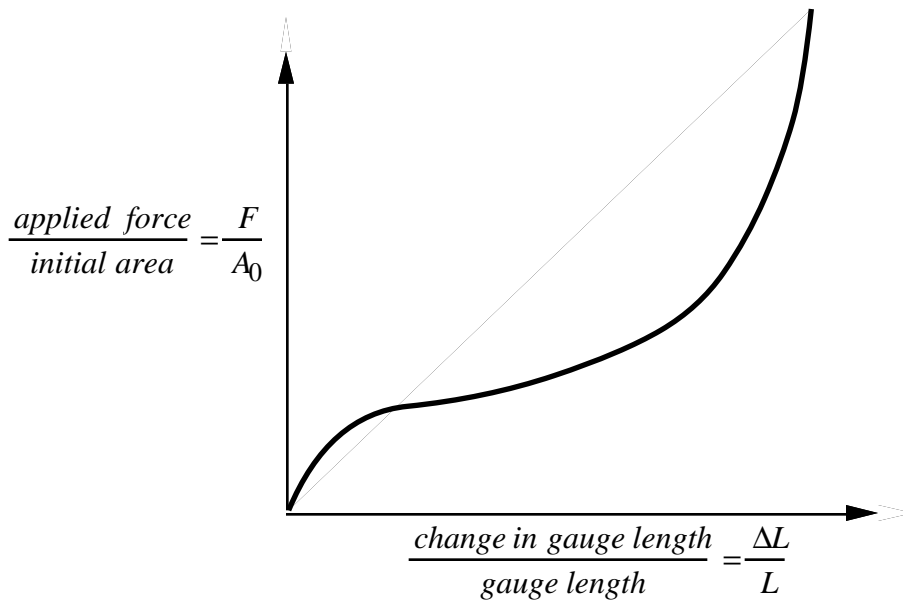
with L<sub>0</sub> being the initial length and L being the actual length.

Alternatively, the stress versus strain curve can also be input by setting the gauge length, thickness, and width to unity (1.0) and defining the engineering strain in place of the change in gauge length and the nominal (engineering) stress in place of the force, see Figure 20.11.

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. It is also advised to use the material driver (see Appendix H) for checking out the material model.



20.10 Uniaxial specimen for experimental data.



**Figure 20.11** The stress versus strain curve can be used instead of the force versus the change in the gauge length by setting the gauge length, thickness, and width to unity (1.0) and defining the engineering strain in place of the change in gauge length and the nominal (engineering) stress in place of the force.

\*MAT\_RESULTANT\_PLASTICITY

This is Material Type 28. A resultant formulation for beam and shell elements including elasto-plastic behavior can be defined. This model is available for the Belytschko-Schwer beam, the C<sup>0</sup> triangular shell, and the Belytschko-Tsay shell. For beams, the treatment is elastic-perfectly plastic, but for shell elements isotropic hardening is approximately modeled. For a detailed description we refer to the Theoretical Manual.

Card Format

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	E	PR	SIGY	ETAN		
Type	I	F	F	F	F	F		
Default	none	none	none	none	none	0.0		

VARIABLE

DESCRIPTION

MID	Material identification. A unique number has to be chosen.
RO	Mass density
E	Young's modulus
PR	Poisson's ratio
SIGY	Yield stress
ETAN	Plastic hardening modulus (for shells only)

**\*MAT\_FORCE\_LIMITED**

This is Material Type 29. With this material model, for the Belytschko-Schwer beam only, plastic hinge forming at the ends of a beam can be modeled using curve definitions. Optionally, collapse can also be modelled.

Description: FORCE LIMITED Resultant Formulation

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	E	PR	DF	AOPT	YTFLAG	ASOFT
Type	I	F	F	F	F	F	F	F
Default	none	none	none	none	0.0	0.0	0.0	0.0

Card 2

Variable	M1	M2	M3	M4	M5	M6	M7	M8
Type	F	F	F	F	F	F	F	F
Default	none	0	0	0	0	0	0	0

Card 3

Variable	LC1	LC2	LC3	LC4	LC5	LC6	LC7	LC8
Type	F	F	F	F	F	F	F	F
Default	none	0	0	0	0	0	0	0



Card 4

Variable	LPS1	SFS1	LPS2	SFS2	YMS1	YMS2		
Type	F	F	F	F	F	F		
Default	0	1.0	LPS1	1.0	1.0E+20	YMS1		

Card 5

Variable	LPT1	SFT1	LPT2	SFT2	YMT1	YMT2		
Type	F	F	F	F	F	F		
Default	0	1.0	LPT1	1.0	1.0E+20	YMT1		

Card 6

Variable	LPR	SFR	YMR					
Type	F	F	F					
Default	0	1.0	1.0E+20					

VARIABLE

DESCRIPTION

- MID            Material identification. A unique number has to be chosen.
- RO            Mass density
- E             Young's modulus
- PR            Poisson's ratio
- DF            Damping factor, see definition in notes below. A proper control for the timestep has to be maintained by the user!

---

<b>VARIABLE</b>	<b>DESCRIPTION</b>
AOPT	Axial load curve option: EQ.0.0: axial load curves are force versus strain, EQ.1.0: axial load curves are force versus change in length .
YTFLAG	Flag to allow beam to yield in tension: EQ.0.0: beam does not yield in tension, EQ.1.0: beam can yield in tension.
ASOFT	Axial elastic softening factor applied once hinge has formed. When a hinge has formed the stiffness is reduced by this factor. If zero, this factor is ignored.
M1, M2,...,M8	Applied end moment for force versus (strain/change in length) curve. At least one must be defined. A maximum of 8 moments can be defined. The values should be in ascending order.
LC1, LC2,...,LC8	Load curve ID (see *DEFINE_CURVE) defining axial force versus strain/change in length (see AOPT) for the corresponding applied end moment. Define the same number as end moments. Each curve must contain the same number of points.
LPS1	Load curve ID for plastic moment versus rotation about s-axis at node 1. If zero, this load curve is ignored.
SFS1	Scale factor for plastic moment versus rotation curve about s-axis at node 1. Default = 1.0.
LPS2	Load curve ID for plastic moment versus rotation about s-axis at node 2. Default: is same as at node 1.
SFS2	Scale factor for plastic moment versus rotation curve about s-axis at node 2. Default: is same as at node 1.
YMS1	Yield moment about s-axis at node 1 for interaction calculations (default set to 1.0E+20 to prevent interaction).
YMS2	Yield moment about s-axis at node 2 for interaction calculations (default set to YMS1).
LPT1	Load curve ID for plastic moment versus rotation about t-axis at node 1. If zero, this load curve is ignored.
SFT1	Scale factor for plastic moment versus rotation curve about t-axis at node 1. Default = 1.0.
LPT2	Load curve ID for plastic moment versus rotation about t-axis at node 2. Default: is the same as at node 1.
SFT2	Scale factor for plastic moment versus rotation curve about t-axis at node 2. Default: is the same as at node 1.

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
YMT1	Yield moment about t-axis at node 1 for interaction calculations (default set to 1.0E+20 to prevent interactions)
YMT2	Yield moment about t-axis at node 2 for interaction calculations (default set to YMT1)
LPR	Load curve ID for plastic torsional moment versus rotation. If zero, this load curve is ignored.
SFR	Scale factor for plastic torsional moment versus rotation (default = 1.0).
YMR	Torsional yield moment for interaction calculations (default set to 1.0E+20 to prevent interaction)

**Remarks:**

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

Axial collapse occurs when the compressive axial load reaches the collapse load. Collapse load versus collapse deflection is specified in the form of a load curve. The points of the load curve are either (true strain, collapse force) or (change in length, collapse force). Both quantities should be entered as positive for all points, and will be interpreted as compressive. The first point should be (zero, initial collapse load).

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

Stiffness-proportional damping may be added using the damping factor  $\lambda$ . This is defined as follows:

$$\lambda = \frac{2 * \xi}{\omega}$$

where  $\xi$  is the damping factor at the reference frequency  $\omega$  (in radians per second). For example if 1% damping at 2Hz is required

$$\lambda = \frac{2 * 0.01}{2 \pi * 2} = 0.001592$$

If damping is used, a small timestep may be required. LS-DYNA does not check this so to avoid instability it may be necessary to control the timestep via a load curve. As a guide, the timestep required for any given element is multiplied by  $0.3L/c\lambda$  when damping is present (L = element length, c = sound speed).

**Moment Interaction:**

Plastic hinges can form due to the combined action of moments about the three axes. This facility is activated only when yield moments are defined in the material input. A hinge forms when the following condition is first satisfied.

$$\left(\frac{M_r}{M_{ryield}}\right)^2 + \left(\frac{M_s}{M_{syield}}\right)^2 + \left(\frac{M_t}{M_{tyield}}\right)^2 \geq 1$$

where,

$M_r, M_s, M_t$  = current moment

$M_{ryield}, M_{syield}, M_{tyield}$  = yield moment

Note that scale factors for hinge behavior defined in the input will also be applied to the yield moments: for example,  $M_{syield}$  in the above formula is given by the input yield moment about the local axis times the input scale factor for the local s axis. For strain-softening characteristics, the yield moment should generally be set equal to the initial peak of the moment-rotation load curve.

On forming a hinge, upper limit moments are set. These are given by

$$M_{rupper} = \text{MAX}\left(M_r, \frac{M_{ryield}}{2}\right)$$

and similar for  $M_s$  and  $M_t$ .

Thereafter the plastic moments will be given by

$$M_{rp} = \min(M_{rupper}, M_{rcurve}) \text{ and similar for s and t}$$

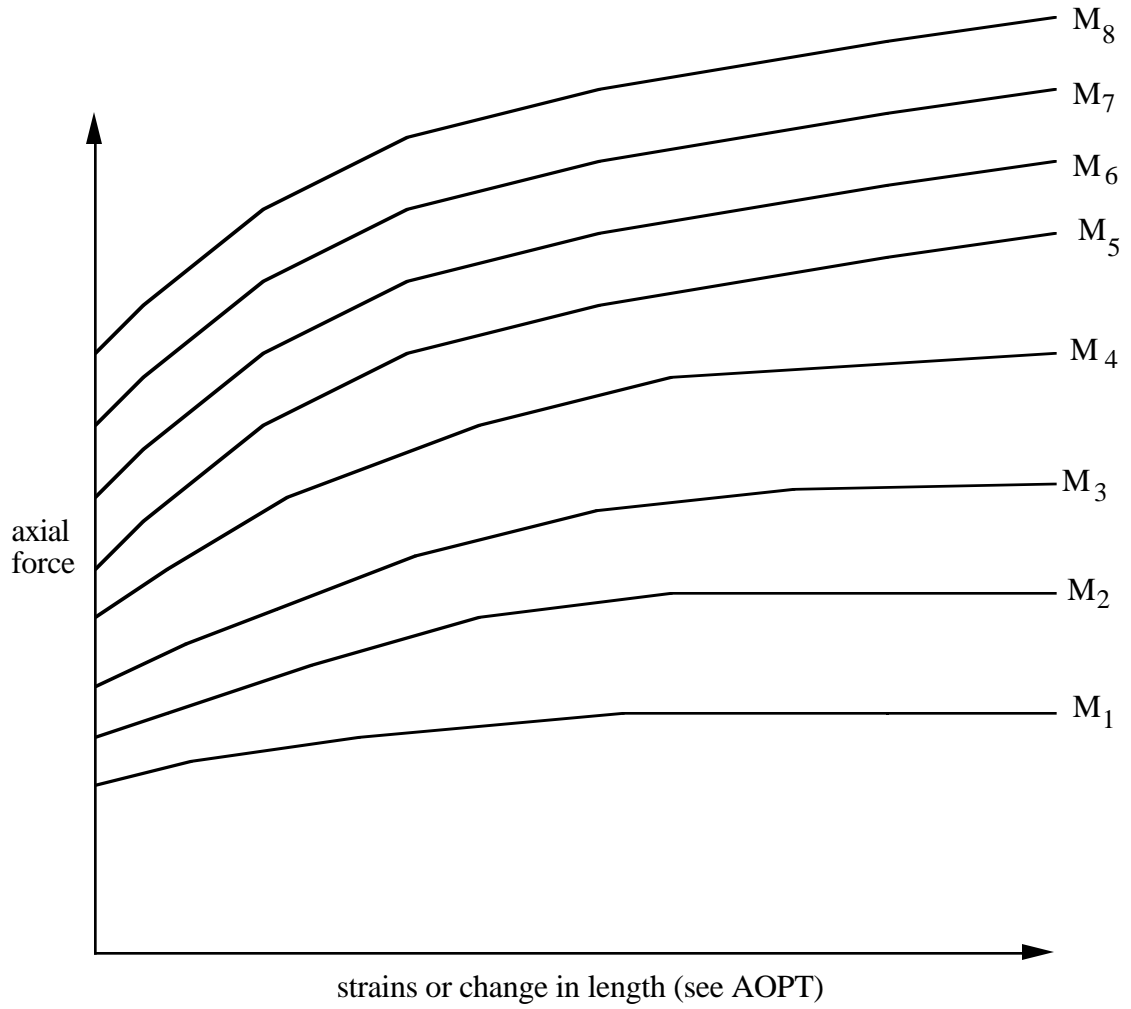
where

$M_{rp}$  = current plastic moment

$M_{rcurve}$  = moment taken from load curve at the current rotation scaled according to the scale factor.

The effect of this is to provide an upper limit to the moment that can be generated; it represents the softening effect of local buckling at a hinge site. Thus if a member is bent about its local s-axis it will then be weaker in torsion and about its local t-axis. For moments-softening curves, the effect is to trim off the initial peak (although if the curves subsequently harden, the final hardening will also be trimmed off).

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



**Figure 20.12.** The force magnitude is limited by the applied end moment. For an intermediate value of the end moment LS-DYNA interpolates between the curves to determine the allowable force value.

**\*MAT\_CLOSED\_FORM\_SHELL\_PLASTICITY**

This is Material Type 30. With this model, a non iterative “exact” treatment of the plane stress constitutive equations for elasto-plastic material can be defined. This model is in general more efficient than a fully iterative treatment. However, on vector computers, the vectorized form (i.e., using only 3 iterations) as optionally available in material type 3 is far more efficient. This model is available for perfect plasticity or kinematic hardening for a bi-linear von Mises model. The implementation is described in [Whirley, Hallquist, and Goudreau 1989].

**Card Format**

	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	SIGY	ETAN		
Type	I	F	F	F	F	F		
Default	none	none	none	none	none	0.0		

**VARIABLE****DESCRIPTION**

MID	Material identification
RO	Density
E	Young’s modulus
PR	Poisson’s ratio
SIGY	Yield stress
ETAN	Plastic tangent modulus

\*MAT\_FRAZER\_NASH\_RUBBER\_MODEL

This is Material Type 31. This model defines rubber from uniaxial test data. It is a modified form of the hyperelastic constitutive law first described in [Kendington 1988]. See also the notes below.

Card Format

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	PR	C100	C200	C300	C400	
Type	I	F	F	F	F	F	F	

Card 2

Variable	C110	C210	C010	C020	EXIT	EMAX	EMIN	REF
Type	F	F	F	F	F	F	F	F

Card 3            1            2            3            4            5            6            7            8

Variable	SGL	SW	ST	LCID				
Type	F	F	F	F				

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MID	Material identification.. A unique number has to be defined.
RO	Mass density.
PR	Poisson's ratio. Values between .49 and .50 are suggested.
C100	C <sub>100</sub> (EQ.1.0 if term is in the least squares fit.).
C200	C <sub>200</sub> (EQ.1.0 if term is in the least squares fit.).

<b>VARIABLE</b>	<b>DESCRIPTION</b>
C300	C <sub>300</sub> (EQ.1.0 if term is in the least squares fit.).
C400	C <sub>400</sub> (EQ.1.0 if term is in the least squares fit.).
C110	C <sub>110</sub> (EQ.1.0 if term is in the least squares fit.).
C210	C <sub>210</sub> (EQ.1.0 if term is in the least squares fit.).
C010	C <sub>010</sub> (EQ.1.0 if term is in the least squares fit.).
C020	C <sub>020</sub> (EQ.1.0 if term is in the least squares fit.).
EXIT	Exit option: EQ. 0.0: stop if strain limits are exceeded (recommended), NE. 0.0: continue if strain limits are exceeded. The curve is then extrapolated.
EMAX	Maximum strain limit, (Green-St, Venant Strain).
EMIN	Minimum strain limit, (Green-St, Venant Strain).
REF	Use reference geometry to initialize the stress tensor. The reference geometriy is defined by the keyword:*INITIAL_FOAM_REFERENCE_GEOMETRY. This option is currently restricted to 8-noded solid elements with one point integration. EQ.0.0: off, EQ.1.0: on.
SGL	Specimen gauge length, see Figure 20.10.
SW	Specimen width, see Figure 20.10.
ST	Specimen thickness, see Figure 20.10.
LCID	Load curve ID, see DEFINE_CURVE, giving the force versus actual change in gauge length. See also Figure 20.11 for an alternative definition.

**Remarks:**

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

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

$$U = C_{100} I_1 + C_{200} I_1^2 + C_{300} I_1^3 + C_{400} I_1^4 + C_{110} I_1 I_2 + C_{210} I_1^2 I_2 + C_{010} I_2 + C_{020} I_2^2 + f(J)$$



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

$$J = |F_{ij}|$$
$$I_1 = E_{ii}$$
$$I_2 = \frac{1}{2!} \delta_{pq}^{ij} E_{pi} E_{qj}$$

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

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

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

The load curve definition that provides the uniaxial data should give the change in gauge length,  $\Delta L$ , and the corresponding force . In compression both the force and the change in gauge length must be specified as negative values. In tension the force and change in gauge length should be input as positive values. The principal stretch ratio in the uniaxial direction,  $\lambda_1$ , is then given by

$$\lambda_1 = \frac{L_o + \Delta L}{L_o}$$

Alternatively, the stress versus strain curve can also be input by setting the gauge length, thickness, and width to unity and defining the engineering strain in place of the change in gauge length and the nominal (engineering) stress in place of the force, see 20.11.

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

**\*MAT\_LAMINATED\_GLASS**

This is Material Type 32. With this material model, a layered glass including polymeric layers can be modeled. Failure of the glass part is possible. See notes below.

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	EG	PRG	SYG	ETG	EFG	EP
Type	I	F	F	F	F	F	F	F

Card 2

Variable	PRP	SYP	ETP					
Type	F	F	F					

**Card Format. Define 1-4 cards with a maximum of 32 number. If less than 4 cards are input, reading is stopped by a “\*” control card.**

Card 3, etc.

Variable	F1	F2	F3	F4	F5	F6	F7	F8
Type	F	F	F	F	F	F	F	F

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MID	Material identification. A unique number has to be defined.
RO	Mass density
EG	Young’s modulus for glass
PRG	Poisson’s ratio for glass

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
SYG	Yield stress for glass
ETG	Plastic hardening modulus for glass
EFG	Plastic strain at failure for glass
EP	Young's modulus for polymer
PRP	Poisson's ratio for polymer
SYP	Yield stress for polymer
ETP	Plastic hardening modulus for polymer
F1,..FN	Integration point material: $f_n = 0.0$ : glass, $f_n = 1.0$ : polymer. A user-defined integration rule must be specified, see *INTEGRATION_SHELL.

**Remarks:**

Isotropic hardening for both materials is assumed. The material to which the glass is bonded is assumed to stretch plastically without failure. A user defined integration rule specifies the thickness of the layers making up the glass.  $F_i$  defines whether the integration point is glass (0.0) or polymer (1.0). The material definition,  $F_i$ , has to be given for the same number of integration points (NIPTS) as specified in the rule. A maximum of 32 layers is allowed.

**\*MAT\_BARLAT\_ANISOTROPIC\_PLASTICITY**

This is Material Type 33. This model was developed by Barlat, Lege, and Brem [1991] for modelling anisotropic material behavior in forming processes. The finite element implementation of this model is described in detail by Chung and Shah [1992] and is used here. It is based on a six parameter model, which is ideally suited for 3D continuum problems, see notes below. For sheet forming problems, material 36 based on a 3-parameter model is recommended.

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	E	PR	K	E0	N	M
Type	I	F	F	F	F	F	F	F

Card 2

Variable	A	B	C	F	G	H		
Type	F	F	F	F	F	F		

Card 3

Variable	AOPT	OFFANG						
Type	F	F						

Card 4

Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card 5

Variable	V1	V2	V3	D1	D2	D3		
Type	F	F	F	F	F	F		

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
E	Young's modulus, E.
PR	Poisson's ratio, $\nu$ .
K	k, strength coefficient, see notes below.
EO	$\epsilon_0$ , strain corresponding to the initial yield, see notes below.
N	n, hardening exponent for yield strength.
M	m, flow potential exponent in Barlat's Model.
A	a, anisotropy coefficient in Barlat's Model.
B	b, anisotropy coefficient in Barlat's Model.
C	c anisotropy coefficient in Barlat's Model.
F	f, anisotropy coefficient in Barlat's Model.
G	g, anisotropy coefficient in Barlat's Model.
H	h, anisotropy coefficient in Barlat's Model.
AOPT	Material axes option: EQ. 0.0: locally orthotropic with material axes determined by element nodes as shown in Figure 20.1. Nodes 1, 2, and 4 of an element are identical to the Nodes used for the definition of a coordinate system as by *DEFINE_COORDINATE_NODES. EQ. 1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center, this is the a-direction.

<u>VARIABLE</u>	<u>DESCRIPTION</u>
	EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR. EQ. 3.0: applicable to shell elements only. This option determines locally orthotropic material axes by offsetting the material axes by an angle, OFFANG, to be specified from a line in the plane of the shell determined by taking the cross product of the vector <b>v</b> defined below with the shell normal vector.
OFFANG	Offset angle for AOPT = 3.
XP YP ZP	Coordinates of point p for AOPT = 1.
A1 A2 A3	Components of vector a for AOPT = 2.
V1 V2 V3	Components of vector v for AOPT = 3.
D1 D2 D3	Components of vector d for AOPT = 2.

**Remarks:**

The yield function  $\Phi$  is defined as:

$$\Phi = |S_1 - S_2|^m + |S_2 - S_3|^m + |S_3 - S_1|^m = 2\bar{\sigma}^m$$

where  $\bar{\sigma}$  is the effective stress and  $S_{i=1,2,3}$  are the principal values of the symmetric matrix  $S_{\alpha\beta}$ ,

$$S_{xx} = [c(\sigma_{xx} - \sigma_{yy}) - b(\sigma_{zz} - \sigma_{xx})]/3$$

$$S_{yy} = [a(\sigma_{yy} - \sigma_{zz}) - c(\sigma_{xx} - \sigma_{yy})]/3$$

$$S_{zz} = [b(\sigma_{zz} - \sigma_{xx}) - a(\sigma_{yy} - \sigma_{zz})]/3$$

$$S_{yz} = f\sigma_{yz}$$

$$S_{zx} = g\sigma_{zx}$$

$$S_{xy} = h\sigma_{xy}$$

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 face centered cubic (FCC) materials  $m=8$  is recommended and for body centered cubic (BCC) materials  $m = 6$  is used. The yield strength of the material is

$$\sigma_y = k (\epsilon^p + \epsilon_0)^n$$

where  $\epsilon_0$  is the strain corresponding to the initial yield stress and  $\epsilon^p$  is the plastic strain.

\*MAT\_BARLAT\_YLD96

This is Material Type 33. This model was developed by Barlat, Maeda, Chung, Yanagawa, Brem, Hayashida, Lege, Matsui, Murtha, Hattori, Becker, and Makosey [1997] for modeling anisotropic material behavior in forming processes in particular for aluminum alloys. This model is available for shell elements only.

Card Format

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	E	PR	K			
Type	I	F	F	F	F			

Card 2

Variable	E0	N	ESR0	M	HARD	A		
Type	F	F	F	F	F	F		

Card 2

Variable	C1	C2	C3	C4	AX	AY	AZ0	AZ1
Type	F	F	F	F	F	F	F	F

Card 4

Variable	AOPT	OFFANG						
Type	F	F						

Card 5

Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card 6

Variable	V1	V2	V3	D1	D2	D3		
Type	F	F	F	F	F	F		

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
E	Young's modulus, E.
PR	Poisson's ratio, $\nu$ .
K	$k$ , strength coefficient or $a$ in Voce, see notes below.
EO	$\epsilon_0$ , strain corresponding to the initial yield or $b$ in Voce, see notes below.
N	$n$ , hardening exponent for yield strength or $c$ in Voce.
ESR0	$\epsilon_{SR0}$ , in powerlaw rate sensitivity.
M	$m$ , exponent for strain rate effects
HARD	Hardening option: LT. 0.0: absolute value defines the load curve ID. EQ. 1.0: powerlaw EQ. 2.0: Voce
A	Flow potential exponent.
C1	$c1$ , see equations below.
C2	$c2$ , see equations below.



VARIABLE	DESCRIPTION
C3	c3, see equations below.
C4	c4, see equations below.
AX	ax, see equations below.
AY	ay, see equations below.
AZ0	az0, see equations below.
AZ1	az1, see equations below.
AOPT	<p>Material axes option:</p> <p>EQ. 0.0: locally orthotropic with material axes determined by element nodes as shown in Figure 20.1. Nodes 1, 2, and 4 of an element are identical to the Nodes used for the definition of a coordinate system as by *DEFINE_COORDINATE_NODES.</p> <p>EQ. 1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center, this is the a-direction.</p> <p>EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR.</p> <p>EQ. 3.0: applicable to shell elements only. This option determines locally orthotropic material axes by offsetting the material axes by an angle, OFFANG, to be specified from a line in the plane of the shell determined by taking the cross product of the vector <b>v</b> defined below with the shell normal vector.</p>
OFFANG	Offset angle for AOPT = 3.
XP YP ZP	Coordinates of point p for AOPT = 1.
A1 A2 A3	Components of vector a for AOPT = 2.
V1 V2 V3	Components of vector v for AOPT = 3.
D1 D2 D3	Components of vector d for AOPT = 2.

### **Remarks:**

The yield stress  $\sigma_y$  is defined three ways. The first, the Swift equation, is given in terms of the input constants as:

$$\sigma_y = k(\varepsilon_0 + \varepsilon^p)^n \left( \frac{\dot{\varepsilon}}{\varepsilon_{SR0}} \right)^m$$

The second, the Voce equation, is defined as:

$$\sigma_y = a - be^{-c\epsilon^p}$$

and the third option is to give a load curve ID that defines the yield stress as a function of effective plastic strain. The yield function  $\Phi$  is defined as:

$$\Phi = \alpha_1 |s_1 - s_2|^a + \alpha_2 |s_2 - s_3|^a + \alpha_3 |s_3 - s_1|^a = 2\sigma_y^a$$

where  $s_i$  is a principle component of the deviatoric stress tensor where in vector notation:

$$\underset{\sim}{s} = \underset{\sim}{L} \underset{\sim}{\sigma}$$

and  $\underset{\sim}{L}$  is given as

$$\underset{\sim}{L} = \begin{bmatrix} \frac{c_1 + c_3}{3} & \frac{-c_3}{3} & \frac{-c_2}{3} & 0 \\ \frac{-c_3}{3} & \frac{c_3 + c_1}{3} & \frac{-c_2}{3} & 0 \\ \frac{-c_2}{3} & \frac{-c_2}{3} & \frac{c_1 + c_2}{3} & 0 \\ 0 & 0 & 0 & c_4 \end{bmatrix}$$

A coordinate transformation relates the material frame to the principle directions of  $\underset{\sim}{s}$  is used to obtain the  $\alpha_k$  coefficients consistent with the rotated principle axes:

$$\alpha_k = \alpha_x p_{1k}^2 + \alpha_y p_{2k}^2 + \alpha_z p_{3k}^2$$

$$\alpha_z = \alpha_{z0} \cos^2 2\beta + \alpha_{z1} \sin^2 2\beta$$

where  $p_{ij}$  are components of the transformation matrix. The angle  $\beta$  defines a measure of the rotation between the frame of the principal value of  $\underset{\sim}{s}$  and the principal anisotropy axes.

\*MAT\_FABRIC

This is Material Type 34. This material is especially developed for airbag materials. The fabric model is a variation on the layered orthotropic composite model of material 22 and is valid for 3 and 4 node membrane elements only. In addition to being a constitutive model, this model also invokes a special membrane element formulation which is more suited to the deformation experienced by fabrics under large deformation. For thin fabrics, buckling can result in an inability to support compressive stresses; thus a flag is included for this option. A linearly elastic liner is also included which can be used to reduce the tendency for these elements to be crushed when the no-compression option is invoked. In LS-DYNA versions after 931 the isotropic elastic option is available.

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	EA	EB	EC	PRBA	PRCA	PRCB
Type	I	F	F	F	F	F	F	F

Card 2

Variable	GAB	GBC	GCA	CSE	EL	PRL	LRATIO	DAMP
Type	F	F	F	F	F	F	F	F
Remarks				1	2	2	2	

Card 3

Variable	AOPT	FLC	FAC	ELA	LNRC	FORM		
Type	F	F	F	F	F	F		
Remarks		3	3		4	0		

Card 4

Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card 5

Variable	V1	V2	V3	D1	D2	D3		
Type	F	F	F	F	F	F		

**VARIABLE****DESCRIPTION**

MID	Material identification. A unique number has to be chosen.
RO	Mass density.
EA	Young's modulus - longitudinal direction. For an isotropic elastic fabric material only EA and PRBA are defined and are used as the isotropic Young's modulus and Poisson's ratio, respectively. The input for the fiber directions and liner should be input as zero for the isotropic elastic fabric.
EB	Young's modulus - transverse direction, set to zero for isotropic elastic material.
EC	Young's modulus - normal direction, set to zero for isotropic elastic material.
PRBA	$\nu_{ba}$ , Poisson's ratio ba direction.
PRCA	$\nu_{ca}$ , Poisson's ratio ca direction, set to zero for isotropic elastic material.
PRCB	$\nu_{cb}$ , Poisson's ratio cb direction, set to zero for isotropic elastic material.
GAB	$G_{ab}$ , shear modulus ab direction, set to zero for isotropic elastic material.
GBC	$G_{bc}$ , shear modulus bc direction, set to zero for isotropic elastic material.
GCA	$G_{ca}$ , shear modulus ca direction, set to zero for isotropic elastic material.

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
CSE	Compressive stress elimination option (default 0.0): EQ.0.0: don't eliminate compressive stresses, EQ.1.0: eliminate compressive stresses (This option does not apply to the liner).
EL	Young's modulus for elastic liner (optional).
PRL	Poisson's ratio for elastic liner (optional).
LRATIO	Ratio of liner thickness to total fabric thickness.
DAMP	Rayleigh damping coefficient. A 0.05 coefficient is recommended corresponding to 5% of critical damping. Sometimes larger values are necessary.
AOPT	Material axes option: EQ. 0.0: locally orthotropic with material axes determined by element nodes as shown in Figure 20.1. Nodes 1, 2, and 4 of an element are identical to the Nodes used for the definition of a coordinate system as by *DEFINE_COORDINATE_NODES. EQ. 1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center, this is the a-direction. EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR. EQ. 3.0: applicable to shell elements only. This option determines locally orthotropic material axes by offsetting the material axes by an angle to be specified from a line in the plane of the shell determined by taking the cross product of the vector $\mathbf{v}$ defined below with the shell normal vector.
FLC	Fabric leakage coefficient (optional), FLC LT.0.0:  FLC  is the load curve ID of the curve defining FLC versus time. See notes below.
FAC	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 remark 3 below.
ELA	Effective leakage area for blocked fabric, ELA, LT.0.0:  ELA  is the load curve ID of the curve defining ELA versus time. The default value of zero assumes that no leakage occurs. A value of .10 would assume that 10% of the blocked fabric is leaking gas.
LNRC	Flag to turn off compression in liner until the reference geometry is reached, i.e., the fabric element becomes tensile. EQ.0.0: off. EQ.1.0: on.

---

---

<b>VARIABLE</b>	<b>DESCRIPTION</b>
FORM	Flag to modify membrane formulation for fabric material: EQ.0.0:default. Least costly and very reliable. EQ.1.0:invariant local membrane coordinate system EQ.2.0:Green-Lagrange strain formulation (available for testing but not yet recommended.)
XP YP ZP	Coordinates of point p for AOPT = 1.
A1 A2 A3	Components of vector a for AOPT = 2.
V1 V2 V3	Components of vector v for AOPT = 3.
D1 D2 D3	Components of vector d for AOPT = 2.

**Remarks:**

1. The no compression option allows the simulation of airbag inflation with far less elements than would be needed for the discretization of the wrinkles which would occur for the case when compressive stresses are not eliminated.
2. When using this material for the analysis of membranes as airbags it is well known from classical theory that only one layer has to be defined. The so-called elastic liner has to be defined for numerical purposes only when the no compression option is invoked.
3. The parameters FLC and FAC are optional for the Wang-Nefske inflation models. It is possible for the airbag to be constructed of multiple fabrics having different values for porosity and permeability. The leakage of gas through the fabric in an airbag then requires an accurate determination of the areas by part ID available for leakage. The leakage area may change over time due to stretching of the airbag fabric or blockage when the bag contacts the structure. LS-DYNA can check the interaction of the bag with the structure and split the areas into regions that are blocked and unblocked depending on whether the regions are in or not in contact, respectively. Typically, FLC and FAC must be determined experimentally and their variation in time with pressure are optional to allow for maximum flexibility.
4. The elastic backing layer always acts in tension and compression since the tension cutoff option, CSE, does not apply. This can sometimes cause difficulties if the elements are very small in relationship to their actual size as defined by the reference geometry (See \*AIRBAG\_REFERENCE\_GEOMETRY.). If the flag, LNRC, is set to 1.0 the elastic liner does not begin to act until the area of defined by the reference geometry is reached.

\*MAT\_PLASTIC\_GREEN-NAGHDI\_RATE

This is Material Type 35. This model is available only for brick elements and is similar to model 3, but uses the Green-Naghdi Rate formulation rather than the Jaumann rate for the stress update. For some cases this might be helpful. This model also has a strain rate dependency following the Cowper-Symonds model.

Card Format

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	E	PR				
Type	I	F	F	F				

Card 2

Variable	SIGY	ETAN	SRC	SRP	BETA			
Type	F	F	F	F	F			

VARIABLE

DESCRIPTION

MID	Material identification
RO	Density
E	Young's modulus
PR	Poisson's ratio
SIGY	Yield stress
ETAN	Plastic hardening modulus
SRC	Strain rate parameter, C
SRP	Strain rate parameter, P
BETA	Hardening parameter, $0 < \beta' < 1$

**\*MAT\_3-PARAMETER\_BARLAT**

This is Material Type 36. This model was developed by Barlat and Lian [1989] for modelling sheets with anisotropic materials under plane stress conditions. This material allows the use of the Lankford parameters for the definition of the anisotropy. This particular development is due to Barlat and Lian [1989].

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	E	PR	HR	P1	P2	
Type	I	F	F	F	F	F	F	

Card 2

Variable	M	R00	R45	R90	LCID	E0	SPI	
Type	F	F	F	F	I	F	F	

Card 3

Variable	AOPT							
Type	F							

Card 4

Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		



Card 5

Variable	V1	V2	V3	D1	D2	D3		
Type	F	F	F	F	F	F		

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
E	Young's modulus, E
PR	Poisson's ratio, $\nu$
HR	Hardening rule: EQ.1.0: linear (default), EQ.2.0: exponential. EQ.3.0: load curve
P1	Material parameter: HR.EQ.1.0: Tangent modulus, HR.EQ.2.0: k, strength coefficient for exponential hardening
P2	Material parameter: HR.EQ.1.0: Yield stress HR.EQ.2.0: n, exponent
M	m, exponent in Barlat's yield surface
R00	$R_{00}$ , Lankford parameter determined from experiments
R45	$R_{45}$ , Lankford parameter determined from experiments
R90	$R_{90}$ , Lankford parameter determined from experiments
LCID	load curve ID for the load curve hardening rule
E0	$\epsilon_0$ for determining initial yield stress for exponential hardening. (Default=0.0)
SPI	$spi$ , if $\epsilon_0$ is zero above. (Default=0.0) EQ.0.0: $\epsilon_0 = (E / k) ** [1 / (n - 1)]$ LE..02: $\epsilon_0 = spi$ GT..02: $\epsilon_0 = (spi / k) ** [1 / n]$

VARIABLE	DESCRIPTION
AOPT	<p>Material axes option:</p> <p>EQ. 0.0: locally orthotropic with material axes determined by element nodes as shown in Figure 20.1. Nodes 1, 2, and 4 of an element are identical to the Nodes used for the definition of a coordinate system as by *DEFINE_COORDINATE_NODES.</p> <p>EQ. 1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center, this is the a-direction.</p> <p>EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR.</p> <p>EQ. 3.0: applicable to shell elements only. This option determines locally orthotropic material axes by offsetting the material axes by an angle to be specified from a line in the plane of the shell determined by taking the cross product of the vector <b>v</b> defined below with the shell normal vector.</p>
XP YP ZP	Coordinates of point p for AOPT = 1.
A1 A2 A3	Components of vector a for AOPT = 2.
V1 V2 V3	Components of vector v for AOPT = 3.
D1 D2 D3	Components of vector d for AOPT = 2.

**Remarks:**

The anisotropic yield criterion  $\Phi$  for plane stress is defined as:

$$\Phi = a|K_1 + K_2|^m + a|K_1 - K_2|^m + c|2K_2|^m = 2\sigma_Y^m$$

where  $\sigma_Y$  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  $R_{90}$ :

$$a = 2 - 2\sqrt{\frac{R_{00}}{1 + R_{00}} \frac{R_{90}}{1 + R_{90}}} \quad c = 2 - a$$

$$h = \sqrt{\frac{R_{00}}{1 + R_{00}} \frac{1 + R_{90}}{R_{90}}}$$

The anisotropy parameter  $p$  is calculated implicitly. According to Barlat and Lian the  $R$  value, width to thickness strain ratio, for any angle  $\phi$  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  $\sigma_\phi$  is the uniaxial tension in the  $\phi$  direction. This expression can be used to iteratively calculate the value of  $p$ . Let  $\phi=45$  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 face centered cubic (FCC) materials  $m=8$  is recommended and for body centered cubic (BCC) materials  $m=6$  may be used. The yield strength of the material can be expressed in terms of  $k$  and  $n$ :

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

where  $\varepsilon_{yp}$  is the elastic strain to yield and  $\bar{\varepsilon}^p$  is the effective plastic strain (logarithmic). If SIGY is set to zero, the strain to yield is 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:

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

# \*MAT

## \*MAT\_TRANSVERSELY\_ANISOTROPIC\_ELASTIC\_PLASTIC

### \*MAT\_TRANSVERSELY\_ANISOTROPIC\_ELASTIC\_PLASTIC

This is Material Type 37. This model is for simulating sheet forming processes with anisotropic material. Only transverse anisotropy can be considered. Optionally an arbitrary dependency of stress and effective plastic strain can be defined via a load curve. This plasticity model is fully iterative and is available only for shell elements. Also see the notes below.

#### Card Format

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	E	PR	SIGY	ETAN	R	HLCID
Type	I	F	F	F	F	F	F	F

#### VARIABLE

#### DESCRIPTION

MID	Material identification. A unique number has to be chosen.
RO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.
SIGY	Yield stress.
ETAN	Plastic hardening modulus.
R	Anisotropic hardening parameter.
HLCID	Load curve ID defining effective yield stress versus effective plastic strain.

#### **Remarks:**

Consider Cartesian reference axes which are parallel to the three symmetry planes of anisotropic behavior. Then, the yield function suggested by [Hill 1948] can be written

$$F(\sigma_{22} - \sigma_{33})^2 + G(\sigma_{33} - \sigma_{11})^2 + H(\sigma_{11} - \sigma_{22})^2 + 2L\sigma_{23}^2 + 2M\sigma_{31}^2 + 2N\sigma_{12}^2 - 1 = 0$$

where  $\sigma_{y1}$ ,  $\sigma_{y2}$ , and  $\sigma_{y3}$ , are the tensile yield stresses and  $\sigma_{y12}$ ,  $\sigma_{y23}$ , and  $\sigma_{y31}$  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  $F = G = H = \frac{1}{2\sigma_y^2}$

and  $L = M = N = \frac{3}{2\sigma_y^2}$ .

For the particular case of transverse anisotropy, where properties do not vary in the  $x_1$ - $x_2$  plane, the following relations hold:

$$2F = 2G = \frac{1}{\sigma_{y3}^2}$$

$$2H = \frac{2}{\sigma_y^2} - \frac{1}{\sigma_{y3}^2}$$

$$N = \frac{2}{\sigma_y^2} - \frac{1}{2} \frac{1}{\sigma_{y3}^2}$$

where it has been assumed that  $\sigma_{y1} = \sigma_{y2} = \sigma_y$ .

Letting  $K = \frac{\sigma_y}{\sigma_{y3}}$ , the yield criteria can be written

$$F(\boldsymbol{\sigma}) = \sigma_e = \sigma_y,$$

where

$$F(\sigma) \equiv \left[ \sigma_{11}^2 + \sigma_{22}^2 + K^2 \sigma_{33}^2 - K^2 \sigma_{33} (\sigma_{11} + \sigma_{22}) - (2 - K^2) \sigma_{11} \sigma_{22} + 2L\sigma_y^2 (\sigma_{23}^2 + \sigma_{31}^2) + 2 \left( 2 - \frac{1}{2} K^2 \right) \sigma_{12}^2 \right]^{1/2}$$

The rate of plastic strain is assumed to be normal to the yield surface so  $\dot{\epsilon}_{ij}^p$  is found from

$$\dot{\epsilon}_{ij}^p = \lambda \frac{\partial F}{\partial \sigma_{ij}}$$

Now consider the case of plane stress, where  $\sigma_{33} = 0$ . Also, define the anisotropy input parameter, R, as the ratio of the in-plane plastic strain rate to the out-of-plane plastic strain rate,

$$R = \frac{\dot{\epsilon}_{22}^p}{\dot{\epsilon}_{33}^p}$$

It then follows that

$$R = \frac{2}{K^2} - 1$$

Using the plane stress assumption and the definition of R, the yield function may now be written

$$F(\sigma) = \left[ \sigma_{11}^2 + \sigma_{22}^2 - \frac{2R}{R+1} \sigma_{11} \sigma_{22} + 2 \frac{2R+1}{R+1} \sigma_{12}^2 \right]^{1/2}$$

Note that there are several differences between this model and other plasticity models for shell elements such as the model, MAT\_PIECEWISE\_LINEAR\_PLASTICITY. First, the yield function for plane stress does not include the transverse shear stress components which are updated elastically, and, secondly, this model is always fully iterative. Consequently, in comparing results for the isotropic case where R=1.0 with other isotropic model, differences in the results are expected, even though they are usually insignificant.

\*MAT\_BLATZ-KO\_FOAM

This is Material Type 38. This model is for the definition of rubber like foams of polyurethane. It is a simple one-parameter model with a fixed Poisson's ratio of .25.

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	G	REF				
Type	I	F	F	F				

VARIABLE

DESCRIPTION

MID	Material identification. A unique number has to be chosen.
RO	Mass density.
G	Shear modulus.
REF	Use reference geometry to initialize the stress tensor. The reference geometry is defined by the keyword: *INITIAL_FOAM_REFERENCE_GEOMETRY. This option is currently restricted to 8-noded solid elements with one point integration. EQ.0.0: off, EQ.1.0: on.

**Remarks:**

The strain energy functional for the compressible foam model is given by

$$W = \frac{G}{2} \left( \frac{II}{III} + 2\sqrt{III} - 5 \right)$$

Blatz and Ko [1962] suggested this form for a 47 percent volume polyurethane foam rubber with a Poisson's ratio of 0.25. In terms of the strain invariants, I, II, and III, the second Piola-Kirchhoff stresses are given as

$$S^{ij} = G \left[ \left( I\delta_{ij} - C_{ij} \right) \frac{1}{III} + \left( \sqrt{III} - \frac{II}{III} \right) C_{ij}^{-1} \right]$$

where  $C_{ij}$  is the right Cauchy-Green strain tensor. This stress measure is transformed to the Cauchy stress,  $\sigma_{ij}$ , according to the relationship

$$\sigma^{ij} = III^{-1/2} F_{ik} F_{jl} S_{lk}$$

where  $F_{ij}$  is the deformation gradient tensor.



\*MAT\_FLD\_TRANSVERSELY\_ANISOTROPIC

This is Material Type 39. This model is for simulating sheet forming processes with anisotropic material. Only transverse anisotropy can be considered. Optionally, an arbitrary dependency of stress and effective plastic strain can be defined via a load curve. A flow limit diagram can be defined using a curve and is used to compute the maximum strain ratio which can be plotted in LS-TAURUS. This plasticity model is fully iterative and is available only for shell elements. Also see the notes below.

Card Format

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	E	PR	SIGY	ETAN	R	HLCID
Type	I	F	F	F	F	F	F	F

Card 2

Variable	LCIDFLD							
Type	F							

VARIABLE

DESCRIPTION

- MID            Material identification. A unique number has to be chosen.
- RO            Mass density.
- E            Young's modulus.
- PR            Poisson's ratio.
- SIGY          Yield stress.
- ETAN          Plastic hardening modulus, see notes for model 37.
- R            Anisotropic hardening parameter, see notes for model 37.

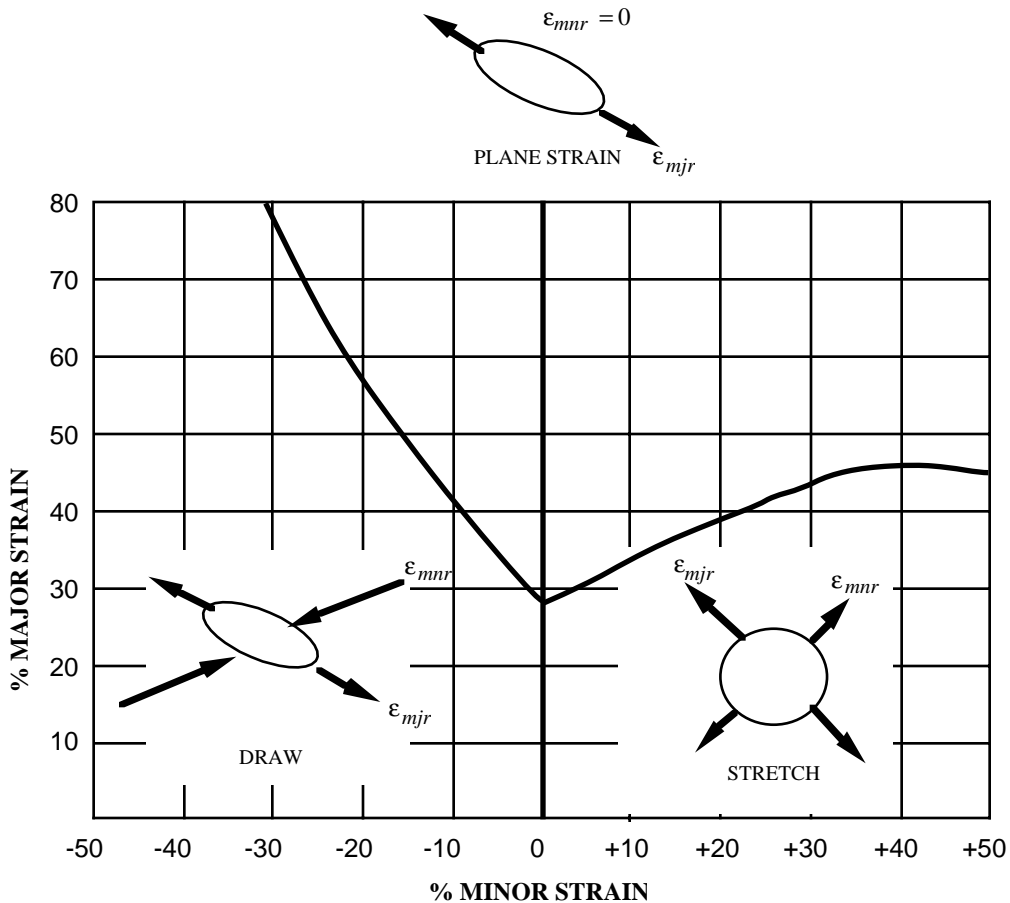
VARIABLE	DESCRIPTION
HLCID	Load curve ID defining effective stress versus effective plastic strain. The yield stress and hardening modulus are ignored with this option.
LCIDFLD	Load curve ID defining the Flow Limit Diagram. Minor strains in percent are defined as abscissa values and Major strains in percent are defined as ordinate values. The flow limit diagram is shown in Figure 20.13. In defining the curve list pairs of minor and major strains starting with the left most point and ending with the right most point, see *DEFINE_CURVE.

**Remarks:**

See material model 37 for the theoretical basis. The first history variable is the maximum strain ratio defined by:

$$\frac{\epsilon_{major_{workpiece}}}{\epsilon_{major_{fld}}}$$

corresponding to  $\epsilon_{minor_{workpiece}}$



**Figure 20.13.** Flow limit diagram.

\*MAT\_NONLINEAR\_ORTHOTROPIC

This is Material Type 40. This model allows the definition of an orthotropic nonlinear elastic material based on a finite strain formulation with the initial geometry as the reference. Failure is optional with two failure criteria available. Optionally, stiffness proportional damping can be defined. In the stress initialization phase, temperatures can be varied to impose the initial stresses. This model is only available for shell elements. We do not recommend using this model at this time since it can be unstable especially if the stress-strain curves increase in stiffness with increasing strain.

Card Format

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	EA	EB	EC	PRBA	PRCA	PRCB
Type	I	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	none	none

Card 2            1            2            3            4            5            6            7            8

Variable	GAB	GBC	GCA	DT	TRAMP	ALPHA		
Type	F	F	F	F	F	F		
Default	none	none	none	0	0	0		

Card 3            1            2            3            4            5            6            7            8

Variable	LCIDA	LCIDB	EFAIL	DFAIL	CDAMP	AOPT		
Type	F	F	F	F	F	F		
Default	0.0	0.0	0.0	0.0	0.0	0.0		

Card 4            1            2            3            4            5            6            7            8

Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card 5            1            2            3            4            5            6            7            8

Variable	V1	V2	V3	D1	D2	D3		
Type	F	F	F	F	F	F		

Card 6            1            2            3            4            5            6            7            8

Variable	LCIDC	LCIDAB	LCIDBC	LCIDCA				
Type	F	F	F	F				
Default	optional	optional	optional	optional				

**VARIABLE****DESCRIPTION**

MID	Material identification. A unique number has to be chosen.
RO	Mass density.
EA	$E_a$ , Young's modulus in a-direction.
EB	$E_b$ , Young's modulus in b-direction.
EC	$E_c$ , Young's modulus in c-direction.
PRBA	$\nu_{ba}$ , Poisson's ratio ba.
PRCA	$\nu_{ca}$ , Poisson's ratio ca.
PRCB	$\nu_{cb}$ , Poisson's ratio cb.
GAB	$G_{ab}$ , shear modulus ab.

VARIABLE	DESCRIPTION
GBC	$G_{bc}$ , shear modulus bc.
GCA	$G_{ca}$ , shear modulus ca.
DT	Temperature increment for isotropic stress initialization. This option can be used during dynamic relaxation.
TRAMP	Time to ramp up to the final temperature.
ALPHA	Thermal expansion coefficient.
LCIDA	Optional load curve ID defining the nominal stress versus strain along a-axis. Strain is defined as $\lambda_a - 1$ where $\lambda_a$ is the stretch ratio along the a axis.
LCIDB	Optional load curve ID defining the nominal stress versus strain along b-axis. Strain is defined as $\lambda_b - 1$ where $\lambda_b$ is the stretch ratio along the b axis.
EFAIL	Failure strain, $\lambda - 1$ .
DTFAIL	Time step for automatic element erosion
CDAMP	Damping coefficient.
AOPT	Material axes option: EQ. 0.0: locally orthotropic with material axes determined by element nodes as shown in Figure 20.1. Nodes 1, 2, and 4 of an element are identical to the Nodes used for the definition of a coordinate system as by *DEFINE_COORDINATE_NODES. EQ. 1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center, this is the a-direction. EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR. EQ. 3.0: applicable to shell elements only. This option determines locally orthotropic material axes by offsetting the material axes by an angle to be specified from a line in the plane of the shell determined by taking the cross product of the vector $\mathbf{v}$ defined below with the shell normal vector.
XP,YP,ZP	$x_p$ $y_p$ $z_p$ , define coordinates of point $\mathbf{p}$ for AOPT = 1
A1,A2,A3	$a_1$ $a_2$ $a_3$ , define components of vector $\mathbf{a}$ for AOPT = 2.
D1,D2,D3	$d_1$ $d_2$ $d_3$ , define components of vector $\mathbf{d}$ for AOPT = 2.
V1,V2,V3	$v_1$ $v_2$ $v_3$ , define components of vector $\mathbf{v}$ for AOPT = 3
<i>The following input is optional and applies to solid elements only.</i>	
LCIDC	Load curve ID defining the nominal stress versus strain along c-axis. Strain is defined as $\lambda_c - 1$ where $\lambda_c$ is the stretch ratio along the c axis.

<b>VARIABLE</b>	<b>DESCRIPTION</b>
LCIDAB	Load curve ID defining the nominal ab shear stress versus ab-strain in the ab-plane. Strain is defined as the $\sin(\gamma_{ab})$ where $\gamma_{ab}$ is the shear angle.
LCIDBC	Load curve ID defining the nominal ab shear stress versus ab-strain in the bc-plane. Strain is defined as the $\sin(\gamma_{bc})$ where $\gamma_{bc}$ is the shear angle.
LCIDCA	Load curve ID defining the nominal ab shear stress versus ab-strain in the ca-plane. Strain is defined as the $\sin(\gamma_{ca})$ where $\gamma_{ca}$ is the shear angle.

**\*MAT\_USER\_DEFINED\_MATERIAL\_MODELS**

These are Material Types 41-50. The user can supply his own subroutines. See also Appendix A. The keyword input has to be used for the user interface with data. Isotopic and anisotropic material models with failure can be handled.

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	MT	LMC	NHV	IORTHO	IBULK	IG
Type	I	F	I	I	I	I	I	I

Card 2

Variable	IVECT	IFAIL						
Type	I	I						

**Define the following two cards if and only if IORTHO=1**

Card 3

Variable	AOPT	MAXC	XP	YP	ZP	A1	A2	A3
Type	F	F	F	F	F	F	F	F

Card 4

Variable	V1	V2	V3	D1	D2	D3		
Type	F	F	F	F	F	F		

Define LMC material parameters using 8 parameters per card.

Card            1            2            3            4            5            6            7            8

Variable	P1	P2	P3	P4	P5	P6	P7	P8
Type	F	F	F	F	F	F	F	F

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
MT	User material type (41-50 inclusive). A number between 41 and 50 has to be chosen.
LMC	Length of material constant array which is equal to the number of material constants to be input.
NHV	Number of history variables to be stored, see Appendix A.
IORTHO	Set to 1 if the material is orthotropic.
IBULK	Address of bulk modulus in material constants array, see Appendix A.
IG	Address of shear modulus in material constants array, see Appendix A.
IVECT	Vectorization flag (on=1). A vectorized user subroutine must be supplied.
IFAIL	Failure flag (on=1). Allows failure of the elements due to a material failure criterion.
AOPT	Material axes option: EQ. 0.0: locally orthotropic with material axes determined by element nodes as shown in Figure 20.1. Nodes 1, 2, and 4 of an element are identical to the Nodes used for the definition of a coordinate system as by *DEFINE_COORDINATE_NODES, EQ. 1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center, this is the a-direction, EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR, EQ. 3.0: applicable to shell elements only. This option determines locally orthotropic material axes by offsetting the material axes by an angle to be specified from a line in the plane of the shell determined by taking the cross product of the vector $\mathbf{v}$ defined below with the shell normal vector.



---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MAXC	Material axes change flag for brick elements for quick changes: EQ.1.0: default, EQ.2.0: switch material axes a and b, EQ.3.0: switch material axes a and c.
XP YP ZP	Coordinates of point p for AOPT = 1.
A1 A2 A3	Components of vector a for AOPT = 2.
V1 V2 V3	Components of vector v for AOPT = 3.
D1 D2 D3	Components of vector d for AOPT = 2.
P1	First material parameter.
P2	Second material parameter.
P3	Third material parameter.
P4	Fourth material parameter.
.	.
.	.
.	.
PLCM	LCMth material parameter.

**\*MAT\_BAMMAN**

This is Material Type 51. It allows the modeling of temperature and rate dependent plasticity with a fairly complex model that has many input parameters [Bamman, 1989].

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	E	PR	T	HC		
Type	I	F	F	F	F	F		

Card 2

Variable	C1	C2	C3	C4	C5	C6	C7	C8
Type	F	F	F	F	F	F	F	F

Card 3

Variable	C9	C10	C11	C12	C13	C14	C15	C16
Type	F	F	F	F	F	F	F	F

Card 4

Variable	C17	C18	A1	A2	A3	A4	A5	A6
Type	F	F	F	F	F	F	F	F

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
E	Young's modulus (psi)
PR	Poisson's ratio
T	Initial temperature (°R)
HC	Heat generation coefficient (°R/psi)
C1	Psi
C2	°R
C3	Psi
C4	°R
C5	1/s
C6	°R
C7	1/psi
C8	°R
C9	Psi
C10	°R
C11	1/psi-s
C12	°R
C13	1/psi
C14	°R
C15	psi
C16	°R
C17	1/psi-s
C18	°R

---

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
A1	$\alpha_1$ , initial value of internal state variable 1
A2	$\alpha_2$ , initial value of internal state variable 2
A3	$\alpha_4$ , initial value of internal state variable 3
A4	$\alpha_5$ , initial value of internal state variable 4
A5	$\alpha_6$ , initial value of internal state variable 5
A6	$\kappa$ , initial value of internal state variable 6

	sec-psi-°R	sec-MPa-°R	sec-MPA-°K
C1		*1/145	*1/145
C2		—	*5/9
C3		*1/145	*1/145
C4		—	*5/9
C5		—	—
C6		—	*5/9
C7		*145	*145
C8		—	*5/9
C9		*1/145	*1/145
C10		—	*5/9
C11		*145	*145
C12		—	*5/9
C13		*145	*145
C14		—	*5/9
C15		*1/145	*1/145
C16		—	*5/9
C17		*145	*145
C18		—	*5/9
C0=HC		*145	*145*5/9
E		*1/145	*1/145
$\nu$		—	—
T		—	*5/9

**Remarks:**

The kinematics associated with the model are discussed in references [Hill 1948, Bammann and Aifantis 1987, Bammann 1989]. The description below is taken nearly verbatim from Bammann [1989].

With the assumption of linear elasticity we can write,

$$\overset{\circ}{\sigma} = \lambda \operatorname{tr}(D^e)1 + 2\mu D^e$$

where the Cauchy stress  $\sigma$  is convected with the elastic spin  $W^e$  as,

$$\overset{\circ}{\sigma} = \dot{\sigma} - W^e \sigma + \sigma W^e$$

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  $W^e$ ,

$$D^e = D - D^p - D^{th}, \quad W^e = W - W^p.$$

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 temperature,  $\kappa$  is the scalar hardening variable, and  $\xi'$  is the difference between the deviatoric Cauchy stress  $\sigma'$  and the tensor variable  $\alpha'$ ,

$$\xi' = \sigma' - \alpha'$$

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,

$$D^{th} = \dot{A} T 1.$$

The evolution of the internal variables  $\alpha$  and  $\kappa$  are prescribed in a hardening minus recovery format as,

$$\overset{\circ}{\alpha} = h(T) D^p - [r_d(T) |D^p| + r_s(T)] |\alpha| \alpha,$$

$$\dot{\kappa} = H(T) D^p - [R_d(T) |D^p| - R_s(T)] \kappa^2$$

where h and H are the hardening moduli,  $r_s(T)$  and  $R_s(T)$  are scalar functions describing the diffusion controlled ‘static’ or ‘thermal’ recovery, and  $r_d(T)$  and  $R_d(T)$  are the functions describing dynamic recovery.

If we assume that  $W^p = 0$ , we recover the Jaumann stress rate which results in the prediction of an oscillatory shear stress response in simple shear when coupled with a Prager kinematic hardening assumption [Johnson and Bammann 1984]. Alternatively we can choose,

$$W^p = R^T \dot{U} U^{-1} R,$$

which recovers the Green-Naghdi rate of Cauchy stress and has been shown to be equivalent to Mandel’s isoclinic state [Bammann and Aifantis 1987]. The model employing this rate allows a reasonable prediction of directional softening for some materials, but in general under-predicts the softening and does not accurately predict the axial stresses which occur in the torsion of the thin walled tube.

The final equation necessary to complete our description of high strain rate deformation is one which allows us to compute the temperature change during the deformation. In the absence of a coupled thermo-mechanical finite element code we assume adiabatic temperature change and follow the empirical assumption that 90 -95% of the plastic work is dissipated as heat. Hence,

$$\dot{T} = \frac{.9}{\rho C_v} (\sigma \cdot D^p),$$

where  $\rho$  is the density of the material and  $C_v$  the specific heat.

In terms of the input parameters the functions defined above become:

$V(T) = C1 \exp(-C2/T)$	$h(T) = C9 \exp(C10/T)$
$Y(T) = C3 \exp(C4/T)$	$r_s(T) = C11 \exp(-C12/T)$
$f(T) = C5 \exp(-C6/T)$	$RD(T) = C13 \exp(-C14/T)$
$rd(T) = C7 \exp(-C8/T)$	$H(T) = C15 \exp(C16/T)$
	$RS(T) = C17 \exp(-C18/T)$

and the heat generation coefficient is

$$HC = \frac{.9}{\rho C_v}.$$

\*MAT\_BAMMAN\_DAMAGE

This is Material Type 52. This is an extension of model 51 which includes the modeling of damage. See [Bamman, et.al., 1990].

Card Format

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	E	PR	T	HC		
Type	I	F	F	F	F	F		

Card 2

Variable	C1	C2	C3	C4	C5	C6	C7	C8
Type	F	F	F	F	F	F	F	F

Card 3

Variable	C9	C10	C11	C12	C13	C14	C15	C16
Type	F	F	F	F	F	F	F	F

Card 4

Variable	C17	C18	A1	A2	A3	A4	A5	A6
Type	F	F	F	F	F	F	F	F

Card 5

Variable	N	D0	FS					
Type	F	F	F					

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MID	Material identification. A unique number has to be chosen.
RO	Mass density
E	Young's modulus (psi)
PR	Poisson's ratio
T	Initial temperature ( <sup>o</sup> R)
HC	Heat generation coefficient ( <sup>o</sup> R/psi)
C1	Psi
C2	<sup>o</sup> R
C3	Psi
C4	<sup>o</sup> R
C5	1/s
C6	<sup>o</sup> R
C7	1/psi
C8	<sup>o</sup> R
C9	Psi
C10	<sup>o</sup> R
C11	1/psi-s
C12	<sup>o</sup> R



VARIABLE	DESCRIPTION
C13	1/psi
C14	°R
C15	psi
C16	°R
C17	1/psi-s
C18	°R
A1	$\alpha_1$ , initial value of internal state variable 1
A2	$\alpha_2$ , initial value of internal state variable 2
A3	$\alpha_3$ , initial value of internal state variable 3
A4	$\alpha_4$ , initial value of internal state variable 4
A5	$\alpha_5$ , initial value of internal state variable 5
A6	$\alpha_6$ , initial value of internal state variable 6
N	Exponent in damage evolution
D0	Initial damage (porosity)
FS	Failure strain for erosion.

**Remarks:**

The evolution of the damage parameter,  $\phi$ , is defined by [Bammann, et al. 1990]

$$\phi = \beta \left[ \frac{1}{(1-\phi)^N} - (1-\phi) \right]^{D^p}$$

in which

$$\beta = \sinh \left[ \frac{2(2N-1)p}{(2N-1)\bar{\sigma}} \right]$$

where  $p$  is the pressure and  $\bar{\sigma}$  is the effective stress.

**\*MAT\_CLOSED\_CELL\_FOAM**

This is Material Type 53. This allows the modeling of low density, closed cell polyurethane foam. It is for simulating impact limitors in automotive applications. The effect of the confined air pressure is included with the air being treated as an ideal gas. The general behavior is isotropic with uncoupled components of the stress tensor.

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	E	A	B	C	P0	PHI
Type	I	F	F	F	F	F	F	F

Card 2

Variable	GAMA0							
Type	F							

**VARIABLE**

**DESCRIPTION**

MID	Material identification. A unique number has to be chosen.
RO	Mass density
E	Young's modulus
A	a, factor for yield stress definition, see notes below.
B	b, factor for yield stress definition, see notes below.
C	c, factor for yield stress definition, see notes below.
P0	Initial foam pressure, P <sub>0</sub>
PHI	Ratio of foam to polymer density, $\phi$
GAMA0	Initial volumetric strain, $\gamma_0$ . The default is zero.

**Remarks:**

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  $\sigma_{ij}^{sk}$  is the skeletal stress and  $\sigma^{air}$  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  $\gamma$  defines the volumetric strain

$$\gamma = V - 1 + \gamma_0$$

where  $V$  is the relative volume, defined as the ratio of the current volume to the initial volume, and  $\gamma_0$  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,  $\sigma^{skt}$

$$\sigma_{ij}^{skt} = \sigma_{ij}^{sk} + E \dot{\epsilon}_{ij} \Delta t$$

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,  $\sigma_i^{skt}$ , exceeds the yield stress,  $\sigma_y$ , then

$$\sigma_i^{sk} = \min(\sigma_y, |\sigma_i^{skt}|) \frac{\sigma_i^{skt}}{|\sigma_i^{skt}|}$$

The yield stress is defined by

$$\sigma_y = a + b (1 + c\gamma)$$

where a, b, and c are user defined input constants and  $\gamma$  is the volumetric strain as defined above. After scaling the principal stresses they are transformed back into the global system and the final stress state is computed

$$\sigma_{ij} = \sigma_{ij}^{sk} - \delta_{ij} \sigma^{air}$$

\*MAT\_ENHANCED\_COMPOSITE\_DAMAGE

These are Material Types 54-55 which are enhanced versions of the composite model material type 22. Arbitrary orthotropic 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. The parameters in parentheses below apply only to solid elements and are therefore always ignored in this material model. They are included for consistency with material types 22 and 59.

Card Format

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	EA	EB	(EC)	PRBA	(PRCA)	(PRCB)
Type	I	F	F	F	F	F	F	F

Card 2

Variable	GAB	GBC	GCA	(KF)	AOPT			
Type	F	F	F	F	F			

Card 3

Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card 4

Variable	V1	V2	V3	D1	D2	D3	DFAILM	DFAILS
Type	F	F	F	F	F	F	F	F

Card 5

Variable	TFAIL	ALPH	SOFT	FBRT	YCFAC	DFAILT	DFAILC	EFS
Type	F	F	F	F	F	F	F	F

Card 6

Variable	XC	XT	YC	YT	SC	CRIT	BETA	
Type	F	F	F	F	F	F	F	

**VARIABLE****DESCRIPTION**

MID	Material identification. A unique number has to be chosen.
RO	Mass density
EA	$E_a$ , Young's modulus - longitudinal direction
EB	$E_b$ , Young's modulus - transverse direction
(EC)	$E_c$ , Young's modulus - normal direction (not used)
PRBA	$\nu_{ba}$ , Poisson's ratio ba
(PRCA)	$\nu_{ca}$ , Poisson's ratio ca (not used)
(PRCB)	$\nu_{cb}$ , Poisson's ratio cb (not used)
GAB	$G_{ab}$ , shear modulus ab
GBC	$G_{bc}$ , shear modulus bc

VARIABLE	DESCRIPTION
GCA	$G_{ca}$ , shear modulus ca
(KF)	Bulk modulus of failed material (not used)
AOPT	Material axes option, see Figure 20.1: EQ. 0.0: locally orthotropic with material axes determined by element nodes as shown in Figure 20.1. Nodes 1, 2, and 4 of an element are identical to the Nodes used for the definition of a coordinate system as by *DEFINE_COORDINATE_NODES. EQ. 1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center, this is the a-direction. EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR. EQ. 3.0: applicable to shell elements only. This option determines locally orthotropic material axes by offsetting the material axes by an angle to be specified from a line in the plane of the shell determined by taking the cross product of the vector $\mathbf{v}$ defined below with the shell normal vector.
XP YP ZP	Define coordinates of point p for AOPT = 1.
A1 A2 A3	Define components of vector a for AOPT = 2.
V1 V2 V3	Define components of vector v for AOPT = 3.
D1 D2 D3	Define components of vector d for AOPT = 2.
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.
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.
TFAIL	Time step size criteria for element deletion: $\leq 0$ : no element deletion by time step size. The crashfront algorithm only works if $t_{fail}$ is set to a value above zero. $0 < t_{fail} \leq 0.1$ : element is deleted when its time step is smaller than the given value, $>.1$ : element is deleted when the quotient of the actual time step and the original time step drops below the given value.
ALPH	Shear stress parameter for the nonlinear term, see Material 22.
SOFT	Softening reduction factor for material strength in crashfront elements (default = 1.0 ). TFAIL must be greater than zero to activate this option.

<b>VARIABLE</b>	<b>DESCRIPTION</b>
FBRT	Softening for fiber tensile strength: EQ.0.0: tensile strength = $X_t$ GT:0.0: tensile strength = $X_t$ , reduced to $X_t * FBRT$ after failure has occurred in compressive matrix mode.
YCFAC	Reduction factor for compressive fiber strength after matrix failure. The compressive strength in the fiber direction after compressive matrix failure is reduced to: $X_c = YCFAC * Y_c$ (default : $YCFAC = 2.0$ )
DFAILT	Maximum strain for fiber tension. (Maximum 1 = 100% strain). The layer in the element is completely removed after the maximum tensile strain in the fiber direction is reached.
DFAILC	Maximum strain for fiber compression (Maximum -1 = 100% compression). The layer in the element is completely removed after the maximum tensile strain in the fiber direction is reached. The input value must have a negative sign.
EFS	Effective failure strain.
XC	Longitudinal compressive strength
XT	Longitudinal tensile strength, see below.
YC	Transverse compressive strength, b-axis, see below.
YT	Transverse tensile strength, b-axis, see below.
SC	Shear strength, ab plane, see below.
CRIT	Failure criterion (material number): EQ.54.0: Chang matrix failure criterion (as Material 22) (default), EQ.55.0: Tsai-Wu criterion for matrix failure.
BETA	Weighting factor for shear term in tensile fiber mode ( $0.0 \leq BETA \leq 1.0$ )

**Remarks:**

The Chang/Chang criteria is given as follows:

for the tensile fiber mode,

$$\sigma_{aa} > 0 \quad \text{then} \quad e_f^2 = \left( \frac{\sigma_{aa}}{X_t} \right)^2 + \beta \left( \frac{\sigma_{ab}}{S_c} \right) - 1 \begin{cases} \geq 0 & \text{failed} \\ < 0 & \text{elastic} \end{cases}$$

$$E_a = E_b = G_{ab} = \nu_{ba} = \nu_{ab} = 0,$$

for the compressive fiber mode,



$$\sigma_{aa} < 0 \quad \text{then} \quad e_c^2 = \left( \frac{\sigma_{aa}}{X_c} \right)^2 - 1 \begin{cases} \geq 0 & \text{failed} \\ < 0 & \text{elastic} \end{cases},$$

$$E_a = \nu_{ba} = \nu_{ab} = 0.$$

for the tensile matrix mode,

$$\sigma_{bb} > 0 \quad \text{then} \quad e_m^2 = \left( \frac{\sigma_{bb}}{Y_t} \right)^2 + \left( \frac{\sigma_{ab}}{S_c} \right)^2 - 1 \begin{cases} \geq 0 & \text{failed} \\ < 0 & \text{elastic} \end{cases}$$

$$E_b = \nu_{ba} = 0. \quad \rightarrow G_{ab} = 0,$$

and for the compressive matrix mode,

$$\sigma_{bb} < 0 \quad \text{then} \quad e_d^2 = \left( \frac{\sigma_{bb}}{2S_c} \right)^2 + \left[ \left( \frac{Y_c}{2S_c} \right)^2 - 1 \right] \frac{\sigma_{bb}}{Y_c} + \left( \frac{\sigma_{ab}}{S_c} \right)^2 - 1 \begin{cases} \geq 0 & \text{failed} \\ < 0 & \text{elastic} \end{cases}$$

$$\nu_b = \nu_{ba} = \nu_{ab} = 0. \quad \rightarrow G_{ab} = 0$$

$$X_c = 2Y_c \quad \text{for 50\% fiber volume}$$

In the Tsai-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:

$$e_{md}^2 = \frac{\sigma_{bb}^2}{Y_c Y_t} + \left( \frac{\sigma_{ab}}{S_c} \right)^2 + \frac{(Y_c - Y_t) \sigma_{bb}}{Y_c Y_t} - 1 \begin{cases} \geq 0 & \text{failed} \\ < 0 & \text{elastic} \end{cases}$$

For  $\beta = 1$  we get the original criterion of Hashin [1980] in the tensile fiber mode. For  $\beta = 0$  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 effective strain is greater than EFS.
4. If TFAIL is greater than zero, failure occurs according to the element timestep as described in the definition of TFAIL above.

When failure has occurred in all the composite layers (through-thickness integration points), the element is deleted. Elements which share nodes with the deleted element become "crashfront" elements and can have their strengths reduced by using the SOFT parameter with TFAIL greater than zero.

Information about the status in each layer (integration point) and element can be plotted using additional integration point variables. The number of additional integration point variables for shells written to the LS-DYNA database is input by the \*DATABASE\_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. $ef(i)$	<i>tensile fiber mode</i>	1 - elastic 0 - failed	81
2. $ec(i)$	<i>compressive fiber mode</i>		82
3. $em(i)$	<i>tensile matrix mode</i>		83
4. $ed(i)$	<i>compressive matrix mode</i>		84
5. $efail$	$max[ef(ip)]$		85
6. $dam$	<i>damage parameter</i>	-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*

**\*MAT\_LOW\_DENSITY\_FOAM**

This is Material Type 57. It is mainly for modeling highly compressible low density foams. Its main applications are for seat cushions and padding on the Side Impact Dummies (SID). Optionally, a tension cut-off failure can be defined. Also, see the notes below.

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	E	LCID	TC	HU	BETA	DAMP
Type	I	F	F	F	F	F	F	F
Default	---	---	---	---	1.E+20	1.		
Remarks	---	---	---	---	---	3	1	---

Card 2

Variable	SHAPE	FAIL	BVFLAG	ED	BETA1	KCON	REF	
Type	F	F	F	F	F	F	F	
Default	1.0	0.0	0.0	0.0	0.0	0.0	0.0	
Remarks	3	---	2					

**VARIABLE****DESCRIPTION**

MID	Material identification. A unique number has to be chosen.
RO	Mass density
E	Young's modulus

VARIABLE	DESCRIPTION
LCID	Load curve ID, see *DEFINE_CURVE, for nominal stress versus strain.
TC	Tension cut-off stress
HU	Hysteretic unloading factor between 0 and 1 (default=1, i.e., no energy dissipation), see also Figure 20.14.
BETA	$\beta$ , decay constant to model creep in unloading
DAMP	Viscous coefficient (.05< recommended value <.50) to model damping effects.
SHAPE	Shape factor for unloading. Active for nonzero values of the hysteretic unloading factor. Values less than one reduces the energy dissipation and greater than one increases dissipation, see also Figure 20.14.
FAIL	Failure option after cutoff stress is reached: EQ.0.0: tensile stress remains at cut-off value, EQ.1.0: tensile stress is reset to zero.
BVFLAG	Bulk viscosity activation flag, see remark below: EQ.0.0: no bulk viscosity (recommended), EQ.1.0: bulk viscosity active.
ED	Optional Young's relaxation modulus, $E_d$ , for rate effects. See comments below.
BETA1	Optional decay constant, $\beta_1$ .
KCON	Stiffness coefficient for contact interface stiffness. Maximum slope in stress vs. strain curve is used. When the maximum slope is taken for the contact, the time step size for this material is reduced for stability. In some cases $\Delta t$ may be significantly smaller, and defining a reasonable stiffness is recommended.
REF	Use reference geometry to initialize the stress tensor. The reference geometry is defined by the keyword:*INITIAL_FOAM_REFERENCE_GEOMETRY. This option is currently restricted to 8-noded solid elements with one point integration. EQ.0.0: off, EQ.1.0: on.

**Remarks:**

The compressive behavior is illustrated in Figure 20.14 where hysteresis on unloading is shown. This behavior under uniaxial loading is assumed not to significantly couple in the transverse directions. In tension the material behaves in a linear fashion until tearing occurs. Although our implementation may be somewhat unusual, it was motivated by Storakers [1986].

The model uses tabulated input data for the loading curve where the nominal stresses are defined as a function of the elongations,  $\epsilon_i$ , which are defined in terms of the principal stretches,  $\lambda_i$ , as:

$$\varepsilon_i = \lambda_i - 1$$

The stretch ratios are found by solving for the eigenvalues of the left stretch tensor,  $V_{ij}$ , which is obtained via a polar decomposition of the deformation gradient matrix,  $F_{ij}$ . Recall that,

$$F_{ij} = R_{ik}U_{kj} = V_{ik}R_{kj}$$

The update of  $V_{ij}$  follows the numerically stable approach of [Taylor and Flanagan 1989]. After solving for the principal stretches, we compute the elongations and, if the elongations are compressive, the corresponding values of the nominal stresses,  $\tau_i$ , are interpolated. If the elongations are tensile, the nominal stresses are given by

$$\tau_i = E\varepsilon_i$$

and the Cauchy stresses in the principal system become

$$\sigma_i = \frac{\tau_i}{\lambda_j \lambda_k}$$

The stresses can now be transformed back into the global system for the nodal force calculations.

#### **Additional Remarks:**

1. When hysteretic unloading is used the reloading will follow the unloading curve if the decay constant,  $\beta$ , is set to zero. If  $\beta$  is nonzero the decay to the original loading curve is governed by the expression:

$$1.-e^{-\beta t}$$

2. The bulk viscosity, which generates a rate dependent pressure, may cause an unexpected volumetric response and, consequently, it is optional with this model.
3. The hysteretic unloading factor results in the unloading curve to lie beneath the loading curve as shown in Figure 20.14. This unloading provide energy dissipation which is reasonable in certain kinds of foam.

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,  $\sigma_{ij}^r$ , augments the stresses determined from the foam,  $\sigma_{ij}^f$ ; consequently, the final stress,  $\sigma_{ij}$ , is taken as the summation of the two contributions:

$$\sigma_{ij} = \sigma_{ij}^f + \sigma_{ij}^r.$$

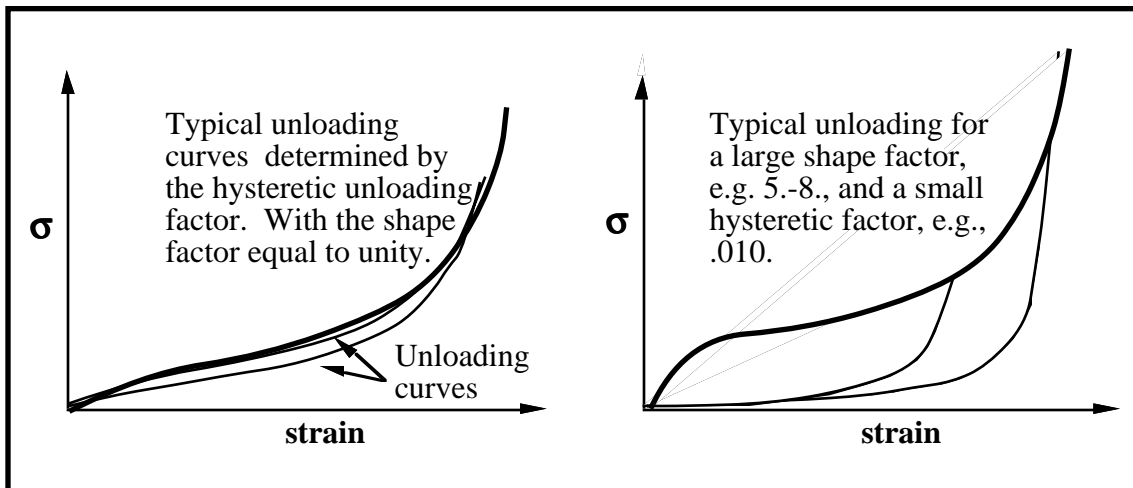
Since we wish to include only simple rate effects, the relaxation function is represented by one term from the Prony series:

$$g(t) = \alpha_0 + \sum_{m=1}^N \alpha_m e^{-\beta t}$$

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,  $\beta_1$ . 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 20.14.** Behavior of the low density urethane foam model.

# \*MAT

## \*MAT\_LAMINATED\_COMPOSITE\_FABRIC

---

### \*MAT\_LAMINATED\_COMPOSITE\_FABRIC

This is Material Type 58. Depending on the type of failure surface, this model may be used to model composite materials with unidirectional layers, complete laminates, and woven fabrics. This model is implemented for shell elements only.

#### Card Format

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	EA	EB	(EC)	PRBA	TAU1	GAMMA1
Type	I	F	F	F	F	F	F	F

Card 2

Variable	GAB	GBC	GCA	SLIMIT1	SLIMC1	SLIMIT2	SLIMC2	SLIMS
Type	F	F	F	F	F	F	F	F

Card 3

Variable	AOPT	TSIZE	ERODS	SOFT	FS			
Type	F	F	F	F	F			

Card 4

Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		



Card 5

Variable	V1	V2	V3	D1	D2	D3		
Type	F	F	F	F	F	F		

Card 6

Variable	E11C	E11T	E22C	E22T	GMS			
Type	F	F	F	F	F			

Card 7

Variable	XC	XT	YC	YT	SC			
Type	F	F	F	F	F			

**VARIABLE**

**DESCRIPTION**

MID	Material identification. A unique number has to be chosen.
RO	Mass density
EA	$E_a$ , Young's modulus - longitudinal direction
EB	$E_b$ , Young's modulus - transverse direction
(EC)	$E_c$ , Young's modulus - normal direction (not used)
PRBA	$\nu_{ba}$ , Poisson's ratio ba
TAU1	$\tau_1$ , 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.
GAMMA1	$\gamma_1$ , strain limit of the first slightly nonlinear part of the of the shear stress versus shear strain curve.

<b>VARIABLE</b>	<b>DESCRIPTION</b>
GAB	$G_{ab}$ , shear modulus ab
GBC	$G_{bc}$ , shear modulus bc
GCA	$G_{ca}$ , shear modulus ca
SLIMIT1	Factor to determine the minimum stress limit after stress maximum (fiber tension).
SLIMC1	Factor to determine the minimum stress limit after stress maximum (fiber compression).
SLIMIT2	Factor to determine the minimum stress limit after stress maximum (matrix tension).
SLIMC2	Factor to determine the minimum stress limit after stress maximum (matrix compression).
SLIMS	Factor to determin the minimum stress limit after stress maximum. (shear).
AOPT	Material axes option, see Figure 20.1: EQ. 0.0: locally orthotropic with material axes determined by element nodes as shown in Figure 20.1. Nodes 1, 2, and 4 of an element are identical to the Nodes used for the definition of a coordinate system as by *DEFINE_COORDINATE_NODES. EQ. 1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center, this is the a-direction. EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR. EQ. 3.0: applicable to shell elements only. This option determines locally orthotropic material axes by offsetting the material axes by an angle to be specified from a line in the plane of the shell determined by taking the cross product of the vector $\mathbf{v}$ defined below with the shell normal vector.
TSIZE	Time step for automatic element deletion.
ERODS	Maximum effective strain for element layer failure. A value of unity would equal 100% strain.
SOFT	Softening reduction factor for strength in the crashfront.
FS	Failure surface type: EQ.1.0:smooth failure surface with a quadratic criterion for both the fiber (a) and transverse (b) directions. This option can be used with complete laminates and fabrics EQ.0.0:smooth failure surface in the transverse (b) direction with a limiting value in the fiber (a) direction. This model is appropriate for unidirectional (UD) layered composites only.

<u>VARIABLE</u>	<u>DESCRIPTION</u>
	EQ.-1.:faceted failure surface. When the strength values are reached then damage evolves in tension and compression for both the fiber and transverse direction. Shear behavior is also considered. This option can be used with complete laminates and fabrics
XP YP ZP	Define coordinates of point p for AOPT = 1.
A1 A2 A3	Define components of vector a for AOPT = 2.
V1 V2 V3	Define components of vector v for AOPT = 3.
D1 D2 D3	Define components of vector d for AOPT = 2.
E11C	Strain at longitudinal compressive strength, a-axis.
E11T	Strain at longitudinal tensile strength, a-axis.
E22C	Strain at transverse compressive strength, b-axis.
E22T	Strain at transverse tensile strength, b-axis.
GMS	Strain at shear strength, ab plane.
XC	Longitudinal compressive strength
XT	Longitudinal tensile strength, see below.
YC	Transverse compressive strength, b-axis, see below.
YT	Transverse tensile strength, b-axis, see below.
SC	Shear strength, ab plane, see below.

**Remarks:**

Parameters to control failure of an element layer are: ERODS, the maximum effective strain, i.e., maximum 1 = 100 % straining. The layer in the element is completely removed after the maximum effective strain (compression/tension including shear) is reached.

The stress limits are factors used to limit the stress in the softening part to a given value,

$$\sigma_{\min} = SLIM_{xx} \cdot strength,$$

thus, the damage value is slightly modified such that elastoplastic like behavior is achieved with the threshold stress. As a factor for  $SLIM_{xx}$  a number between 0.0 and 1.0 is possible. With a factor of 1.0, the stress remains at a maximum value identical to the strength, which is similar to ideal elastoplastic behavior. For tensile failure a small value for  $SLIM_{Tx}$  is often reasonable; however, for compression  $SLIM_{Cx} = 1.0$  is preferred. This is also valid for the corresponding shear value. If  $SLIM_{xx}$  is smaller than 1.0 then localization can be observed depending on the total behavior of the lay-up. If the user is intentionally using  $SLIM_{xx} < 1.0$ , it is generally recommended to avoid a drop to zero and set the value to something in between 0.05 and 0.10. Then elastoplastic behaviour is achieved in the limit which often leads to less numerical problems. Defaults for  $SLIM_{XX} = 1.0E-8$ .

The crashfront-algorithm is started if and only if a value for TSIZE (time step size, with element elimination after the actual time step becomes smaller than TSIZE) is input .

The damage parameters can be written to the postprocessing database for each integration point as the first three additional element variables and can be visualized.

Material models with FS=1 or FS=-1 are favorable for complete laminates and fabrics, as all directions are treated in a similar fashion.

For material model FS=1 an interaction between normal stresses and the shear stresses is assumed for the evolution of damage in the a and b-directions. For the shear damage is always the maximum value of the damage from the criterion in a or b-direction is taken.

For material model FS=-1 it is assumed that the damage evolution is independent of any of the other stresses. A coupling is only present via the elastic material parameters and the complete structure.

In tensile and compression directions and in a as well as in b- direction different failure surfaces can be assumed. The damage values, however, increase only also when the loading direction changes.

#### **Special control of shear behavior of fabrics**

For fabric materials a nonlinear stress strain curve for the shear part for failure surface FS=-1 can be assumed as given below. This is not possible for other values of FS.

The curve, shown in Figure 20.15 is defined by three points:

- a) the origin (0,0) is assumed,
- b) the limit of the first slightly nonlinear part (must be input), stress (TAU1) and strain (GAMMA1), see below.
- c) the shear strength at failure and shear strain at failure.

In addition a stress limiter can be used to keep the stress constant via the *SLIMS* parameter. This value must be less or equal 1.0 but positive, and leads to an elastoplastic behavior for the shear part. The default is 1.0E-08, assuming almost brittle failure once the strength limit SC is reached.

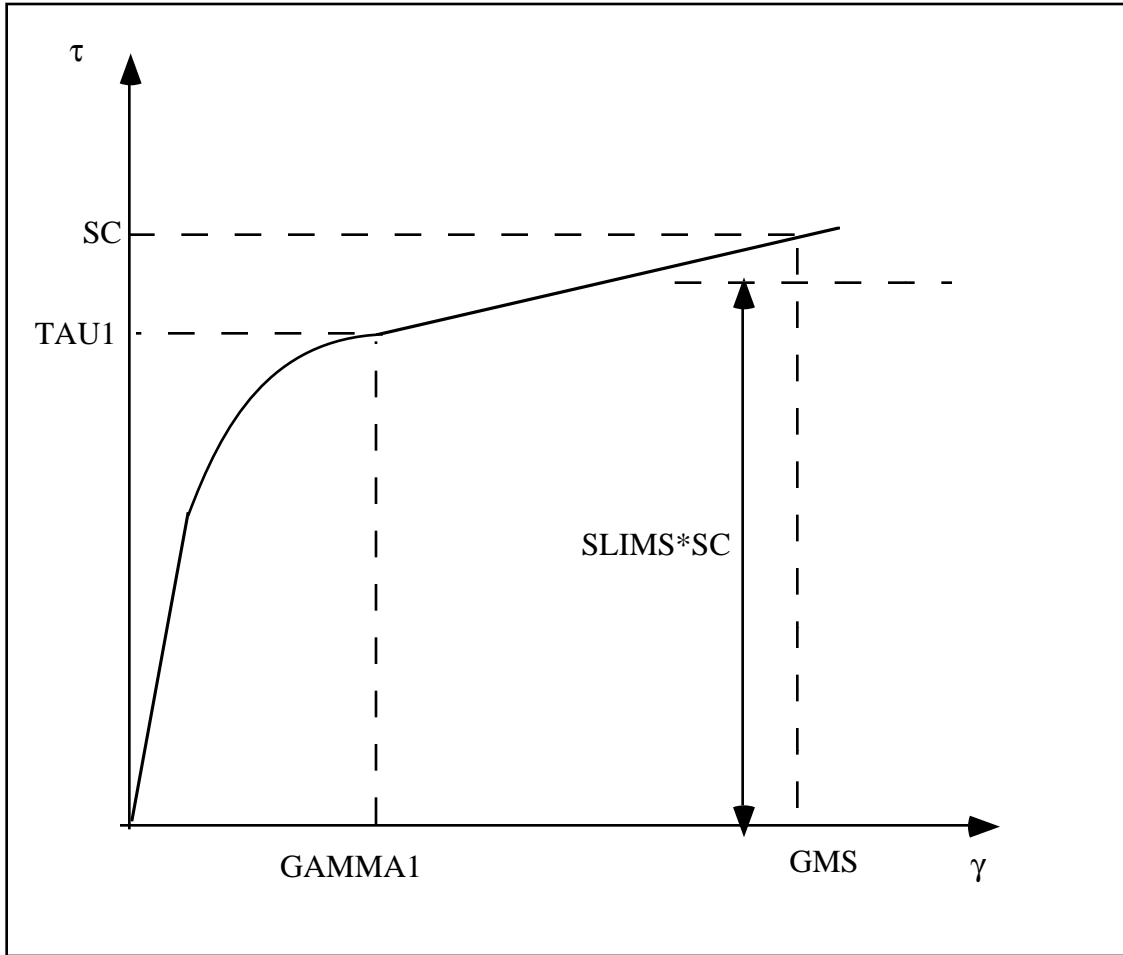


Figure 20.15. Stress-strain diagram for shear.

# \*MAT

## \*MAT\_COMPOSITE\_FAILURE\_OPTION\_MODEL

---

### \*MAT\_COMPOSITE\_FAILURE\_OPTION\_MODEL

This is Material Type 59.

Where *OPTION* is either **SHELL** or **SOLID** depending on the element type the material is to be used with, see \*PART.

**For both options define cards 1 to 4 below**

#### Card Format

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	EA	EB	EC	PRBA	PRCA	PRCB
Type	I	F	F	F	F	F	F	F

Card 2

Variable	GAB	GBC	GCA	KF	AOPT	MAFLAG		
Type	F	F	F	F	F	F		

Card 3

Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card 4

Variable	V1	V2	V3	D1	D2	D3		
Type	F	F	F	F	F	F		

**Cards 5 and 6 for SHELL option**

Card 5

Variable	TSIZE	ALP	SOFT	FBRT	SR	SF		
Type	F	F	F	F	F	F		

Card 6

Variable	XC	XT	YC	YT	SC			
Type	F	F	F	F	F			

**Cards 5 and 6 for SOLID option**

Card 5

Variable	SBA	SCA	SCB	XXC	YYC	ZZC		
Type	F	F	F	F	F	F		

Card 6

Variable	XXT	YYT	ZZT					
Type	F	F	F					

<b>VARIABLE</b>	<b>DESCRIPTION</b>
MID	Material identification
RO	Density
EA	$E_a$ , Young's modulus - longitudinal direction
EB	$E_b$ , Young's modulus - transverse direction
EC	$E_c$ , Young's modulus - normal direction
PRBA	$\nu_{ba}$ Poisson's ratio ba
PRCA	$\nu_{ca}$ Poisson's ratio ca
PRCB	$\nu_{cb}$ Poisson's ratio cb
GAB	$G_{ab}$ Shear Stress
GBC	$G_{bc}$ Shear Stress
GCA	$G_{ca}$ Shear Stress
KF	Bulk modulus of failed material
AOPT	Material axes option: EQ.0.0: locally orthotropic with material axes determined by element nodes $n_1$ , $n_2$ , and $n_4$ as shown in Figure 20.1. EQ.1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center. EQ.2.0: globally orthotropic with material axes determined by vectors defined below. EQ.3.0: applicable to shell elements only. This option determines locally orthotropic material axes by offsetting the material axes by an angle to be specified from a line in the plane of the shell determined by taking the cross product of the vector defined below with the shell normal vector.
MAFLAG	Material axes change flag for brick elements.
XP YP ZP	Define coordinates of point p for AOPT = 1.
A1 A2 A3	Define components of vector a for AOPT = 2.
V1 V2 V3	Define components of vector v for AOPT = 3 (shells only).
D1 D2 D3	Define components of vector d for AOPT = 2: EQ.1.0: default, EQ.2.0: switch material axes a and b, EQ.3.0: switch material axes a and c.



---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
TSIZE	Time step for automatic element deletion
ALP	Nonlinear shear stress parameter
SOFT	Softening reduction factor for strength in crush
FBRT	Softening of fiber tensile strength
SR	s <sub>r</sub> , reduction factor(default=0.447)
SF	s <sub>f</sub> , softening factor(default=0.0)
XC	Longitudinal compressive strength, a-axis
XT	Longitudinal tensile strength, a-axis
YC	Transverse compressive strength, b-axis
YT	Transverse tensile strength, b-axis
SC	Shear strength, ab plane: GT:0.0: faceted failure surface theory, LT:0.0: ellipsoidal failure surface theory.
SBA	In plane shear strength..
SCA	Transverse shear strength.
SCB	Transverse shear strength.
XXC	Longitudinal compressive strength x-axis.
YYC	Transverse compressive strength b-axis.
ZZC	Normal compressive strength c-axis.
XXT	Longitudinal tensile strength a-axis.
YYT	Transverse tensile strength b-axis.
ZZT	Normal tensile strength c-axis.

**\*MAT\_ELASTIC\_WITH\_VISCOSITY**

This is Material Type 60 which was developed to simulate forming of glass products (e.g., car windshields) at high temperatures. Deformation is by viscous flow but elastic deformations can also be large. The material model, in which the viscosity may vary with temperature, is suitable for treating a wide range of viscous flow problems and is implemented for brick and shell elements.

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	V0	A	B	C	LCID	
Type	I	F	F	F	F	F	F	

Card 2

Variable	PR1	PR2	PR3	PR4	PR5	PR6	PR7	PR8
Type	F	F	F	F	F	F	F	F

Card 3

Variable	T1	T2	T3	T4	T5	T6	T7	T8
Type	F	F	F	F	F	F	F	F

Card 4

Variable	V1	V2	V3	V4	V5	V6	V7	V8
Type	F	F	F	F	F	F	F	F

Card 5

Variable	E1	E2	E3	E4	E5	E6	E7	E8
Type	F	F	F	F	F	F	F	F

Card 6

Variable	ALPHA1	ALPHA2	ALPHA3	ALPHA4	ALPHA5	ALPHA6	ALPHA7	ALPHA8
Type	F	F	F	F	F	F	F	F

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MID	Material identification. A unique number has to be chosen.
RO	Mass density
V0	Temperature independent viscosity coefficient, $V_0$ . If defined, the temperature dependent viscosity defined below is skipped, see type (i) and (ii) definitions for viscosity below.
A	Viscosity coefficient, see type (i) and (ii) definitions for viscosity below.
B	Viscosity coefficient, see type (i) and (ii) definitions for viscosity below.
C	Viscosity coefficient, see type (i) and (ii) definitions for viscosity below.
LCID	Load curve , see *DEFINE_CURVE, defining factor on viscosity versus time. (Optional).
T1, T2,...TN	Temperatures, define up to 8 values
PR1, PR2,...PRN	Poisson's ratios for the temperatures $T_i$
V1, V2,...VN	Corresponding viscosity coefficients (define only one if not varying with temperature).
E1, E2,...EN	Corresponding Young's moduli coefficients (define only one if not varying with temperature).
ALPHA....	Corresponding thermal expansion coefficients

**Remarks:**

This material model was developed to simulate forming of glass products (e.g. car windshields) at high temperatures. Deformation is by viscous flow but elastic deformations can also be large. The material model, in which the viscosity may vary with temperature, is suitable for treating a wide range of viscous flow problems and is implemented for brick and shell elements.

Volumetric behavior is treated as linear elastic. The deviatoric strain rate is considered to be the sum of elastic and viscous strain rates:

$$\dot{\underline{\underline{\varepsilon}}}'_{\sim total} = \dot{\underline{\underline{\varepsilon}}}'_{\sim elastic} + \dot{\underline{\underline{\varepsilon}}}'_{\sim viscous} = \frac{\underline{\underline{\sigma}}'}{2G} + \frac{\underline{\underline{\sigma}}'}{2\nu}$$

where G is the elastic shear modulus,  $\nu$  is the viscosity coefficient, and  $\sim$  indicates a tensor. The stress increment over one timestep  $dt$  is

$$d\underline{\underline{\sigma}}' = 2G \dot{\underline{\underline{\varepsilon}}}'_{\sim total} dt - \frac{G}{\nu} dt \underline{\underline{\sigma}}'$$

The stress before the update is used for  $\underline{\underline{\sigma}}'$ . For shell elements the through-thickness strain rate is calculated as follows.

$$d\sigma_{33} = 0 = K \left( \dot{\varepsilon}_{11} + \dot{\varepsilon}_{22} + \dot{\varepsilon}_{33} \right) dt + 2G \dot{\varepsilon}'_{33} dt - \frac{G}{\nu} dt \sigma'_{33}$$

where the subscript  $ij = 33$  denotes the through-thickness direction and K is the elastic bulk modulus. This leads to:

$$\dot{\varepsilon}_{33} = -a \left( \dot{\varepsilon}_{11} + \dot{\varepsilon}_{22} \right) + bp$$

$$a = \frac{\left( K - \frac{2}{3}G \right)}{\left( K + \frac{4}{3}G \right)}$$

$$b = \frac{Gdt}{\nu \left( K + \frac{4}{3}G \right)}$$

in which p is the pressure defined as the negative of the hydrostatic stress.

The variation of viscosity with temperature can be defined in any one of the 3 ways.

- ( i ) Constant,  $\nu = \nu_0$  Do not define constants, A, B, and C or the piecewise curve.(leave card 4 blank)
- ( ii )  $\nu = \nu_0 \times 10^{**} (A/(T-B) + C)$
- (iii) Piecewise curve: define the variation of viscosity with temperature.

**Note:** Viscosity is inactive during dynamic relaxation.

\*MAT\_KELVIN-MAXWELL\_VISCOELASTIC

This is Material Type 61. It is a classical Kelvin-Maxwell model for modelling viscoelastic bodies, e.g., foams. Only valid for solid elements. See also notes below.

Card Format

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	BULK	G0	GI	DC	FO	SO
Type	I	F	F	F	F	F	F	F
Default	none	none	none	none	none	0.0	0.0	0.0

VARIABLE

DESCRIPTION

MID	Material identification. A unique number has to be chosen.
RO	Mass density
BULK	Bulk modulus (elastic)
G0	Short-time shear modulus, $G_0$
GI	Long-time (infinite) shear modulus, $G_\infty$
DC	Maxwell decay constant, $\beta$ [FO=0.0] or Kelvin relaxation constant, $\tau$ [FO=1.0]
FO	Formulation option: EQ.0.0: Maxwell, EQ.1.0: Kelvin.
SO	Strain (logarithmic) output option to be plotted as component 7 in LS-TAURUS (D3PLOT file) 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 values that occurs during the calculation, EQ.2.0: maximum effective strain that occurs during the calculation.

**Remarks:**

The shear relaxation behavior is described for the Maxwell model by:

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

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  $D_{ij}$ . For the Kelvin model the stress evolution equation is defined as:

$$\dot{s}_{ij} + \frac{1}{\tau} s_{ij} = (1 + \delta_{ij}) G_0 \dot{e}_{ij} + (1 + \delta_{ij}) \frac{G_{\infty}}{\tau} \dot{e}_{ij}$$

The strain data as written to the LS-DYNA database may be used to predict damage, see [Bandak 1991].

\*MAT\_VISCOUS\_FOAM

This is Material Type 62. It was written to represent the Confor Foam on the ribs of EuroSID side impact dummy. It is only valid for solid elements, mainly under compressive loading.

Card Format

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	E1	N1	V2	E2	N2	PR
Type	I	F	F	F	F	F	F	F

VARIABLE

DESCRIPTION

MID	Material identification. A unique number has to be chosen.
RO	Mass density
E1	Initial Young's modulus ( $E_1$ )
N1	Exponent in power law for Young's modulus ( $n_1$ )
V2	Viscous coefficient ( $V_2$ )
E2	Elastic modulus for viscosity ( $E_2$ ), see notes below.
N2	Exponent in power law for viscosity ( $n_2$ )
PR	Poisson's ratio, $\nu$

**Remarks:**

The model consists of a nonlinear elastic stiffness in parallel with a viscous damper. The elastic stiffness is intended to limit total crush while the viscosity absorbs energy. The stiffness  $E_2$  exists to prevent timestep problems. It is used for time step calculations as long as  $E_1^t$  is smaller than  $E_2$ . It has to be carefully chosen to take into account the stiffening effects of the viscosity. Both  $E_1$  and  $V_2$  are nonlinear with crush as follows:

$$E_1^t = E_1(V^{-n_1})$$

$$V_2^t = V_2(abs(1 - V))^{n_2}$$

where viscosity generates a shear stress given by

$$\tau = V_2 \dot{\gamma}$$

$\dot{\gamma}$  is the engineering shear strain rate, and V is the relative volume defined by the ratio of the current to initial volume. Typical values are (units of N, mm, s)

$$E_1 = 0.0036$$

$$n_1 = 4.0$$

$$V_2 = 0.0015$$

$$E_2 = 100.0$$

$$n_2 = 0.2$$

$$v = 0.05$$



\*MAT\_CRUSHABLE\_FOAM

This is Material Type 63 which is dedicated to modeling crushable foam with optional damping and tension cutoff. Unloading is fully elastic. Tension is treated as completely elastic-plastic.

Card Format

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	E	PR	LCID	TSC	DAMP	
Type	I	F	F	F	F	F	F	

VARIABLE

DESCRIPTION

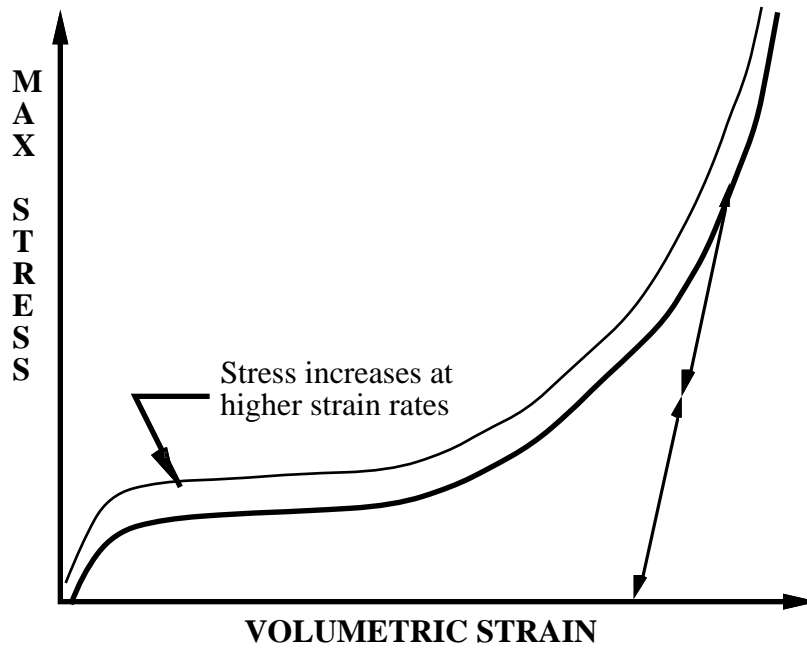
MID	Material identification. A unique number has to be chosen.
RO	Mass density
E	Young's modulus
PR	Poisson's ratio
LCID	Load curve ID defining yield stress versus volumetric strain, $\gamma$ , see Figure 20.16.
TSC	Tensile stress cutoff
DAMP	Rate sensitivity via damping coefficient (.05<recommended value<.50).

**Remarks:**

The volumetric strain is defined in terms of the relative volume, V, as:

$$\gamma = 1.-V$$

The relative volume is defined as the ratio of the current to the initial volume.



**Figure 20.16.** Behavior of strain rate sensitive crushable foam. Unloading is elastic to the tension cutoff. Subsequent reloading follows the unloading curve.

**\*MAT\_RATE\_SENSITIVE\_POWERLAW\_PLASTICITY**

This is Material Type 64 which will model strain rate sensitive elasto-plastic material with a power law hardening. Optionally, the coefficients can be defined as functions of the effective plastic strain.

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	E	PR	K	M	N	E0
Type	I	F	F	F	F	F	F	F
Default	---	---	---	---	---	0.0001	---	0.0002

Card 2            1            2            3            4            5            6            7            8

Variable	VP	EPS0						
Type	F	F						
Default	0.0	1.0						

**VARIABLE**

**DESCRIPTION**

- MID            Material identification. A unique number has to be chosen.
- RO            Mass density
- E            Young's modulus of elasticity
- PR            Poisson's ratio
- K            Material constant, k. If k<0 the absolute value of k is taken as the load curve number that defines k as a function of effective plastic strain.
- M            Strain hardening coefficient, m. If m<0 the absolute value of m is taken as the load curve number that defines m as a function of effective plastic strain.

<u>VARIABLE</u>	<u>DESCRIPTION</u>
N	Strain rate sensitivity coefficient, n. If n<0 the absolute value of n is taken as the load curve number that defines n as a function of effective plastic strain.
E0	Initial strain rate (default = 0.0002)
VP	Formulation for rate effects: EQ.0.0: Scale yield stress (default) EQ.1.0: Viscoplastic formulation
EPS0	Factor to normalize strain rate EQ.1.0: Time units of seconds (default) EQ.1.E-3: Time units of milliseconds EQ.1.E-6: Time units of microseconds

**Remarks:**

This material model follows a constitutive relationship of the form:

$$\sigma = k\varepsilon^m \dot{\varepsilon}^n$$

where  $\sigma$  is the yield stress,  $\varepsilon$  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 in results can be dramatic.

\*MAT\_MODIFIED\_ZERILLI\_ARMSTRONG

This is Material Type 65 which is a rate and temperature sensitive plasticity model which is sometimes preferred in ordnance design calculations.

Card Format

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	G	E0	N	TROOM	PC	SPALL
Type	I	F	F	F	F	F	F	F

Card 2            1            2            3            4            5            6            7            8

Variable	C1	C2	C3	C4	C5	C6	EFAIL	VP
Type	F	F	F	F	F	F	F	F

Card 3            1            2            3            4            5            6            7            8

Variable	B1	B2	B3	G1	G2	G3	G4	BULK
Type	F	F	F	F	F	F	F	

VARIABLE

DESCRIPTION

- MID            Material identification. A unique number has to be chosen.
- RO            Mass density
- G            Shear modulus
- E0             $\dot{\epsilon}_0$ , factor to normalize strain rate
- N            n, exponent for bcc metal
- TROOM         $T_r$ , room temperature

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
PC	<i>pc</i> , Pressure cutoff
SPALL	Spall Type: EQ.1.0: minimum pressure limit, EQ.2.0: maximum principal stress, EQ.3.0: minimum pressure cutoff.
C1	C <sub>1</sub> , coefficients for flow stress, see notes below.
C2	C <sub>2</sub> , coefficients for flow stress, see notes below.
C3	C <sub>3</sub> , coefficients for flow stress, see notes below.
C4	C <sub>4</sub> , coefficients for flow stress, see notes below.
C5	C <sub>5</sub> , coefficients for flow stress, see notes below.
C6	C <sub>6</sub> , coefficients for flow stress, see notes below.
EFAIL	Failure strain for erosion
VP	Formulation for rate effects: EQ.0.0: Scale yield stress (default) EQ.1.0: Viscoplastic formulation
B1	B <sub>1</sub> , coefficients for polynomial to represent temperature dependency of flow stress yield.
B2	B <sub>2</sub>
B3	B <sub>3</sub>
G1	G <sub>1</sub> , coefficients for defining heat capacity and temperature dependency of heat capacity.
G2	G <sub>2</sub>
G3	G <sub>3</sub>
G4	G <sub>4</sub>
BULK	Bulk modulus defined for shell elements only. Do not input for solid elements.

**Remarks:**

The Armstrong-Zerilli Material Model expresses the flow stress as follows.

For fcc metals ( $n=0$ ),

$$\sigma = C_1 + \left\{ C_2 (\varepsilon^p)^{1/2} \left[ e^{(-C_3 + C_4 \ln(\dot{\varepsilon}^*))T} \right] + C_5 \right\} \left( \frac{\mu(T)}{\mu(293)} \right)$$

$\varepsilon^p$  = effective plastic strain

$\dot{\varepsilon}^* = \frac{\dot{\varepsilon}}{\varepsilon_0}$  effective plastic strain rate where  $\varepsilon_0 = 1, 1e-3, 1e-6$  for time units of seconds, milliseconds, and microseconds, respectively.

For bcc metals ( $n > 0$ ),

$$\sigma = C_1 + C_2 e^{(-C_3 + C_4 \ln(\dot{\varepsilon}^*))T} + \left[ C_5 (\varepsilon^p)^n + C_6 \right] \left( \frac{\mu(T)}{\mu(293)} \right)$$

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.

**\*MAT\_LINEAR\_ELASTIC\_DISCRETE\_BEAM**

This is Material Type 66. This material model is defined for simulating the effects of a linear elastic beam by using six springs each acting about one of the six local degrees-of-freedom. The two nodes defining a beam may be coincident to give a zero length beam, or offset to give a finite length beam. The distance between the nodes of a beam should not affect the behavior of this model. A triad is used to orient the beam for the directional springs. Translational/rotational stiffness and viscous damping effects are considered for a local cartesian system, see notes below. Applications for this element include the modeling of joint stiffnesses.

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	TKR	TKS	TKT	RKR	RKS	RKT
Type	I	F	F	F	F	F	F	F

Card 2

Variable	TDR	TDS	TDT	RDR	RDS	RDT		
Type	F	F	F	F	F	F		

**VARIABLE****DESCRIPTION**

MID	Material identification. A unique number has to be chosen.
RO	Mass density, see also “volume” in the *SECTION_BEAM definition.
TKR	Translational stiffness about local r-axis, see notes below.
TKS	Translational stiffness about local s-axis
TKT	Translational stiffness about local t-axis
RKR	Rotational stiffness about the local r-axis
RKS	Rotational stiffness about the local s-axis
RKT	Rotational stiffness about the local t-axis



---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
TDR	Translational viscous damper about local r-axis. (Optional).
TDS	Translational viscous damper about local s-axis. (Optional).
TDT	Translational viscous damper about local t-axis. (Optional).
RDR	Rotational viscous damper about the local r-axis. (Optional).
RDS	Rotational viscous damper about the local s-axis. (Optional).
RDT	Rotational viscous damper about the local t-axis. (Optional).

**Remarks:**

The formulation of the discrete beam (type 6) assumes that the beam is of zero length and requires no orientation node. A small distance between the nodes joined by the beam is permitted. The local coordinate system which determines (r,s,t) is given by the coordinate ID, see \*DEFINE\_COORDINATE\_OPTION, in the cross sectional input, see \*SECTION\_BEAM, where the global system is the default. The local coordinate system axes can rotate with either node of the beam or an average rotation of both nodes (see SCOR variable in \*SECTION\_BEAM).

For null stiffness coefficients, no forces corresponding to these null values will develop. The viscous damping coefficients are optional.

**\*MAT\_NONLINEAR\_ELASTIC\_DISCRETE\_BEAM**

This is Material Type 67. This material model is defined for simulating the effects of nonlinear elastic and nonlinear viscous beams by using six springs each acting about one of the six local degrees-of-freedom. The two nodes defining a beam may be coincident to give a zero length beam, or offset to give a finite length beam. The distance between the nodes of a beam should not affect the behavior of this material model. A triad is used to orient the beam for the directional springs. Arbitrary curves to model transitional/ rotational stiffness and damping effects are allowed. See notes below.

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	LCIDTR	LCIDTS	LCIDTT	LCIDRR	LCIDRS	LCIDRT
Type	I	F	F	F	F	F	F	F

Card 2

Variable	LCIDTDR	LCIDTDS	LCIDTDT	LCIDRDR	LCIDRDS	LCIDRDT		
Type	F	F	F	F	F	F		

**VARIABLE****DESCRIPTION**

MID

Material identification. A unique number has to be chosen.

RO

Mass density, see also volume in \*SECTION\_BEAM definition.

LCIDTR

Load curve ID defining translational force resultant along local r-axis versus relative translational displacement, see Remarks and Figure 20.17.

LCIDTS

Load curve ID defining translational force resultant along local s-axis versus relative translational displacement.

LCIDTT

Load curve ID defining translational force resultant along local t-axis versus relative translational displacement.

LCIDRR

Load curve ID defining rotational moment resultant about local r-axis versus relative rotational displacement.

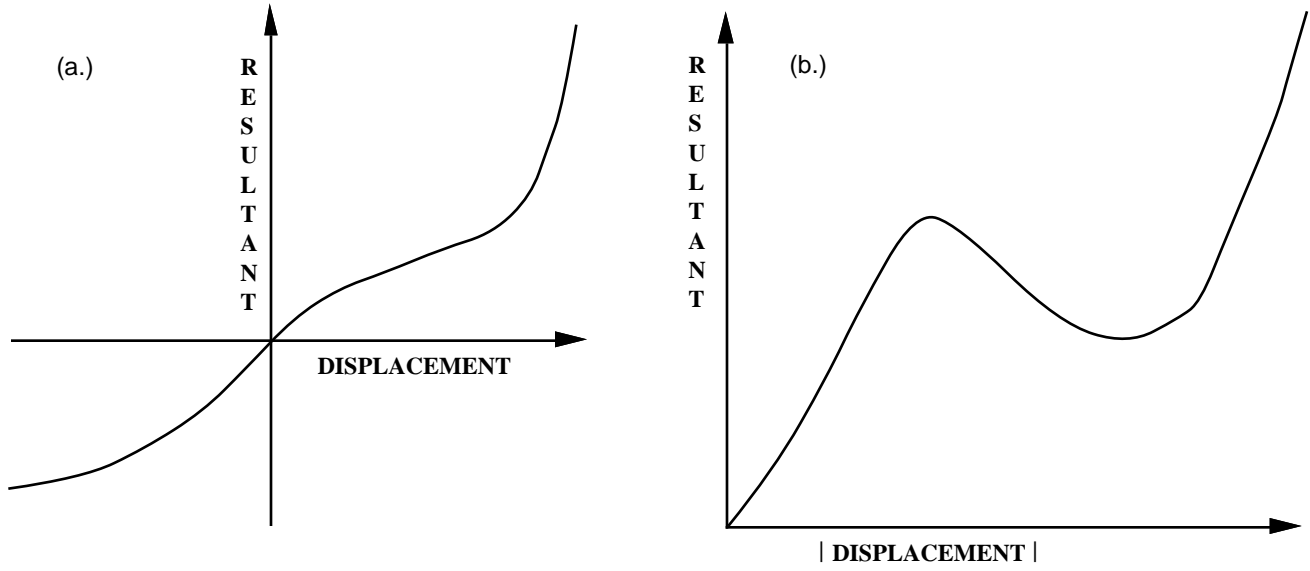
<u>VARIABLE</u>	<u>DESCRIPTION</u>
LCIDRS	Load curve ID defining rotational moment resultant about local s-axis versus relative rotational displacement.
LCIDRT	Load curve ID defining rotational moment resultant about local t-axis versus relative rotational displacement.
LCIDTDR	Load curve ID defining translational damping force resultant along local r-axis versus relative translational velocity.
LCIDTDS	Load curve ID defining translational damping force resultant along local s-axis versus relative translational velocity.
LCIDTDT	Load curve ID defining translational damping force resultant along local t-axis versus relative translational velocity.
LCIDRDR	Load curve ID defining rotational damping moment resultant about local r-axis versus relative rotational velocity.
LCIDRDS	Load curve ID defining rotational damping moment resultant about local s-axis versus relative rotational velocity.
LCIDRDT	Load curve ID defining rotational damping moment resultant about local t-axis versus relative rotational velocity.

**Remarks:**

For null load curve ID's, no forces are computed.

The formulation of the discrete beam (type 6) assumes that the beam is of zero length and requires no orientation node. A small distance between the nodes joined by the beam is permitted. The local coordinate system which determines (r,s,t) is given by the coordinate ID, see \*DEFINE\_COORDINATE\_OPTION, in the cross sectional input, see \*SECTION\_BEAM, where the global system is the default. The local coordinate system axes can rotate with either node of the beam or an average rotation of both nodes (see SCOOR variable in \*SECTION\_BEAM).

If different behavior in tension and compression is desired in the calculation of the force resultants, the load curve(s) must be defined in the negative quadrant starting with the most negative displacement then increasing monotonically to the most positive. If the load curve behaves similarly in tension and compression, define only the positive quadrant. Whenever displacement values fall outside of the defined range, the resultant forces will be extrapolated. Figure 20.17 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 20.17.** 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.

\*MAT\_NONLINEAR\_PLASTIC\_DISCRETE\_BEAM

This is Material Type 68. This material model is defined for simulating the effects of nonlinear elastoplastic, linear viscous behavior of beams by using six springs each acting about one of the six local degrees-of-freedom. The two nodes defining a beam may be coincident to give a zero length beam, or offset to give a finite length beam. The distance between the nodes of a beam should not affect the behavior of this material model. A triad is used to orient the beam for the directional springs. Translational/rotational stiffness and damping effects can be considered. The plastic behavior is modelled using force/moment curves versus displacements/ rotation. Optionally, failure can be specified based on a force/moment criterion and a displacement/ rotation criterion. See also notes below.

Card Format

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	TKR	TKS	TKT	RKR	RKS	RKT
Type	I	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	none	none

Card 2

Variable	TDR	TDS	TDT	RDR	RDS	RDT		
Type	F	F	F	F	F	F		
Default	none	none	none	none	none	none		

Card 3

Variable	LCPDR	LCPDS	LCPDT	LCPMR	LCPMS	LCPMT		
Type	F	F	F	F	F	F		
Default	0.0	0.0	0.0	0.0	0.0	0.0		

Card 4

1 2 3 4 5 6 7 8

Variable	FFAILR	FFAILS	FFAILT	MFAILR	MFAILS	MFAILT		
Type	F	F	F	F	F	F		
Default	0.0	0.0	0.0	0.0	0.0	0.0		

Card 5

Variable	UFAILR	UFAILS	UFAILT	TFAILR	TFAILS	TFAILT		
Type	F	F	F	F	F	F		
Default	0.0	0.0	0.0	0.0	0.0	0.0		

**VARIABLE****DESCRIPTION**

MID Material identification. A unique number has to be chosen.

RO Mass density, see also volume on \*SECTION\_BEAM definition.

TKR Translational stiffness about local r-axis

TKS Translational stiffness about local s-axis

TKT Translational stiffness about local t-axis

RKR Rotational stiffness about the local r-axis

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
RKS	Rotational stiffness about the local s-axis
RKT	Rotational stiffness about the local t-axis
TDR	Translational viscous damper about local r-axis
TDS	Translational viscous damper about local s-axis
TDT	Translational viscous damper about local t-axis
RDR	Rotational viscous damper about the local r-axis
RDS	Rotational viscous damper about the local s-axis
RDT	Rotational viscous damper about the local t-axis
LCPDR	Load curve ID-yield force versus plastic displacement r-axis. If the curve ID zero, and if TKR is nonzero, then nonlinear elastic behavior is obtained for this component.
LCPDS	Load curve ID-yield force versus plastic displacement s-axis. If the curve ID zero, and if TKS is nonzero, then nonlinear elastic behavior is obtained for this component.
LCPDT	Load curve ID-yield force versus plastic displacement t-axis. If the curve ID zero, and if TKT is nonzero, then nonlinear elastic behavior is obtained for this component.
LCPMR	Load curve ID-yield moment versus plastic rotation r-axis. If the curve ID zero, and if RKR is nonzero, then nonlinear elastic behavior is obtained for this component.
LCPMS	Load curve ID-yield moment versus plastic rotation s-axis. If the curve ID zero, and if RKS is nonzero, then nonlinear elastic behavior is obtained for this component.
LCPMT	Load curve ID-yield moment versus plastic rotation t-axis. If the curve ID zero, and if RKT is nonzero, then nonlinear elastic behavior is obtained for this component.
FFAILR	Optional failure parameter. If zero, the corresponding force, $F_r$ , is not considered in the failure calculation.
FFAILS	Optional failure parameter. If zero, the corresponding force, $F_s$ , is not considered in the failure calculation.
FFAILT	Optional failure parameter. If zero, the corresponding force, $F_t$ , is not considered in the failure calculation.
MFAILR	Optional failure parameter. If zero, the corresponding moment, $M_r$ , is not considered in the failure calculation.

---

<b>VARIABLE</b>	<b>DESCRIPTION</b>
MFAILS	Optional failure parameter. If zero, the corresponding moment, $M_s$ , is not considered in the failure calculation.
MFAILT	Optional failure parameter. If zero, the corresponding moment, $M_t$ , is not considered in the failure calculation.
UFAILR	Optional failure parameter. If zero, the corresponding displacement, $u_r$ , is not considered in the failure calculation.
UFAILS	Optional failure parameter. If zero, the corresponding displacement, $u_s$ , is not considered in the failure calculation.
UFAILT	Optional failure parameter. If zero, the corresponding displacement, $u_t$ , is not considered in the failure calculation.
TFAILR	Optional failure parameter. If zero, the corresponding rotation, $\theta_r$ , is not considered in the failure calculation.
TFAILS	Optional failure parameter. If zero, the corresponding rotation, $\theta_s$ , is not considered in the failure calculation.
TFAILT	Optional failure parameter. If zero, the corresponding rotation, $\theta_t$ , is not considered in the failure calculation.

**Remarks:**

For the translational and rotational degrees of freedom where elastic behavior is desired, set the load curve ID to zero.

The formulation of the discrete beam (type 6) assumes that the beam is of zero length and requires no orientation node. A small distance between the nodes joined by the beam is permitted. The local coordinate system which determines (r,s,t) is given by the coordinate ID (see \*DEFINE\_COORDINATE\_OPTION) in the cross sectional input, see \*SECTION\_BEAM, where the global system is the default. The local coordinate system axes can rotate with either node of the beam or an average rotation of both nodes (see SCOOR variable in \*SECTION\_BEAM).

Catastrophic failure based on force resultants occurs if the following inequality is satisfied.

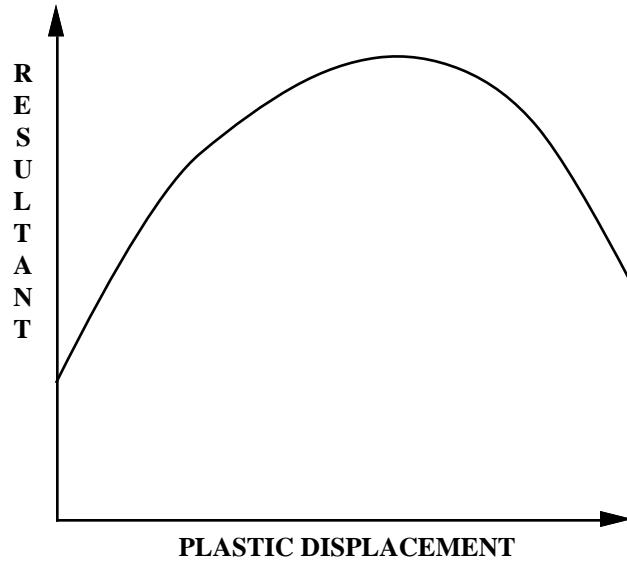
$$\left(\frac{F_r}{F_r^{fail}}\right)^2 + \left(\frac{F_s}{F_s^{fail}}\right)^2 + \left(\frac{F_t}{F_t^{fail}}\right)^2 + \left(\frac{M_r}{M_r^{fail}}\right)^2 + \left(\frac{M_s}{M_s^{fail}}\right)^2 + \left(\frac{M_t}{M_t^{fail}}\right)^2 - 1 \geq 0.$$

After failure the discrete element is deleted. Likewise, catastrophic failure based on displacement resultants occurs if the following inequality is satisfied:

$$\left(\frac{u_r}{u_r^{fail}}\right)^2 + \left(\frac{u_s}{u_s^{fail}}\right)^2 + \left(\frac{u_t}{u_t^{fail}}\right)^2 + \left(\frac{\theta_r}{\theta_r^{fail}}\right)^2 + \left(\frac{\theta_s}{\theta_s^{fail}}\right)^2 + \left(\frac{\theta_t}{\theta_t^{fail}}\right)^2 - 1 \geq 0.$$

After failure the discrete element is deleted. If failure is included either one or both of the criteria may be used.





**Figure 20.18.** The resultant forces and moments are limited by the yield definition. The initial yield point corresponds to a plastic displacement of zero.

**\*MAT\_SID\_DAMPER\_DISCRETE\_BEAM**

This is Material Type 69. The side impact dummy uses a damper that is not adequately treated by the nonlinear force versus relative velocity curves since the force characteristics are dependent on the displacement of the piston. See also notes below.

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	ST	D	R	H	K	C
Type	I	F	F	F	F	F	F	F

Card 2

Variable	C3	STF	RHOF	C1	C2	LCIDF	LCIDD	S0
Type	F	F	F	F	F	F	F	F

**Read in up to 15 orifice locations with orifice location per card. Input is terminated when a “\*” card is found. On the first card below the optional input parameters SF and DF may be specified.**

Cards 3,...

Variable	ORFLOC	ORFRAD	SF	DC				
Type	F	F	F	F				

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density, see also volume on *SECTION_BEAM definition.
ST	$S_t$ , piston stroke. $S_t$ must equal or exceed the length of the beam element, see Figure 20.19 below.
D	d, piston diameter
R	R, default orifice radius
H	h, orifice controller position
K	K, damping constant  LT.0.0:  K  is the load curve number ID, see *DEFINE_CURVE, defining the damping coefficient as a function of the <u>absolute</u> value of the relative velocity.
C	C, discharge coefficient
C3	Coefficient for fluid inertia term
STF	k, stiffness coefficient if piston bottoms out
RHOF	$\rho_{fluid}$ , fluid density
C1	$C_1$ , coefficient for linear velocity term
C2	$C_2$ , coefficient for quadratic velocity term
LCIDF	Load curve number ID defining force versus piston displacement, s, i.e., term $f(s + s_0)$ . Compressive behavior is defined in the positive quadrant of the force displacement curve. Displacements falling outside of the defined force displacement curve are extrapolated. Care must be taken to ensure that extrapolated values are reasonable.
LCIDD	Load curve number ID defining damping coefficient versus piston displacement, s, i.e., $g(s + s_0)$ . Displacements falling outside the defined curve are extrapolated. Care must be taken to ensure that extrapolated values are reasonable.
S0	Initial displacement $s_0$ , typically set to zero. A positive displacement corresponds to compressive behavior.
ORFLOC	$d_i$ , orifice location of ith orifice relative to the fixed end.
ORFRAD	$r_i$ , orifice radius of ith orifice, if zero the default radius is used.

<b>VARIABLE</b>	<b>DESCRIPTION</b>
SF	Scale factor on calculated force. The default is set to 1.0
DC	c, linear viscous damping coefficient used after damper bottoms out either in tension or compression.

**Remarks:**

As the damper moves, the fluid flows through the open orifices to provide the necessary damping resistance. While moving as shown in Figure 20.19, 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,

$$F = SF \left\{ KA_p V_p \left\{ \frac{C_1}{A_0^t} + C_2 |V_p| \rho_{fluid} \left[ \left( \frac{A_p}{CA_0^t} \right)^2 - 1 \right] \right\} - f(s + s_0) + V_p g(s + s_0) \right\}$$

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,  $\rho_{fluid}$  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 20.20.

The factor,  $SF$ , which scales the force defaults to 1.0 and is analogous to the adjusting ring on the damper.

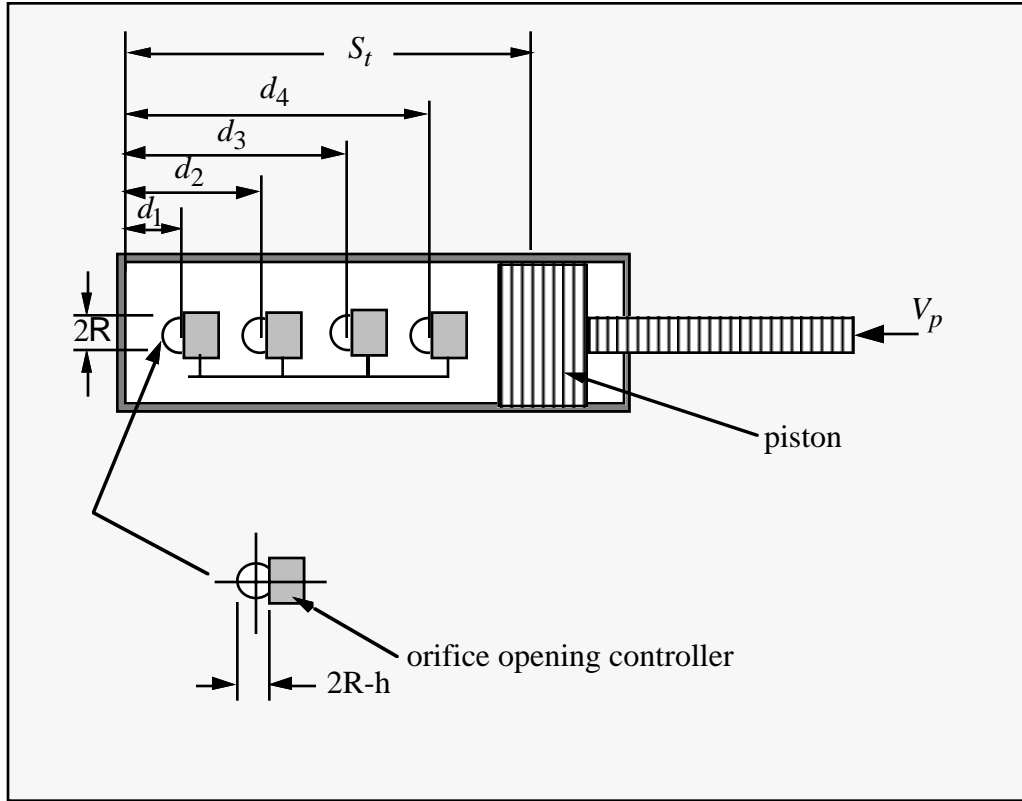


Figure 20.19. Mathematical model for the Side Impact Dummy damper.

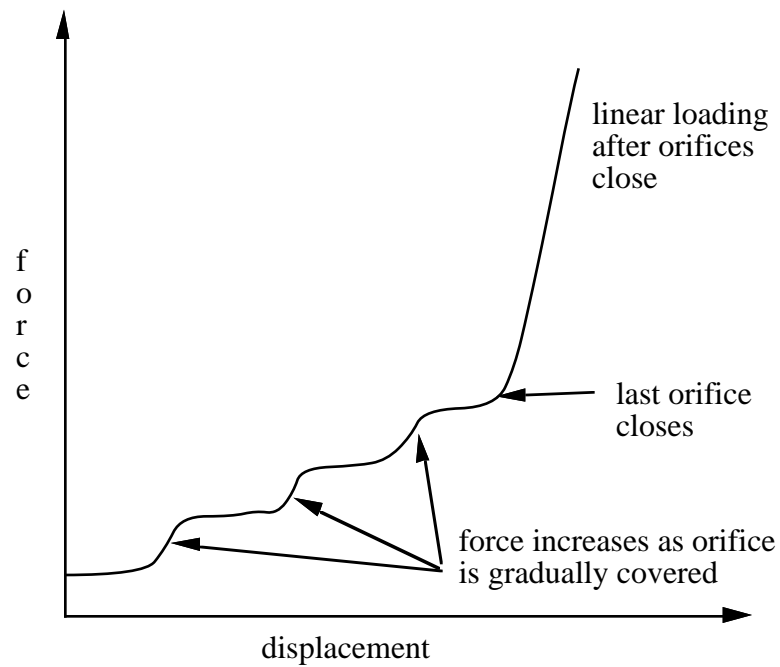


Figure 20.20. Force versus displacement as orifices are covered at a constant relative velocity. Only the linear velocity term is active.

# \*MAT

## \*MAT\_HYDRAULIC\_GAS\_DAMPER\_DISCRETE\_BEAM

### \*MAT\_HYDRAULIC\_GAS\_DAMPER\_DISCRETE\_BEAM

This is Material Type 70. This special purpose element represents a combined hydraulic and gas-filled damper which has a variable orifice coefficient. A schematic of the damper is shown in Figure 20.21. Dampers of this type are sometimes used on buffers at the end of railroad tracks and as aircraft undercarriage shock absorbers. This material can be used only as a discrete beam element. See also notes below.

#### Card Format

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	CO	N	P0	PA	AP	KH
Type	I	F	F	F	F	F	F	F

Card 2

Variable	LCID	FR	SCLF	CLEAR				
Type	F	F	F	F				

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MID	Material identification. A unique number has to be chosen.
RO	Mass density, see also volume in *SECTION_BEAM defintion.
CO	Length of gas column, $C_0$
N	Adiabatic constant
P0	Initial gas pressure, $P_0$
PA	Atmospheric pressure, $P_a$
AP	Piston cross sectional area, $A_p$
KH	Hydraulic constant, $K$

VARIABLE	DESCRIPTION
LCID	Load curve ID, see *DEFINE_CURVE, defining the orifice area, $a_0$ , versus element deflection.
FR	Return factor on orifice force. This acts as a factor on the hydraulic force only and is applied when unloading. It is intended to represent a valve that opens when the piston unloads to relieve hydraulic pressure. Set it to 1.0 for no such relief.
SCLF	Scale factor on force. (Default = 1.0)
CLEAR	Clearance (if nonzero, no tensile force develops for positive displacements and negative forces develop only after the clearance is closed).

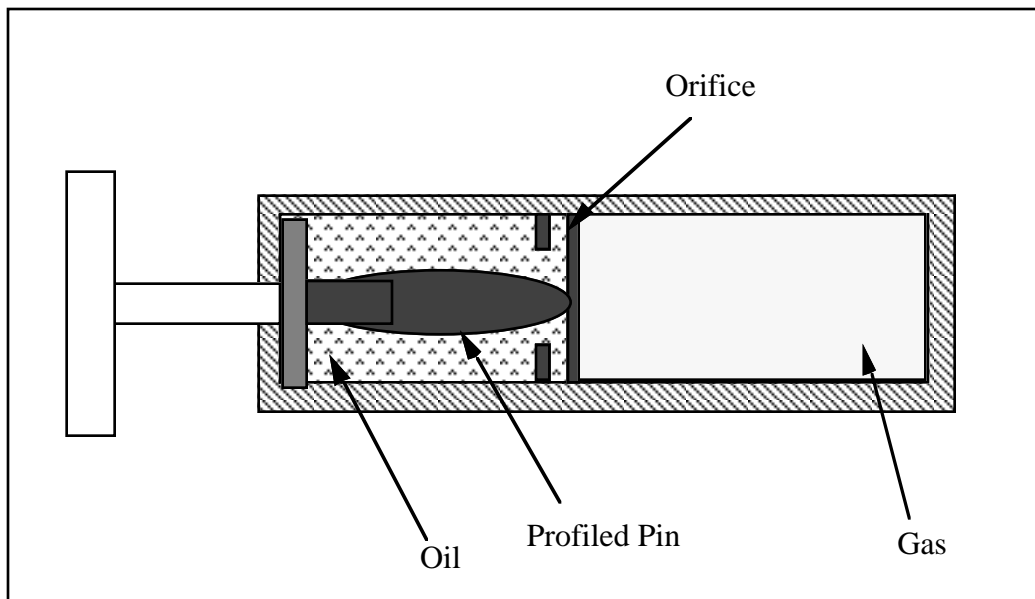


Figure 20.21. Schematic of Hydraulic/Gas damper.

**Remarks:**

As the damper is compressed two actions contribute to the force which develops. First, the gas is adiabatically compressed into a smaller volume. Secondly, oil is forced through an orifice. A profiled pin may occupy some of the cross-sectional area of the orifice; thus, the orifice area available for the oil varies with the stroke. The force is assumed proportional to the square of the velocity and inversely proportional to the available area.

The equation for this element is:

$$F = SCLF \cdot \left\{ K_h \left( \frac{V}{a_0} \right)^2 + \left[ P_0 \left( \frac{C_0}{C_0 - S} \right)^n - P_a \right] \cdot A_p \right\}$$

where S is the element deflection and V is the relative velocity across the element.

**\*MAT\_CABLE\_DISCRETE\_BEAM**

This is Material Type 71. This model permits elastic cables to be realistically modelled; thus, no force will develop in compression.

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	E	LCID	F0			
Type	I	F	F	F	F			

**VARIABLE****DESCRIPTION**

MID	Material identification. A unique number has to be chosen.
RO	Mass density, see also volume in *SECTION_BEAM definition.
E	GT. 0.0: Young's modulus LT. 0.0: Stiffness
LCID	Load curve ID, see *DEFINE_CURVE, defining the stress versus engineering strain. (Optional).
F0	Initial tensile force. If F0 is defined, an offset is not needed for an initial tensile force.

**Remarks:**

The force,  $F$ , generated by the cable is nonzero if and only if the cable is tension. The force is given by:

$$F = \max(F_0 + K\Delta L, 0.)$$

where  $\Delta L$  is the change in length

$$\Delta L = \text{current length} - (\text{initial length} - \text{offset})$$

and the stiffness ( $E > 0.0$  only ) is defined as:

$$K = \frac{E \cdot \text{area}}{(\text{initial length} - \text{offset})}$$

Note that a constant force element can be obtained by setting:



$$F_0 > 0 \quad \text{and} \quad K = 0$$

although the application of such an element is unknown.

The area and offset are defined on either the cross section or element cards. For a slack cable the offset should be input as a negative length. For an initial tensile force the offset should be positive.

If a load curve is specified the Young's modulus will be ignored and the load curve will be used instead. The points on the load curve are defined as engineering stress versus engineering strain, i.e., the change in length over the initial length. The unloading behavior follows the loading.

**\*MAT\_CONCRETE\_DAMAGE**

This is Material Type 72. This model has been used to analyze buried steel reinforced concrete structures subjected to impulsive loadings.

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	PR					
Type	I	F	F					
Default	none	none	none					

Card 2

Variable	SIGF	A0	A1	A2				
Type	F	F	F	F				
Default	0.0	0.0	0.0	0.0				

Card 3

Variable	A0Y	A1Y	A2Y	A1F	A2F	B1	B2	B3
Type	F	F	F	F	F	F	F	F
Default	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0

Card 4

Variable	PER	ER	PRR	SIGY	ETAN	LCP	LCR	
Type	F	F	F	F	F	F	F	
Default	0.0	0.0	0.0	none	0.0	none	none	

Card 5

Variable	$\lambda_1$	$\lambda_2$	$\lambda_3$	$\lambda_4$	$\lambda_5$	$\lambda_6$	$\lambda_7$	$\lambda_8$
Type	F	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	none	none

Card 6

Variable	$\lambda_9$	$\lambda_{10}$	$\lambda_{11}$	$\lambda_{12}$	$\lambda_{13}$			
Type	F	F	F	F	F			
Default	none	none	none	none	none			

Card 7

1 2 3 4 5 6 7 8

Variable	$\eta_1$	$\eta_2$	$\eta_3$	$\eta_4$	$\eta_5$	$\eta_6$	$\eta_7$	$\eta_8$
Type	F	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	none	none

Card 8

Variable	$\eta_9$	$\eta_{10}$	$\eta_{11}$	$\eta_{12}$	$\eta_{13}$			
Type	F	F	F	F	F			
Default	none	none	none	none	none			

**VARIABLE****DESCRIPTION**

MID	Material identification. A unique number has to be chosen.
RO	Mass density.
PR	Poisson's ratio.
SIGF	Maximum principal stress for failure.
A0	Cohesion.
A1	Pressure hardening coefficient.
A2	Pressure hardening coefficient.
A0Y	Cohesion for yield
A1Y	Pressure hardening coefficient for yield limit
A2Y	Pressure hardening coefficient for yield limit
A1F	Pressure hardening coefficient for failed material.
A2F	Pressure hardening coefficient for failed material.
B1	Damage scaling factor.
B2	Damage scaling factor for uniaxial tensile path.
B3	Damage scaling factor for triaxial tensile path.
PER	Percent reinforcement.
ER	Elastic modulus for reinforcement.
PRR	Poisson's ratio for reinforcement.

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
SIGY	Initial yield stress.
ETAN	Tangent modulus/plastic hardening modulus.
LCP	Load curve ID giving rate sensitivity for principal material, see *DEFINE_CURVE.
LCR	Load curve ID giving rate sensitivity for reinforcement, see *DEFINE_CURVE.
$\lambda_1$ – $\lambda_{13}$	Tabulated damage function
$\eta_1$ – $\eta_{13}$	Tabulated scale factor.

**Remarks:**

Cohesion for failed material  $a_0f = 0.0$   
 $b_3$  must be positive or zero.  
 $\lambda_n < \lambda_{n+1}$  . The first point must be zero.

**\*MAT\_LOW\_DENSITY\_VISCOUS\_FOAM**

This is Material Type 73. It is mainly for Modeling Low Density Urethane Foam with high compressibility and with rate sensitivity which can be characterized by a relaxation curve. Its main applications are for seat cushions, padding on the Side Impact Dummies (SID), bumpers, and interior foams. Optionally, a tension cut-off failure can be defined. Also, see the notes below and the discription of material 57: \*MAT\_LOW\_DENSITY\_FOAM.

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	E	LCID	TC	HU	BETA	DAMP
Type	I	F	F	F	F	F	F	F
Default	---	---	---	---	1.E+20	1.		
Remarks	---	---	---	---	---	3	1	---

Card 2

Variable	SHAPE	FAIL	BVFLAG	KCON	LCID2	BSTART	TRAMP	NV
Type	F	F	F	F	F	F	F	I
Default	1.0	0.0	0.0	0.0	0	0.0	0.0	6

If LCID2 = 0 then defined the following viscoelastic constants. Up to 6 cards may be input. A keyword card (with a “\*” in column 1) terminates this input if less than 6 cards are used. If LCID2 is nonzero skip this input. The variable REF is taken from the first card of this sequence.

Optional Cards	1	2	3	4	5	6	7	8
Variable	GI	BETAI	REF					
Type	F	F	F					

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MID	Material identification. A unique number has to be chosen.
RO	Mass density
E	Young’s modulus
LCID	Load curve ID, see *DEFINE_CURVE, for nominal stress versus strain.
TC	Tension cut-off stress
HU	Hysteretic unloading factor between 0 and 1 (default=1, i.e., no energy dissipation), see also Figure 20.14.
BETA	$\beta$ , decay constant to model creep in unloading. EQ.0: No relaxation.
DAMP	Viscous coefficient (.05< recommended value <.50) to model damping effects.
SHAPE	Shape factor for unloading. Active for nonzero values of the hysteretic unloading factor. Values less than one reduces the energy dissipation and greater than one increases dissipation, see also Figure 20.14.
FAIL	Failure option after cutoff stress is reached: EQ.0.0: tensile stress remains at cut-off value, EQ.1.0: tensile stress is reset to zero.
BVFLAG	Bulk viscosity activation flag, see remark below: EQ.0.0: no bulk viscosity (recommended), EQ.1.0: bulk viscosity active.

<b>VARIABLE</b>	<b>DESCRIPTION</b>
KCON	Stiffness coefficient for contact interface stiffness. Maximum slope in stress vs. strain curve is used. When the maximum slope is taken for the contact, the time step size for this material is reduced for stability. In some cases $\Delta t$ may be significantly smaller, and defining a reasonable stiffness is recommended.
LCID2	Load curve ID of relaxation curve. If constants $\beta_i$ are determined via a least squares fit. This relaxation curve is shown in Figure 20.23. This model ignores the constant stress.
BSTART	Fit parameter. In the fit, $\beta_1$ is set to zero, $\beta_2$ is set to BSTART, $\beta_3$ is 10 times $\beta_2$ , $\beta_4$ is 100 times greater than $\beta_3$ , and so on. If zero, BSTART=.01.
TRAMP	Optional ramp time for loading.
NV	Number of terms in fit. If zero, the default is 6. Currently, the maximum number is set to 6. Values of 2 or 3 are recommended, since each term used adds significantly to the cost. Caution should be exercised when taking the results from the fit. Preferably, all generated coefficients should be positive. Negative values may lead to unstable results. Once a satisfactory fit has been achieved it is recommended that the coefficients which are written into the output file be input in future runs.
GI	Optional shear relaxation modulus for the <i>i</i> th term
BETAI	Optional decay constant if <i>i</i> th term
REF	Use reference geometry to initialize the stress tensor. The reference geometry is defined by the keyword: *INITIAL_FOAM_REFERENCE_GEOMETRY. This option is currently restricted to 8-noded solid elements with one point integration. EQ.0.0: off, EQ.1.0: on.

**Remarks:**

This viscoelastic foam model is available to model highly compressible viscous foams. The hyperelastic formulation of this models follows that of material 57.

Rate effects are accounted for through linear viscoelasticity by a convolution integral of the form

$$\sigma_{ij}^r = \int_0^t g_{ijkl}(t - \tau) \frac{\partial \epsilon_{kl}}{\partial \tau} d\tau$$

where  $g_{ijkl}(t - \tau)$  is the relaxation function. The stress tensor,  $\sigma_{ij}^r$ , augments the stresses determined from the foam,  $\sigma_{ij}^f$ ; consequently, the final stress,  $\sigma_{ij}$ , 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.

**\*MAT\_BILKHU/DUBOIS\_FOAM**

This is Material Type 75. This model is for the simulation of isotropic crushable forms. Uniaxial and triaxial test data have to be used. For the elastic response, the Poisson ratio is set to zero.

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	YM	LCPY	LCUYS	VC		
Type	I	F	F	F	F	F		

**VARIABLE****DESCRIPTION**

MID	Material identification. A unique number has to be chosen.
RO	Mass density
YM	Young's modulus (E)
LCPY	Load curve ID giving pressure for plastic yielding versus volumetric strain, see Figure 20.22.
LCUYS	Load curve ID giving uniaxial yield stress versus volumetric strain, see Figure 20.22.
VC	Viscous damping coefficient (.05<recommended value<.50).

**Remarks:**

The logarithmic volumetric strain is defined in terms of the relative volume,  $V$ , as:

$$\gamma = -\ln(V)$$

In defining the curves the stress and strain pairs should be positive values starting with a volumetric strain value of zero.

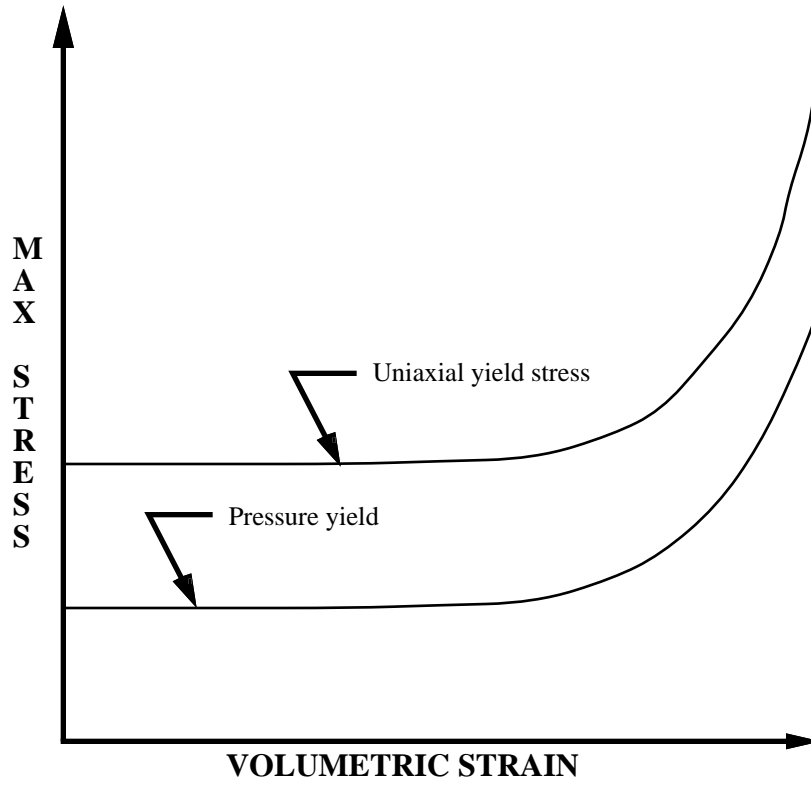


Figure 20.22. Behavior of crushable foam. Unloading is elastic.

The yield surface is defined as an ellipse in the equivalent pressure and von Mises stress plane.

**\*MAT\_GENERAL\_VISCOELASTIC**

This is Material Type 76. This material model provides a general viscoelastic Maxell model having up to 6 terms in the prony series expansion and is useful for modeling dense continuum rubbers and solid explosives. Either the coefficients of the prony series expansion or a relaxation curve may be specified to define the viscoelastic deviatoric and bulk behavior.

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	BULK					
Type	I	F	F					

**Insert a blank card here if constants are defined on cards 3,4,... below.**

Card 2            1            2            3            4            5            6            7            8

Variable	LCID	NT	BSTART	TRAMP	LCIDK	NTK	BSTARTK	TRAMPK
Type	F	I	F	F	F	I	F	F

**Card Format for viscoelastic constants. Up to 6 cards may be input. A keyword card (with a "\*" in column 1) terminates this input if less than 6 cards are used. These cards are not needed if relaxation data is defined. The number of terms for the shear behavior may differ from that for the bulk behavior: simply insert zero if a term is not included.**

Optional            1            2            3            4            5            6            7            8  
Cards

Variable	GI	BETAI	KI	BETAKI				
Type	F	F	F	F				

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
BULK	Elastic bulk modulus.
LCID	Load curve ID for deviatoric behavior if constants, $G_i$ , and $\beta_i$ are determined via a least squares fit. This relaxation curve is shown below.
NT	Number of terms in shear fit. If zero the default is 6. Currently, the maximum number is set to 6.
BSTART.	In the fit, $\beta_1$ is set to zero, $\beta_2$ is set to BSTART, $\beta_3$ is 10 times $\beta_2$ , $\beta_4$ is 100 times greater than $\beta_3$ , and so on. If zero, BSTART=.01.
TRAMP	Optional ramp time for loading.
LCIDK	Load curve ID for bulk behavior if constants, $K_i$ , and $\beta\kappa_i$ are determined via a least squares fit. This relaxation curve is shown below.
NTK	Number of terms desired in bulk fit. If zero the default is 6. Currently, the maximum number is set to 6.
BSTARTK.	In the fit, $\beta\kappa_1$ is set to zero, $\beta\kappa_2$ is set to BSTARTK, $\beta\kappa_3$ is 10 times $\beta\kappa_2$ , $\beta\kappa_4$ is 100 times greater than $\beta\kappa_3$ , and so on. If zero, BSTARTK=.01.
TRAMPK	Optional ramp time for bulk loading.
GI	Optional shear relaxation modulus for the ith term
BETAI	Optional shear decay constant for the ith term
KI	Optional bulk relaxation modulus for the ith term
BETAKI	Optional bulk decay constant for the ith term

**Remarks:**

Rate effects are taken into account through linear viscoelasticity by a convolution integral of the form:

$$\sigma_{ij} = \int_0^t g_{ijkl}(t - \tau) \frac{\partial \epsilon_{kl}}{\partial \tau} d\tau$$

where  $g_{ijkl}(t - \tau)$  is the relaxation functions for the different stress measures. This stress is added to the stress tensor determined from the strain energy functional.

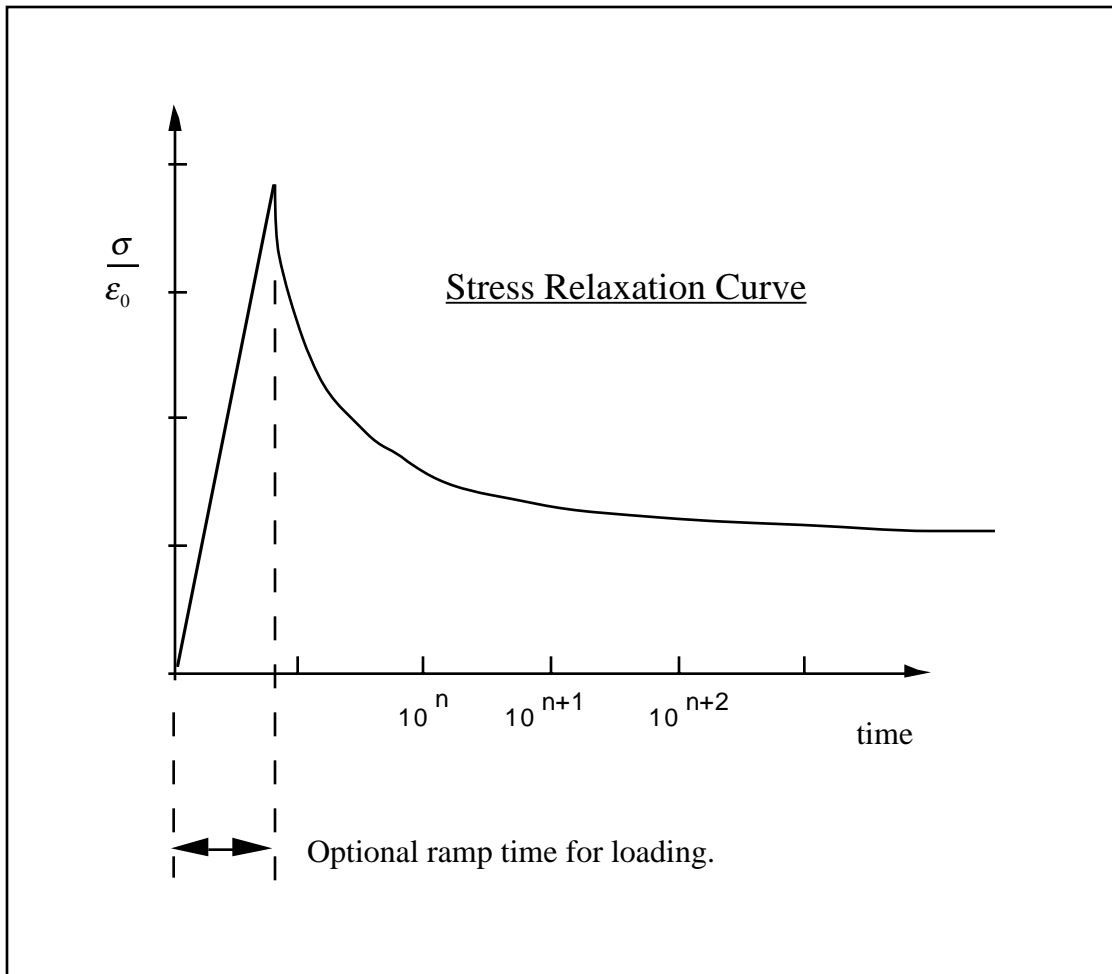
If we wish to include only simple rate effects, the relaxation function is represented by six terms from the Prony series:

$$g(t) = \sum_{m=1}^N G_m e^{-\beta_m t}$$

We characterize this in the input by shear moduli,  $G_i$ , and decay constants,  $\beta_i$ . An arbitrary number of terms, up to 6, may be used when applying the viscoelastic model.

For volumetric relaxation, the relaxation function is also represented by the Prony series in terms of bulk moduli:

$$k(t) = \sum_{m=1}^N K_m e^{-\beta_{k_m} t}$$



**Figure 20.23.** Relaxation curve. This curve defines stress versus time where time is defined on a logarithmic scale. For best results, the points defined in the load curve should be equally spaced on the logarithmic scale. Furthermore, the load curve should be smooth and defined in the positive quadrant. If nonphysical values are determined by least squares fit, LS-DYNA will terminate with an error message after the initialization phase is completed. If the ramp time for loading is included, then the relaxation which occurs during the loading phase is taken into account. This effect may or may not be important.

**\*MAT\_HYPERELASTIC\_RUBBER**

This is Material Type 77. This material model provides a general hyperelastic rubber model combined optionally with linear viscoelasticity as outlined by [Christensen 1980].

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	PR	N	NV			
Type	I	F	F	I	I			

**Card 2 if  $N > 0$ , a least squares fit is computed from uniaxial data**

**Card Format**

Card 2            1            2            3            4            5            6            7            8

Variable	SGL	SW	ST	LCID1	DATA	LCID2	not used	TRAMP
Type	F	F	F	F	F	F		F

**Card 2 if  $N = 0$  define the following constants**

**Card Format**

Card 2            1            2            3            4            5            6            7            8

Variable	C10	C01	C11	C20	C02	C30		
Type	F	F	F	F	F	F		



Card Format for Viscoelastic Constants. Up to 6 cards may be input. A keyword card (with a "\*" in column 1) terminates this input if less than 6 cards are used.

Optional Cards

1 2 3 4 5 6 7 8

Variable	GI	BETAI						
Type	F	F						

VARIABLE

DESCRIPTION

MID	Material identification. A unique number has to be chosen.
RO	Mass density
PR	Poissons ratio (>.49 is recommended, smaller values may not work and should not be used).
N	Number of constants to solve for: EQ.1: Solve for C10 and C01 EQ.2: Solve for C10, C01, C11, C20, and C02 EQ.3: Solve for C10, C01, C11, C20, C02, and C30
NV	Number of Prony series terms in fit. If zero, the default is 6. Currently, the maximum number is set to 6. Values less than 6, possibly 3-5 are recommended, since each term used adds significantly to the cost. Caution should be exercised when taking the results from the fit. Preferably, all generated coefficients should be positive. Negative values may lead to unstable results. Once a satisfactory fit has been achieved it is recommended that the coefficients which are written into the output file be input in future runs.

f N>0 test information from a uniaxial test are used:

SGL	Specimen gauge length
SW	Specimen width
ST	Specimen thickness
LCID1	Load curve ID giving the force versus actual change in the gauge length
DATA	Type of experimental data. EQ.0.0: uniaxial data (Only option for this model)

<b>VARIABLE</b>	<b>DESCRIPTION</b>
LCID2	Load curve ID of relaxation curve If constants $\beta t$ are determined via a least squares fit. This relaxation curve is shown in Figure 20.23. This model ignores the constant stress.
TRAMP	Optional ramp time for loading.

If N=0, the following constants have to be defined:

C10	C <sub>10</sub>
C01	C <sub>01</sub>
C11	C <sub>11</sub>
C20	C <sub>20</sub>
C02	C <sub>02</sub>
C30	C <sub>30</sub>
GI	Optional shear relaxation modulus for the ith term
BETA I	Optional decay constant if ith term

### **Remarks:**

Rubber is generally considered to be fully incompressible since the bulk modulus greatly exceeds the shear modulus in magnitude. To model the rubber as an unconstrained material a hydrostatic work term,  $W_H(J)$ , is included in the strain energy functional which is function of the relative volume,  $J$ , [Ogden, 1984]:

$$W(J_1, J_2, J) = \sum_{p,q=0}^n C_{pq} (J_1 - 3)^p (J_2 - 3)^q + W_H(J)$$

$$J_1 = I_1 J^{-1/3}$$

$$J_2 = I_2 J^{-2/3}$$

In order to prevent volumetric work from contributing to the hydrostatic work the first and second invariants are modified as shown. This procedure is described in more detail by Sussman and Bathe [1987].

Rate effects are taken into account through linear viscoelasticity by a convolution integral of the form:

$$\sigma_{ij} = \int_0^t g_{ijkl}(t-\tau) \frac{\partial \epsilon_{kl}}{\partial \tau} d\tau$$

or in terms of the second Piola-Kirchhoff stress,  $S_{ij}$ , and Green's strain tensor,  $E_{ij}$ ,

$$S_{ij} = \int_0^t G_{ijkl}(t-\tau) \frac{\partial E_{kl}}{\partial \tau} d\tau$$

where  $g_{ijkl}(t-\tau)$  and  $G_{ijkl}(t-\tau)$  are the relaxation functions for the different stress measures. This stress is added to the stress tensor determined from the strain energy functional.

If we wish to include only simple rate effects, the relaxation function is represented by six terms from the Prony series:

$$g(t) = \alpha_0 + \sum_{m=1}^N \alpha_m e^{-\beta t}$$

given by,

$$g(t) = \sum_{i=1}^n G_i e^{-\beta_i t}$$

This model is effectively a Maxwell fluid which consists of a dampers and springs in series. We characterize this in the input by shear moduli,  $G_i$ , and decay constants,  $\beta_i$ . The viscoelastic behavior is optional and an arbitrary number of terms may be used.

The Mooney-Rivlin rubber model (model 27) is obtained by specifying  $n=1$ . In spite of the differences in formulations with Model 27, we find that the results obtained with this model are nearly identical with those of Material 27 as long as large values of Poisson's ratio are used.

**\*MAT\_OGDEN\_RUBBER**

This is also Material Type 77. This material model provides the Ogden [1984] rubber model combined optionally with linear viscoelasticity as outlined by [Christensen 1980].

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	PR	N	NV			
Type	I	F	F	I	I			

**Card 2 if N > 0, a least squares fit is computed from uniaxial data**

**Card Format**

Card 2            1            2            3            4            5            6            7            8

Variable	SGL	SW	ST	LCID1	DATA	LCID2	not used	TRAMP
Type	F	F	F	F	F	F		F

**Cards 2,3 if N = 0 define the following constants**

**Card Format**

Card 2            1            2            3            4            5            6            7            8

Variable	MU1	MU2	MU3	MU4	MU5	MU6	MU7	MU8
Type	F	F	F	F	F	F	F	F

Card 3            1            2            3            4            5            6            7            8

Variable	ALPHA1	ALPHA2	ALPHA3	ALPHA4	ALPHA5	ALPHA6	ALPHA7	ALPHA8
Type	F	F	F	F	F	F	F	F

**Card Format for Viscoelastic Constants. Up to 6 cards may be input. A keyword card (with a “\*” in column 1) terminates this input if less than 6 cards are used.**

Optional Cards            1            2            3            4            5            6            7            8

Variable	GI	BETA1						
Type	F	F						

VARIABLE

DESCRIPTION

- MID            Material identification. A unique number has to be chosen.
- RO            Mass density
- PR            Poissons ratio ( $\geq .49$  is recommended, smaller values may not work and should not be used).
- N            Order of fit to the Ogden model, (currently  $<5$ , 2 generally works okay). The constants generated during the fit are printed in the output file and can be directly input in future runs, thereby, saving the cost of performing the nonlinear fit.
- NV            Number of terms in fit. If zero, the default is 6. Currently, the maximum number is set to 6. Values less than 6, possibly 3-5 are recommended, since each term used adds significantly to the cost. Caution should be exercised when taking the results from the fit. Preferably, all generated coefficients should be positive. Negative values may lead to unstable results. Once a satisfactory fit has been achieved it is recommended that the coefficients which are written into the output file be input in future runs.

if  $N > 0$  test information from a uniaxial test are used:

- SGL            Specimen gauge length
- SW            Specimen width

VARIABLE	DESCRIPTION
ST	Specimen thickness
LCID1	Load curve ID giving the force versus actual change in the gauge length
DATA	Type of experimental data. EQ.0.0: uniaxial data EQ.1.0: biaxial data
LCID2	Load curve ID of relaxation curve If constants $\beta t$ are determined via a least squares fit. This relaxation curve is shown in Figure 20.23. This model ignores the constant stress.
TRAMP	Optional ramp time for loading. If N=0, the constants MU <sub>i</sub> and ALPHA <sub>i</sub> have to be defined:
MU <sub>i</sub>	$\mu_i$ , the <i>i</i> th shear modulus, <i>i</i> varies up to 8. See discussion below.
ALPHA <sub>i</sub>	$\alpha_i$ , the <i>i</i> th exponent, <i>i</i> varies up to 8. See discussion below.
GI	Optional shear relaxation modulus for the <i>i</i> th term
BETA <sub>i</sub>	Optional decay constant if <i>i</i> th term

**Remarks:**

Rubber is generally considered to be fully incompressible since the bulk modulus greatly exceeds the shear modulus in magnitude. To model the rubber as an unconstrained material a hydrostatic work term is included in the strain energy functional which is function of the relative volume, *J*, [Ogden, 1984]:

$$W^* = \sum_{i=1}^3 \sum_{j=1}^n \frac{\mu_j}{\alpha_j} (\lambda_i^{*\alpha_j} - 1) + \frac{1}{2} K(J - 1)^2$$

The asterisk (\*) indicates that the volumetric effects have been eliminated from the principal stretches,  $\lambda_j^*$ . The number of terms, *n*, may vary between 1 to 8 inclusive, and *K* is the bulk modulus.

Rate effects are taken into account through linear viscoelasticity by a convolution integral of the form:

$$\sigma_{ij} = \int_0^t g_{ijkl}(t - \tau) \frac{\partial \epsilon_{kl}}{\partial \tau} d\tau$$

or in terms of the second Piola-Kirchhoff stress, *S<sub>ij</sub>*, and Green's strain tensor, *E<sub>ij</sub>*,

$$S_{ij} = \int_0^t G_{ijkl}(t - \tau) \frac{\partial E_{kl}}{\partial \tau} d\tau$$

where  $g_{ijkl}(t - \tau)$  and  $G_{ijkl}(t - \tau)$  are the relaxation functions for the different stress measures. This stress is added to the stress tensor determined from the strain energy functional.

If we wish to include only simple rate effects, the relaxation function is represented by six terms from the Prony series:

$$g(t) = \alpha_0 + \sum_{m=1}^N \alpha_m e^{-\beta t}$$

given by,

$$g(t) = \sum_{i=1}^n G_i e^{-\beta_i t}$$

This model is effectively a Maxwell fluid which consists of a dampers and springs in series. We characterize this in the input by shear moduli,  $G_i$ , and decay constants,  $\beta_i$ . The viscoelastic behavior is optional and an arbitrary number of terms may be used.

The Mooney-Rivlin rubber model (model 27) is obtained by specifying  $n=1$ . In spite of the differences in formulations with Model 27, we find that the results obtained with this model are nearly identical with those of Material 27 as long as large values of Poisson's ratio are used.

**\*MAT\_SOIL\_CONCRETE**

This is Material Type 78. This model permits concrete and soil to be efficiently modelled. See the explanations below.

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	G	K	LCPV	LCYP	LCFP	LCRP
Type	I	F	F	F	F	F	F	F

Card 2            1            2            3            4            5            6            7            8

Variable	PC	OUT	B	FAIL				
Type	F	F	F	F				

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MID	Material identification. A unique number has to be chosen.
RO	Mass density
G	Shear modulus
K	Bulk modulus
LCPV	Load curve ID for pressure versus volumetric strain. The pressure versus volumetric strain curve is defined in compression only. The sign convention requires that both pressure and compressive strain be defined as positive values where the compressive strain is taken as the negative value of the natural logarithm of the relative volume.
LCYP	Load curve ID for yield versus pressure: GT.0: von Mises stress versus pressure, LT.0: Second stress invariant, $J_2$ , versus pressure. This curve must be defined.
LCFP	Load curve ID for plastic strain at which fracture begins versus pressure. This load curve ID must be defined if $B > 0.0$ .



<u>VARIABLE</u>	<u>DESCRIPTION</u>
LCRP	Load curve ID for plastic strain at which residual strength is reached versus pressure. This load curve ID must be defined if B>0.0.
PC	Pressure cutoff for tensile fracture
OUT	Output option for plastic strain in database: EQ.0: volumetric plastic strain, EQ.1: deviatoric plastic strain.
B	Residual strength factor after cracking, see Figure 20.24.
FAIL	Flag for failure: EQ.0: no failure, EQ.1: When pressure reaches failure pressure element is eroded, EQ.2: When pressure reaches failure pressure element loses it ability to carry tension.

**Remarks:**

Pressure is positive in compression. Volumetric strain is defined as the natural log of the relative volume and is *positive* in compression where the relative volume, V, is the ratio of the current volume to the initial volume. The tabulated data should be given in order of increasing compression. If the pressure drops below the cutoff value specified, it is reset to that value and the deviatoric stress state is eliminated.

If the load curve ID (LCYP) is provided as a positive number, the deviatoric, perfectly plastic, pressure dependent, yield function  $\phi$ , is given as

$$\phi = \sqrt{3J_2} - F(p) = \sigma_y - F(p)$$

where ,  $F(p)$  is a tabulated function of yield stress versus pressure, and the second invariant,  $J_2$ , is defined in terms of the deviatoric stress tensor as:

$$J_2 = \frac{1}{2} S_{ij} S_{ij}$$

assuming that . If the ID is given as negative then the yield function becomes:

$$\phi = J_2 - F(p)$$

being the deviatoric stress tensor.

If cracking is invoked by setting the residual strength factor, B, on card 2 to a value between 0.0 and 1.0, the yield stress is multiplied by a factor f which reduces with plastic strain according to a trilinear law as shown in Figure 20.24.

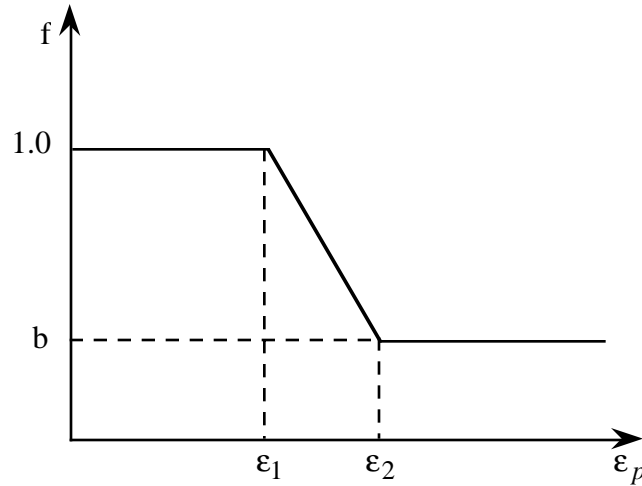


Figure 20.24. Strength reduction factor.

$b$  = residual strength factor

$\epsilon_1$  = plastic stain at which cracking begins.

$\epsilon_2$  = plastic stain at which residual strength is reached.

$\epsilon_1$  and  $\epsilon_2$  are tabulated function of pressure that are defined by load curves, see Figure 20.25. 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 20.26.

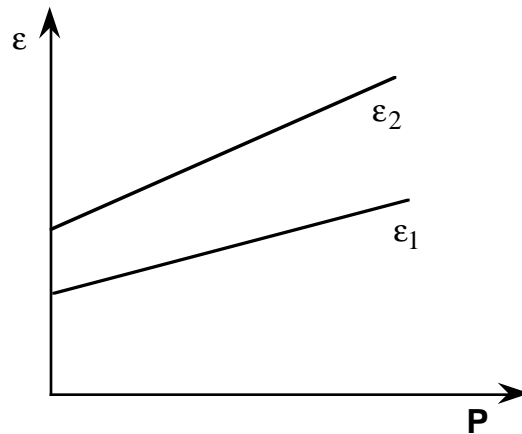


Figure 20.25. Cracking strain versus pressure.

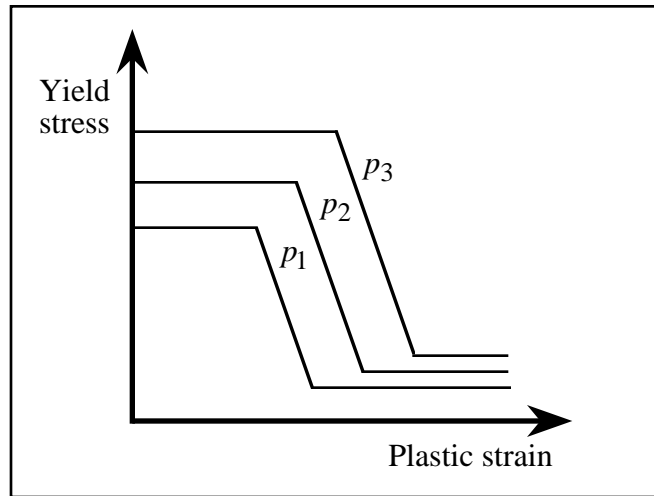


Figure 20.26.

**\*MAT\_HYSTERETIC\_SOIL**

This is Material Type 79. This model is a nested surface model with five superposed “layers” of elasto-perfectly plastic material, each with its own elastic moduli and yield values. Nested surface models give hysteric behavior, as the different “layers” yield at different stresses. See notes below.

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	K0	P0	B	A0	A1	A2
Type	I	F	F	F	F	F	F	F

Card 2            1            2            3            4            5            6            7            8

Variable	DF	RP	LCID	SFLC				
Type	F	F	F	F				

Card 3            1            2            3            4            5            6            7            8

Variable	GAM1	GAM2	GAM3	GAM4	GAM5			
Type	F	F	F	F	F			

Card 4            1            2            3            4            5            6            7            8

Variable	TAU1	TAU2	TAU3	TAU4	TAU5			
Type	F	F	F	F	F			

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density
K0	Bulk modulus at the reference pressure
P0	Cut-off/datum pressure (must be $0 \leq$ i.e. tensile). Below this pressure, stiffness and strength disappears; this is also the “zero” pressure for pressure-varying properties.
B	Exponent for pressure-sensitive moduli, b: $G = G_0(p - p_0)^b$ $K = K_0(p - p_0)^b$ b, must lie in the range $0 \leq b < 1$ . Values close to 1 are not recommended because the pressure becomes indeterminate.
A0	Yield function constant $a_0$ (Default = 1.0), see Material Type 5.
A1	Yield function constant $a_1$ (Default = 0.0), see Material Type 5.
A2	Yield function constant $a_2$ (Default = 0.0), see Material Type 5.
DF	Damping factor. Must be in the range $0 \leq df \leq 1$ : EQ.0: no damping, EQ.1: maximum damping.
RP	Reference pressure for following input data.
LCID	Load curve ID defining shear strain verses shear stress. Upto ten points may be defined in the load curve. See *DEFINE_CURVE.
SFLD	Scale factor to apply to shear stress in LCID.
GAM1	$\gamma_1$ , shear strain (ignored if LCID is non zero).
GAM2	$\gamma_2$ , shear strain (ignored if LCID is non zero).
GAM3	$\gamma_3$ , shear strain (ignored if LCID is non zero).
GAM4	$\gamma_4$ , shear strain (ignored if LCID is non zero).
GAM5	$\gamma_5$ , shear strain (ignored if LCID is non zero).

<b>VARIABLE</b>	<b>DESCRIPTION</b>
TAU1	$\tau_1$ , shear stress at $\gamma_1$ (ignored if LCID is non zero).
TAU2	$\tau_2$ , shear stress at $\gamma_2$ (ignored if LCID is non zero).
TAU3	$\tau_3$ , shear stress at $\gamma_3$ (ignored if LCID is non zero).
TAU4	$\tau_4$ , shear stress at $\gamma_4$ (ignored if LCID is non zero).
TAU5	$\tau_5$ , shear stress at $\gamma_5$ (ignored if LCID is non zero).

**Remarks:**

The constants  $a_0$ ,  $a_1$ ,  $a_2$  govern the pressure sensitivity of the yield stress. Only the ratios between these values are important - the absolute stress values are taken from the stress-strain curve.

The stress strain pairs define a shear stress versus shear strain curve. The first point on the curve is assumed by default to be (0,0) and does not need to be entered. The slope of the curve must decrease with increasing  $\gamma$ . This curve 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, SOIL\_AND\_FOAM.

The elastic moduli  $G$  and  $K$  are pressure sensitive.

$$G = G_0(p - p_0)^b$$

$$K = K_0(p - p_0)^b$$

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:

$$p = [-K_0 \ln(V)]^{1/b}$$

where  $V$  is the relative volume, i.e., the ratio between the original and current volume.

\*MAT\_RAMBERG-OSGOOD

This is Material Type 80. This model is intended as a simple model of shear behavior and can be used in seismic analysis.

Card Format

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	GAMY	TAUY	ALPHA	R	BULK	
Type	I	F	F	F	F	F	F	
Default	none	none	none	none	none	none	none	

VARIABLE

DESCRIPTION

- MID            Material identification. A unique number has to be chosen.
- RO            Mass density.
- GAMY        Reference shear strain ( $\gamma_y$ )
- TAUY        Reference shear stress ( $\tau_y$ )
- ALPHA       Stress coefficient ( $\alpha$ )
- R            Stress exponent ( $r$ )
- BULK        Elastic bulk modulus

**Remarks:**

The Ramberg-Osgood equation is an empirical constitutive relation to represent the one-dimensional elastic-plastic behavior of many materials, including soils. This model allows a simple rate independent representation of the hysteretic energy dissipation observed in soils subjected to cyclic shear deformation. For monotonic loading, the stress-strain relationship is given by:

$$\frac{\gamma}{\gamma_y} = \frac{\tau}{\tau_y} + \alpha \left| \frac{\tau}{\tau_y} \right|^r \quad \text{if } \gamma \geq 0$$

$$\frac{\gamma}{\gamma_y} = \frac{\tau}{\tau_y} - \alpha \left| \frac{\tau}{\tau_y} \right|^r \quad \text{if } \gamma < 0$$

where  $\gamma$  is the shear and  $\tau$  is the stress. The model approaches perfect plasticity as the stress exponent  $r \rightarrow \infty$ . 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

$$\frac{(\gamma - \gamma_0)}{2\gamma_y} = \frac{(\tau - \tau_0)}{2\tau_y} + \alpha \left| \frac{(\tau - \tau_0)}{2\tau_y} \right|^r \quad \text{if } \gamma \geq 0$$
$$\frac{(\gamma - \gamma_0)}{2\gamma_y} = \frac{(\tau - \tau_0)}{2\tau_y} - \alpha \left| \frac{(\tau - \tau_0)}{2\tau_y} \right|^r \quad \text{if } \gamma < 0$$

where  $\gamma_0$  and  $\tau_0$  represent the values of strain and stress at the point of load reversal. Subsequent load reversals are detected by  $(\gamma - \gamma_0)\dot{\gamma} < 0$ .

The Ramberg-Osgood equations are inherently one-dimensional and are assumed to apply to shear components. To generalize this theory to the multidimensional case, it is assumed that each component of the deviatoric stress and deviatoric tensorial strain is independently related by the one-dimensional stress-strain equations. A projection is used to map the result back into deviatoric stress space if required. The volumetric behavior is elastic, and, therefore, the pressure  $p$  is found by

$$p = -K\varepsilon_v$$

where  $\varepsilon_v$  is the volumetric strain.



**\*MAT\_PLASTICITY\_WITH\_DAMAGE**

This is Material Type 81. An elasto-visco-plastic material with an arbitrary stress versus strain curve and arbitrary strain rate dependency can be defined. Damage is considered before rupture occurs. Also, failure based on a plastic strain or a minimum time step size can be defined.

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	E	PR	SIGY	ETAN	EPPF	TDEL
Type	I	F	F	F	F	F	F	F
Default	none	none	none	none	none	0.0	10.E+20	10.E+20

Card 2

Variable	C	P	LCSS	LCSR	EPPFR	VP	LCDM	
Type	F	F	F	F	F	F	F	
Default	0	0	0	0	0	0	0	

Card 3

Variable	EPS1	EPS2	EPS3	EPS4	EPS5	EPS6	EPS7	EPS8
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

Card 4

Variable	ES1	ES2	ES3	ES4	ES5	ES6	ES7	ES8
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

**VARIABLE****DESCRIPTION**

MID	Material identification. A unique number has to be chosen.
RO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.
SIGY	Yield stress.
ETAN	Tangent modulus, ignored if (LCSS.GT.0) is defined.
EPPF	Plastic strain, $f_s$ , at which material softening begins (logarithmic).
TDEL	Minimum time step size for automatic element deletion.
C	Strain rate parameter, C, see formula below.
P	Strain rate parameter, P, see formula below.
LCSS	Load curve ID. Load curve ID defining effective stress versus effective plastic strain. If defined EPS1-EPS8 and ES1-ES8 are ignored.
LCSR	Load curve ID defining strain rate scaling effect on yield stress.
EPPFR	Plastic strain at which material ruptures (logarithmic).
VP	Formulation for rate effects: EQ.0.0: Scale yield stress (default), EQ.1.0: Viscoplastic formulation.
LCDM	Load curve ID defining nonlinear damage curve.
EPS1-EPS8	Effective plastic strain values (optional if SIGY is defined). At least 2 points should be defined.

VARIABLE	DESCRIPTION
ES1-ES8	Corresponding yield stress values to EPS1 - EPS8.

**Remarks:**

The stress strain behavior may be treated by a bilinear stress strain curve by defining the tangent modulus, ETAN. Alternately, a curve similar to that shown in Figure 20.4 is expected to be defined by (EPS1,ES1) - (EPS8,ES8); however, an effective stress versus effective plastic strain curve (LCSS) may be input instead if eight points are insufficient. The cost is roughly the same for either approach. The most general approach is to use the table definition (LCSS) discussed below.

Two options to account for strain rate effects are possible.

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

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

where  $\dot{\epsilon}$  is the strain rate.  $\dot{\epsilon} = \sqrt{\dot{\epsilon}_{ij} \dot{\epsilon}_{ij}}$ .

If the viscoplastic option is active, VP=1.0, and if SIGY is > 0 then the dynamic yield stress is computed from the sum of the static stress,  $\sigma_y^s(\epsilon_{eff}^p)$ , which is typically given by a load curve ID, and the initial yield stress, SIGY, multiplied by the Cowper-Symonds rate term as follows:

$$\sigma_y(\epsilon_{eff}^p, \dot{\epsilon}_{eff}^p) = \sigma_y^s(\epsilon_{eff}^p) + SIGY \cdot \left( \frac{\dot{\epsilon}_{eff}^p}{C} \right)^{1/p}$$

where the plastic strain rate is used. With this latter approach similar results can be obtained between this model and material model: \*MAT\_ANISOTROPIC\_VISCOPLASTIC. If SIGY=0, the following equation is used instead where the static stress,  $\sigma_y^s(\epsilon_{eff}^p)$ , must be defined by a load curve:

$$\sigma_y(\epsilon_{eff}^p, \dot{\epsilon}_{eff}^p) = \sigma_y^s(\epsilon_{eff}^p) \left[ 1 + \left( \frac{\dot{\epsilon}_{eff}^p}{C} \right)^{1/p} \right]$$

This latter equation is always used if the viscoplastic option is off.

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

The constitutive properties for the damaged material are obtained from the undamaged material properties. The amount of damage evolved is represented by the constant,  $\omega$ , which varies from zero if no damage has occurred to unity for complete rupture. For uniaxial loading, the nominal stress in the damaged material is given by

$$\sigma_{nominal} = \frac{P}{A}$$

where P is the applied load and A is the surface area. The true stress is given by:

$$\sigma_{true} = \frac{P}{A - A_{loss}}$$

where  $A_{loss}$  is the void area. The damage variable can then be defined:

$$\omega = \frac{A_{loss}}{A} \quad 0 \leq \omega \leq 1$$

In this model damage is defined in terms of plastic strain after the failure strain is exceeded:

$$\omega = \frac{\epsilon_{eff}^p - \epsilon_{failure}^p}{\epsilon_{rupture}^p - \epsilon_{failure}^p} \quad \text{if} \quad \epsilon_{failure}^p \leq \epsilon_{eff}^p \leq \epsilon_{rupture}^p$$

After exceeding the failure strain softening begins and continues until the rupture strain is reached.

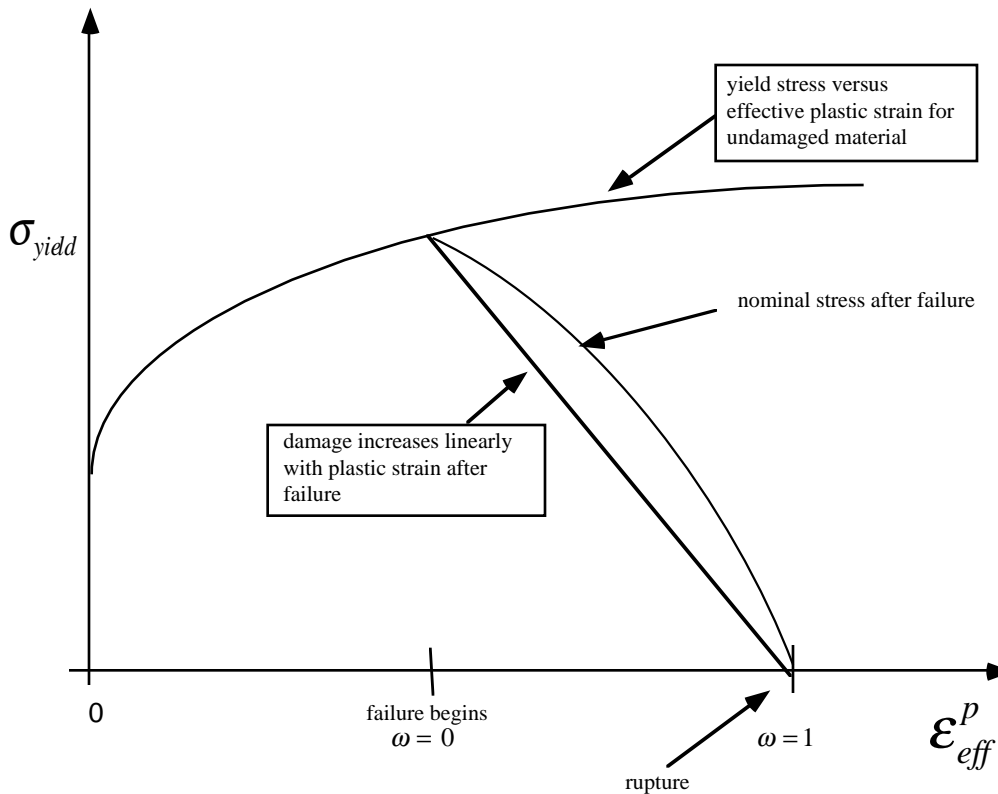
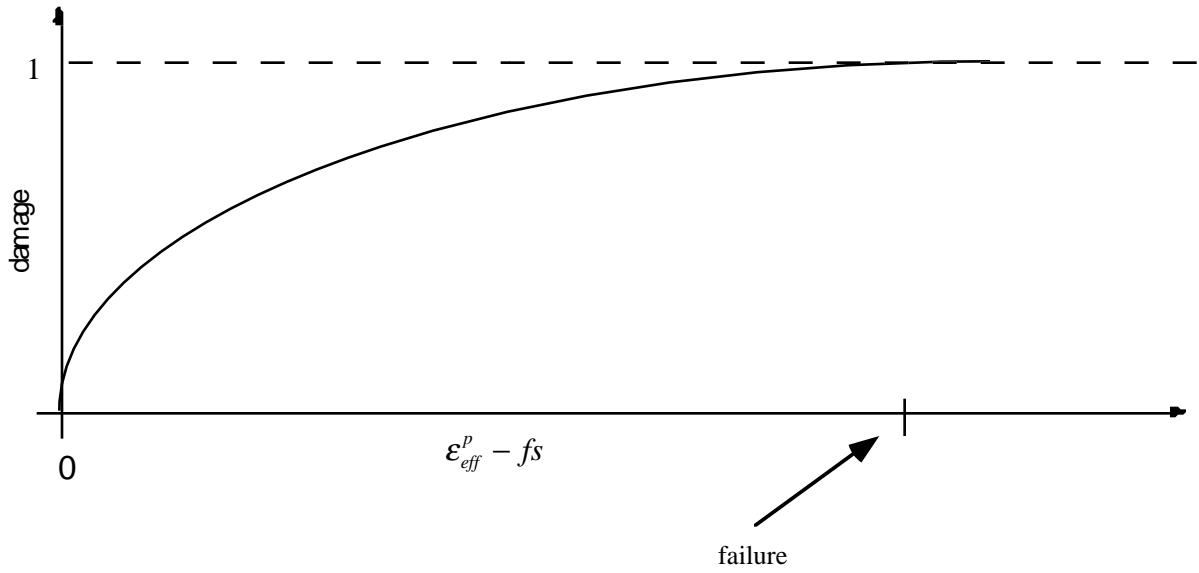


Figure 20.27. Stress strain behavior when damage is included.



**Figure 20.28.** A nonlinear damage curve is optional. Note that the origin of the curve is at (0,0). It is permissible to input the failure strain,  $fs$ , as zero for this option. The nonlinear damage curve is useful for controlling the softening behavior after the failure strain is reached.

**\*MAT\_FU\_CHANG\_FOAM**

This is Material Type 83. Rate effects can be modelled in low and medium density foams, see Figure 20.29. Hysteretic unloading behavior in this model is a function of the rate sensitivity with the most rate sensitive foams providing the largest hysteresis 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.

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	E	ED	TC	FAIL	DAMP	TBID
Type	I	F	F	F	F	F	F	F
Default	---	---	---	---	1.E+20			
Remarks	---	---	---	---	---			---

Card 2

Variable	BVFLAG	SFLAG	RFLAG	TFLAG	PVID			
Type	F	F	F	F	F			
Default	1.0	0.0	0.0	0.0	0.0			
Remarks	1	2	3		4			

Card 3

Variable	D0	N0	N1	N2	N3	C0	C1	C2
Type	F	F	F	F	F	F	F	F
Default	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0

Card 4

Variable	C3	C4	C5	AIJ	SIJ	MINR	MAXR	
Type	F	F	F	F	F	F	F	
Default	0.0	0.0	0.0	0.0	0.0	0.0	0.0	

**VARIABLE**

**DESCRIPTION**

MID	Material identification. A unique number has to be chosen.
RO	Mass density
E	Young's modulus
ED	Optional Young's relaxation modulus, $E_d$ , for rate effects. See comments below. EQ.0.0: Maximum slope in stress vs. strain curve is used. When the maximum slope is taken for the contact, the time step size for this material is reduced for stability. In some cases $\Delta t$ may be significantly smaller, and defining a reasonable stiffness is recommended.
TC	Tension cut-off stress
FAIL	Failure option after cutoff stress is reached: EQ.0.0: tensile stress remains at cut-off value, EQ.1.0: tensile stress is reset to zero.
DAMP	Viscous coefficient ( $.05 <$ recommended value $<.50$ ) to model damping effects.

---

<b>VARIABLE</b>	<b>DESCRIPTION</b>
TBID	Table ID, see *DEFINE_TABLE, for nominal stress strain data as a function of strain rate. If the table ID is provided, cards 3 and 4 may be left blank and the fit will be done internally.
BVFLAG	Bulk viscosity activation flag, see remark below: EQ.0.0: no bulk viscosity (recommended), EQ.1.0: bulk viscosity active.
D0	material constant, see equations below.
N0	material constant, see equations below.
N1	material constant, see equations below.
N2	material constant, see equations below.
N3	material constant, see equations below.
C0	material constant, see equations below.
C1	material constant, see equations below.
C2	material constant, see equations below.
C3	material constant, see equations below.
C4	material constant, see equations below.
C5	material constant, see equations below.
<i>A<sub>ij</sub></i>	material constant, see equations below.
<i>S<sub>ij</sub></i>	material constant, see equations below.
MINR	Ratemin, minimum strain rate of interest.
MAXR	Ratemax, maximum strain rate of interest.
SFLAG	Strain rate flag (see remark 2 below): EQ.0.0: true constant strain rate, EQ.1.0: engineering strain rate.
RFLAG	Strain rate evaluation flag: EQ.0.0: first principal direction, EQ.1.0: principal strain rates for each principal direction, EQ.2.0: volumetric strain rate.
TFLAG	Tensile stress evaluation: EQ.0.0: linear in tension. EQ.1.0: input via load curves with the tensile response corresponds to negative values of stress and strain.

---



VARIABLE	DESCRIPTION
PVID	Optional load curve ID defining pressure versus volumetric strain.

**Remarks:**

The strain is divided into two parts: a linear part and a non-linear part of the strain

$$E(t) = E^L(t) + E^N(t)$$

and the strain rate become

$$\dot{E}(t) = \dot{E}^L(t) + \dot{E}^N(t)$$

$\dot{E}^N$  is an expression for the past history of  $E^N$ . A postulated constitutive equation may be written as:

$$\sigma(t) = \int_{\tau=0}^{\infty} [E_t^N(\tau), S(t)] d\tau$$

where  $S(t)$  is the state variable and  $\int_{\tau=0}^{\infty}$  is a functional of all values of  $\tau$  in  $T_t : 0 \leq \tau \leq \infty$  and

$$E_t^N(\tau) = E^N(t - \tau)$$

where  $\tau$  is the history parameter:

$$E_t^N(\tau = \infty) \Leftrightarrow \text{the virgin material}$$

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:

$$\sigma(t) = \sigma^*(E^N(t), \dot{E}^N(t), S(t))$$

where we have replaced  $\frac{\partial E_t^N}{\partial t}$  by  $\dot{E}^N$ , and  $\sigma^*$  is a function of its arguments.

For a special case,

$$\sigma(t) = \sigma^*(\dot{E}^N(t), S(t))$$

we may write

$$\dot{E}_t^N = f(S(t), s(t))$$

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:

$$\dot{E}^N = \frac{\sigma}{\|\sigma\|} D_0 \exp \left[ -c_0 \left( \frac{\text{tr}(\sigma S)}{\|\sigma\|^2} \right)^{2n_0} \right]$$

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.

$$\dot{S}_{ij} = \left[ c_1 (a_{ij} R - c_2 S_{ij}) P + c_3 W^{n_1} \left( \|\dot{E}^N\| \right)^{n_2} I_{ij} \right] R$$

$$R = 1 + c_4 \left( \frac{\|\dot{E}^N\|}{c_5} - 1 \right)^{n_3}$$

$$P = \text{tr}(\sigma \dot{E}^N)$$

$$W = \int \text{tr}(\sigma dE)$$

where  $c_1$ ,  $c_2$ ,  $c_3$ ,  $c_4$ ,  $c_5$ ,  $n_1$ ,  $n_2$ ,  $n_3$ , and  $a_{ij}$  are material constants and:

$$\|\sigma\| = \left( \sigma_{ij} \sigma_{ij} \right)^{\frac{1}{2}}$$

$$\|\dot{E}\| = \left( \dot{E}_{ij} \dot{E}_{ij} \right)^{\frac{1}{2}}$$

$$\|\dot{E}^N\| = \left( \dot{E}_{ij}^N \dot{E}_{ij}^N \right)^{\frac{1}{2}}$$

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.

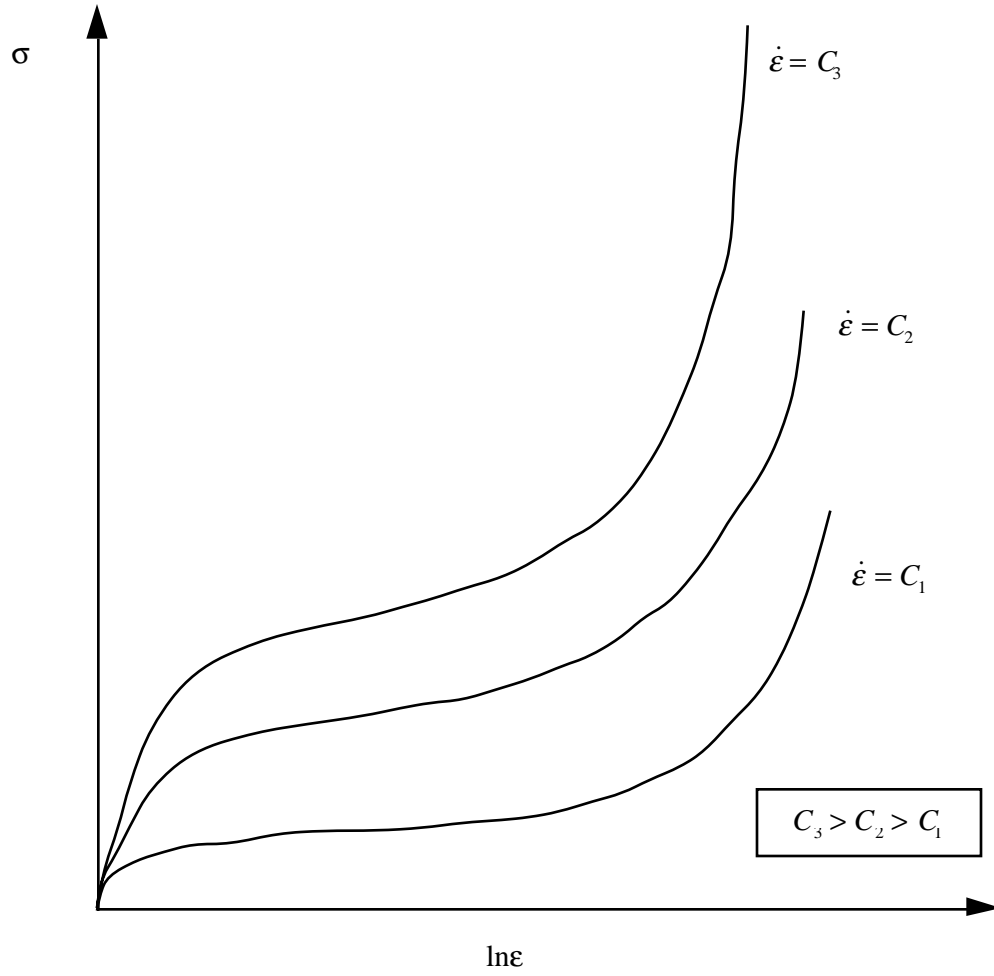


Figure 20.29. Rate effects in Fu Chang's foam model.

**Additional Remarks:**

1. The bulk viscosity, which generates a rate dependent pressure, may cause an unexpected volumetric response and, consequently, it is optional with this model.
2. Dynamic compression tests at the strain rates of interest in vehicle crash are usually performed with a drop tower. In this test the loading velocity is nearly constant but the true strain rate, which depends on the instantaneous specimen thickness, is not. Therefore, the engineering strain rate input is optional so that the stress strain curves obtained at constant velocity loading can be used directly.
3. To further improve the response under multiaxial loading, the strain rate parameter can either be based on the principal strain rates or the volumetric strain rate.
4. Correlation under triaxial loading is achieved by directly inputting the results of hydrostatic testing in addition to the uniaxial data. Without this additional information which is fully optional, triaxial response tends to be underestimated.

\*MAT\_ORTHOTROPIC\_VISCOELASTIC

This is Material Type 86. It allows the definition of an orthotropic material with a viscoelastic part. This model applies to shell elements.

Card Format

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	EA	EB	EC	VF	K	
Type	I	F	F	F	F	F	F	

Card 2            1            2            3            4            5            6            7            8

Variable	G0	GINF	BETA	PRBA	PRCA	PRCB		
Type	F	F	F	F	F	F		

Card 3

Variable	GAB	GBC	GCA	AOPT	MANGLE			
Type	F	F	F	F	F			

Card 4

Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card 5

Variable	V1	V2	V3	D1	D2	D3		
Type	F	F	F	F	F	F		

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MID	Material identification. A unique number has to be chosen.
RO	Mass density
EA	Young's Modulus $E_a$
EB	Young's Modulus $E_b$
EC	Young's Modulus $E_c$
VF	Volume fraction of viscoelastic material
K	Elastic bulk modulus
G0	$G_0$ , short-time shear modulus
GINF	$G_\infty$ , long-time shear modulus
BETA	$\beta$ , decay constant
PRBA	Poisson's ratio, $\nu_{ba}$
PRCA	Poisson's ratio, $\nu_{ca}$
PRCB	Poisson's ratio, $\nu_{cb}$
GAB	Shear modulus, $G_{ab}$
GBC	Shear modulus, $G_{bc}$
GCA	Shear modulus, $G_{ca}$

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
AOPT	Material axes option, see Figure 20.1: EQ. 0.0: locally orthotropic with material axes determined by element nodes as shown in Figure 20.1. Nodes 1, 2, and 4 of an element are identical to the Nodes used for the definition of a coordinate system as by *DEFINE_COORDINATE_NODES. EQ. 1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center, this is the a-direction. EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR. EQ. 3.0: applicable to shell elements only. This option determines locally orthotropic material axes by offsetting the material axes by an angle to be specified from a line in the plane of the shell determined by taking the cross product of the vector <b>v</b> defined below with the shell normal vector.
MANGLE	Material angle, may be overwritten on the element card AOPT = 3.
XP YP ZP	Define coordinates of point p for AOPT = 1 and 4.
A1 A2 A3	Define components of vector a for AOPT = 2.
V1 V2 V3	Define components of vector v for AOPT = 3 and 4.
D1 D2 D3	Define components of vector d for AOPT = 2.

**Remarks:**

For the orthotropic definition it is referred to Material Type 2 and 21.

**\*MAT\_CELLULAR\_RUBBER**

This is Material Type 87. This material model provides a cellular rubber model with confined air pressure combined with linear viscoelasticity as outlined by Christensen [1980]. See Figure 20.30.

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	PR	N				
Type	I	F	F	I				

**Card 2 if  $N > 0$ , a least squares fit is computed from uniaxial data**

**Card Format**

Card 2            1            2            3            4            5            6            7            8

Variable	SGL	SW	ST	LCID				
Type	F	F	F	F				

**Card 2 if  $N = 0$ , define the following constants**

**Card Format**

Card 2            1            2            3            4            5            6            7            8

Variable	C10	C01	C11	C20	C02			
Type	F	F	F	F	F			



Card Format

Card 3            1            2            3            4            5            6            7            8

Variable	P0	PHI	IVS	G	BETA			
Type	F	F	F	F	F			

VARIABLE

DESCRIPTION

- MID            Material identification. A unique number has to be chosen.
- RO            Mass density
- PR            Poisson's ratio, typical values are between .0 to .2. Due to the large compressibility of air, large values of Poisson's ratio generates physically meaningless results.
- N            Order of fit (currently < 3). If  $n > 0$  then a least square fit is computed with uniaxial data. The parameters given on card 2 should be specified. Also see \*MAT\_MOONEY\_RIVLIN\_RUBBER (material model 27). A Poisson's ratio of .5 is assumed for the void free rubber during the fit. The Poisson's ratio defined on Card 1 is for the cellular rubber. A void fraction formulation is used.

Define, if  $N > 0$ :

- SGL            Specimen gauge length  $l_0$
- SW            Specimen width
- ST            Specimen thickness
- LCID            Load curve ID giving the force versus actual change  $\Delta L$  in the gauge length.

Define, if  $N = 0$ :

- C10            Coefficient,  $C_{10}$
- C01            Coefficient,  $C_{01}$
- C11            Coefficient,  $C_{11}$
- C20            Coefficient,  $C_{20}$
- C02            Coefficient,  $C_{02}$

VARIABLE	DESCRIPTION
P0	Initial air pressure, $P_0$
PHI	Ratio of cellular rubber to rubber density, $\Phi$
IVS	Initial volumetric strain, $\gamma_0$
G	Optional shear relaxation modulus, $G$ , for rate effects (viscosity)
BETA	Optional decay constant, $\beta_1$

**Remarks:**

Rubber is generally considered to be fully incompressible since the bulk modulus greatly exceeds the shear modulus in magnitude. To model the rubber as an unconstrained material a hydrostatic work term,  $W_H(J)$ , is included in the strain energy functional which is function of the relative volume,  $J$ , [Ogden, 1984]:

$$W(J_1, J_2, J) = \sum_{p,q=0}^n C_{pq} (J_1 - 3)^p (J_2 - 3)^q + W_H(J)$$

$$J_1 = I_1 I_3^{-1/3} \qquad \sigma_y = \left[ 3 (a_0 + a_1 p + a_2 p^2) \right]^{1/2}$$

$$J_2 = I_2 I_3^{-2/3}$$

In order to prevent volumetric work from contributing to the hydrostatic work the first and second invariants are modified as shown. This procedure is described in more detail by Sussman and Bathe [1987].

The effects of confined air pressure in its overall response characteristics is included by augmenting the stress state within the element by the air pressure.

$$\sigma_{ij} = \sigma_{ij}^{sk} - \delta_{ij} \sigma^{air}$$

where  $\sigma_{ij}^{sk}$  is the bulk skeletal stress and  $\sigma^{air}$  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  $\gamma$  defines the volumetric strain

$$\gamma = V - 1 + \gamma_0$$

where  $V$  is the relative volume of the voids and  $\gamma_0$  is the initial volumetric strain which is typically zero. The rubber skeletal material is assumed to be incompressible.

Rate effects are taken into account through linear viscoelasticity by a convolution integral of the form:

$$\sigma_{ij} = \int_0^t g_{ijkl}(t - \tau) \frac{\partial \epsilon_{kl}}{\partial \tau} d\tau$$

or in terms of the second Piola-Kirchhoff stress,  $S_{ij}$ , and Green's strain tensor,  $E_{ij}$ ,

$$S_{ij} = \int_0^t G_{ijkl}(t - \tau) \frac{\partial E_{kl}}{\partial \tau} d\tau$$

where  $g_{ijkl}(t - \tau)$  and  $G_{ijkl}(t - \tau)$  are the relaxation functions for the different stress measures. This stress is added to the stress tensor determined from the strain energy functional.

Since we wish to include only simple rate effects, the relaxation function is represented by one term from the Prony series:

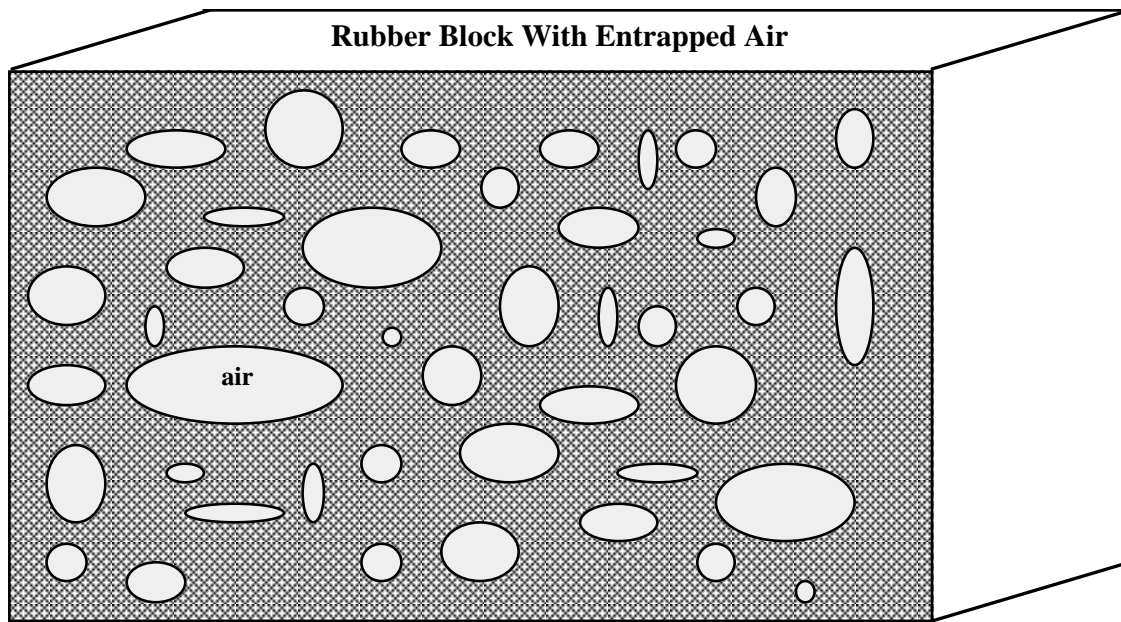
$$g(t) = \alpha_0 + \sum_{m=1}^N \alpha_m e^{-\beta t}$$

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,  $\beta_1$ .

The Mooney-Rivlin rubber model (model 27) is obtained by specifying  $n=1$  without air pressure and viscosity. In spite of the differences in formulations with Model 27, we find that the results obtained with this model are nearly identical with those of material type 27 as long as large values of Poisson's ratio are used.



**Figure 20.30.** Cellular rubber with entrapped air. By setting the initial air pressure to zero, an open cell, cellular rubber can be simulated.

\*MAT\_MTS

This is Material Type 88. The MTS model is due to 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 value for the mechanical threshold stress, the flow stress is determined using either a thermal-activation-controlled or a drag-controlled kinetics relationship.. An equation-of-state is required for solid elements and a bulk modulus must be defined below for shell elements.

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	SIGA	SIGI	SIGS	SIG0	BULK	
Type	I	F	F	F	F	F	F	

Card 2            1            2            3            4            5            6            7            8

Variable	HF0	HF1	HF2	SIGS0	EDOTS0	BURG	CAPA	BOLTZ
Type	F	F	F	F	F	F	F	F

Card 3            1            2            3            4            5            6            7            8

Variable	SM0	SM1	SM2	EDOT0	GO	PINV	QINV	EDOTI
Type	F	F	F	F	F	F	F	F

Card 4            1            2            3            4            5            6            7            8

Variable	G0I	PINVI	QINVI	EDOTS	G0S	PINVS	QINVS	
Type	F	F	F	F	F	F	F	

Card 5            1            2            3            4            5            6            7            8

Variable	RHOCPR	TEMPRF	ALPHA	EPS0				
Type	F	F						

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MID	Material identification. A unique number has to be chosen.
RO	Mass density
SIGA	$\hat{\sigma}_a$ , dislocation interactions with long-range barriers (force/area).
SIGI	$\hat{\sigma}_i$ , dislocation interactions with interstitial atoms (force/area).
SIGS	$\hat{\sigma}_s$ , dislocation interactions with solute atoms (force/area).
SIG0	$\hat{\sigma}_0$ , initial value of $\hat{\sigma}$ at zero plastic strain (force/area) NOT USED.
HF0	$a_0$ , dislocation generation material constant (force/area).
HF1	$a_1$ , dislocation generation material constant (force/area).
HF2	$a_2$ , dislocation generation material constant (force/area).
SIGS0	$\hat{\sigma}_{eso}$ , saturation threshold stress at 0° K (force/area).
BULK	Bulk modulus defined for shell elements only. Do not input for solid elements.

VARIABLE	DESCRIPTION
EDOTS0	$\dot{\epsilon}_{eso}$ , reference strain-rate (time <sup>-1</sup> ).
BURG	Magnitude of Burgers vector (interatomic slip distance), (distance)
CAPA	Material constant, A.
BOLTZ	Boltzmann's constant ,k (energy/degree).
SM0	$G_0$ , shear modulus at zero degrees Kelvin (force/area).
SM1	$b_1$ , shear modulus constant (force/area).
SM2	$b_2$ , shear modulus constant (degree).
EDOT0	$\dot{\epsilon}_o$ , reference strain-rate (time <sup>-1</sup> ).
G0	$g_0$ , normalized activation energy for a .dislocation/dislocation interaction.
PINV	$\frac{1}{p}$ , material constant.
QINV	$\frac{1}{q}$ , material constant.
EDOTI	$\dot{\epsilon}_{o,i}$ , reference strain-rate (time <sup>-1</sup> ).
G0I	$g_{0,i}$ , normalized activation energy for a dislocation/interstitial interaction.
PINVI	$\frac{1}{p_i}$ , material constant.
QINVI	$\frac{1}{q_i}$ , material constant.
EDOTS	$\dot{\epsilon}_{o,s}$ , reference strain-rate (time <sup>-1</sup> ).
G0S	$g_{0,s}$ normalized activation energy for a dislocation/solute interaction.
PINVS	$\frac{1}{p_s}$ , material constant.
QINVS	$\frac{1}{q_s}$ , material constant.

VARIABLE	DESCRIPTION
RHOCPR	$\rho c_p$ , product of density and specific heat.
TEMPRF	$T_{ref}$ , initial element temperature in degrees K.
ALPHA	$\alpha$ , material constant (typical value is between 0 and 2).
EPS0	$\epsilon_o$ , factor to normalize strain rate in the calculation of $\Theta_o$ . (Use 1., $10^{-3}$ , or $10^{-6}$ for time units of seconds, milliseconds, or microseconds, respectively.)

**Remarks:**

The flow stress  $\sigma$  is given by:

$$\sigma = \hat{\sigma}_a + \frac{G}{G_0} [s_{th} \hat{\sigma} + s_{th,i} \hat{\sigma}_i + s_{th,s} \hat{\sigma}_s]$$

The first product in the equation for  $\sigma$  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  $\dot{\epsilon}^p$ . The evolution equation for  $\hat{\sigma}$  is a differential hardening law representing dislocation-dislocation interactions:

$$\frac{\partial \hat{\sigma}}{\partial \epsilon^p} \equiv \Theta_o \left[ 1 - \frac{\tanh\left(\alpha \frac{\hat{\sigma}}{\hat{\sigma}_{es}}\right)}{\tanh(\alpha)} \right]$$

The term,  $\frac{\partial \hat{\sigma}}{\partial \epsilon^p}$ , represents the hardening due to dislocation generation and the stress ratio,  $\frac{\hat{\sigma}}{\hat{\sigma}_{es}}$ , represents softening due to dislocation recovery. The threshold stress at zero strain-hardening  $\hat{\sigma}_{es}$  is called the saturation threshold stress. Relationships for  $\Theta_o$ ,  $\hat{\sigma}_{es}$  are:

$$\Theta_o = a_o + a_1 \ln\left(\frac{\dot{\epsilon}^p}{\epsilon_0}\right) + a_2 \sqrt{\frac{\dot{\epsilon}^p}{\epsilon_0}}$$

which contains the material constants,  $a_o$ ,  $a_1$ , and  $a_2$ . The constant,  $\hat{\sigma}_{es}$ , is given as:

$$\hat{\sigma}_{es} = \hat{\sigma}_{eso} \left( \frac{\dot{\epsilon}^p}{\dot{\epsilon}_{eso}} \right)^{kT/Gb^3A}$$



which contains the input constants:  $\hat{\sigma}_{\epsilon_{so}}$ ,  $\dot{\epsilon}_{\epsilon_{so}}$ ,  $b$ ,  $A$ , 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:

$$s_{th} = \left[ 1 - \left( \frac{kT \ln \left( \frac{\dot{\epsilon}_0}{\dot{\epsilon}^p} \right)}{Gb^3 g_0} \right)^{\frac{1}{q}} \right]^{\frac{1}{p}}$$

The absolute temperature is given as:

$$T = T_{ref} + \rho c_p E$$

where  $E$  is the internal energy density per unit initial volume.

**\*MAT\_PLASTICITY\_POLYMER**

This is Material Type 89. An elasto-plastic material with an arbitrary stress versus strain curve and arbitrary strain rate dependency can be defined. It is intended for applications where the elastic and plastic sections of the response are not so clearly distinguishable as they are for metals. Rate dependency of failure strain is included. Many polymers show a more brittle response at high rates of strain. The material model is currently available only for shell elements.

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	E	PR				
Type	I	F	F	F				
Default	none	none	none	none				

Card 2

Variable	C	P	LCSS	LCSR				
Type	F	F	F	F				
Default	0	0	0	0				

Card 3

Variable	EFTX	DAMP	RATEFAC	LCFAIL				
Type	F	F	F	F				
Default	0	0	0	0				

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.
C	Strain rate parameter, C, see formula below.
P	Strain rate parameter, P, see formula below.
LCSS	Load curve ID defining effective stress versus total effective strain. The stress versus effective plastic strain curve for the lowest value of strain rate is used if the strain rate falls below the minimum value. Likewise, the stress versus effective plastic strain curve for the highest value of strain rate is used if the strain rate exceeds the maximum value. The strain rate parameters: C and P;
LCSR	Load curve ID defining strain rate scaling effect on yield stress.
EFTX	Failure flag. EQ.0.0: failure determined by maximum tensile strain (default), EQ.1.0: failure determined only by tensile strain in local x direction, EQ.2.0: failure determined only by tensile strain in local y direction.
DAMP	Stiffness-proportional damping ratio. Typical values are 1e-3 or 1e-4. If set too high instabilities can result.
RATEFAC	Filtering factor for strain rate effects. Must be between 0 (no filtering) and 1 (infinite filtering) The filter is a simple low pass filter to remove high frequency oscillation from the strain rates before they are used in rate effect calculations. The cut off frequency of the filter is $[(1 - RATEFAC) / \text{timestep}]$ rad/sec.
LCFAIL	Load curve ID giving variation of failure strain with strain rate. The points on the x-axis should be natural log of strain rate, the y-axis should be the true strain to failure. Typically this is measured by uniaxial tensile test, and the strain values converted to true strain.

**Remarks:**

1. Unlike other LS-DYNA material models, both the input stress-strain curve and the strain to failure are defined as total true strain, not plastic strain. The input can be defined from uniaxial tensile tests; nominal stress and nominal strain from the tests must be converted to true stress and true strain. The elastic component of strain must not be subtracted out.
2. The stress-strain curve is permitted to have sections steeper (i.e. stiffer) than the elastic modulus. When these are encountered the elastic modulus is increased to prevent spurious energy generation.

3. Sixty-four bit precision is recommended when using this material model, especially if the strains become high.
4. Invariant shell numbering is recommended when using this material model. See \*CONTROL\_ACCURACY.

\*MAT\_ACOUSTIC

This is Material Type 90. This model is appropriate for tracking low pressure stress waves in an acoustic media such as air or water and can be used only with the acoustic pressure element formulation. The acoustic pressure element requires only one unknown per node. This element is very cost effective. Optionally, cavitation can be allowed.

Card Format

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	C	BETA	CF	ATMOS	GRAV	
Type	I	F	F	F	F	F	F	

Card 2            1            2            3            4            5            6            7            8

Variable	XP	YP	ZP	XN	YN	ZN		
Type	F	F	F	F	F	F		

VARIABLE

DESCRIPTION

MID	Material identification. A unique number has to be chosen.
RO	Mass density
C	Sound speed
BETA	Damping factor. Recommend values are between 0.1 and 1.0.
CF	Cavitation flag: EQ.0.0: off, EQ.1.0: on.
ATMOS	Atmospheric pressure (optional)
GRAV	Gravitational acceleration constant (optional)
XP	x-coordinate of free surface point

<b>VARIABLE</b>	<b>DESCRIPTION</b>
XP	y-coordinate of free surface point
YP	z-coordinate of free surface point
XN	x-direction cosine of free surface normal vector
YN	y-direction cosine of free surface normal vector
ZN	z-direction cosine of free surface normal vector

**\*MAT\_BRITTLE\_DAMAGE**

This is Material Type 96.

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	E	PR	TLIMIT	SLIMIT	FTOUGH	SRETEN
Type	I	F	F	F	F	F	F	F

Card 2            1            2            3            4            5            6            7            8

Variable	VISC	FRA_RF	E_RF	YS_RF	EH_RF	FS_RF	SIGY	
Type	F	F	F	F	F	F	F	

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.
TLIMIT	Tensile limit.
SLIMIT	Shear limit.
FTOUGH	Fracture toughness.
SRETEN	Shear retention.
VISC	Viscosity.
FRA_RF	Fraction of reinforcement in section.
E_RF	Young's modulus of reinforcement.

<b>VARIABLE</b>	<b>DESCRIPTION</b>
YS_RF	Yield stress of reinforcement.
EH_RF	Hardening modulus of reinforcement.
FS_RF	Failure strain (true) of reinforcement.
SIGY	Compressive yield stress. EQ.0: no compressive yield

**Remarks:**

A full description of the tensile and shear damage parts of this material model is given in Govindjee, Kay and Simo[1994,1995]. It is an anisotropic brittle damage model designed primarily for concrete though it can be applied to a wide variety of brittle materials. It admits progressive degradation of tensile and shear strengths across smeared cracks that are initiated under tensile loadings. Compressive failure is governed by a simplistic J2 flow correction that can be disabled if not desired. Damage is handled by treating the rank 4 elastic stiffness tensor as an evolving internal variable for the material. Softening induced mesh dependencies are handled by a characteristic length method (Oliver [1989]).

Description of properties:

1.  $E$  is the Young's modulus of the undamaged material also known as the virgin modulus.
2.  $\nu$  is the Poisson's ratio of the undamaged material also known as the virgin Poisson's ratio.
3.  $f_n$  is the initial principal tensile strength (stress) of the material. Once this stress has been reached at a point in the body a smeared crack is initiated there with a normal that is co-linear with the 1st principal direction. Once initiated, the crack is fixed at that location, though it will convect with the motion of the body. As the loading progresses the allowed tensile traction normal to the crack plane is progressively degraded to a small machine dependent constant.

The degradation is implemented by reducing the material's modulus normal to the smeared crack plane according to a maximum dissipation law that incorporates exponential softening. The restriction on the normal tractions is given by

$$\phi_t = (\mathbf{n} \otimes \mathbf{n}) : \boldsymbol{\sigma} - f_n + (1 - \varepsilon) f_n (1 - \exp[-H\alpha]) \leq 0$$

where  $\mathbf{n}$  is the smeared crack normal,  $\varepsilon$  is the small constant,  $H$  is the softening modulus, and  $\alpha$  is an internal variable.  $H$  is set automatically by the program; see  $g_c$  below.  $\alpha$  measures the crack field intensity and is output in the equivalent plastic strain field,  $\bar{\varepsilon}^p$ , in a normalized fashion.

The evolution of alpha is governed by a maximum dissipation argument. When the normalized value reaches unity it means that the material's strength has been reduced to 2% of its original value in the normal and parallel directions to the smeared crack. Note that for plotting purposes it is never output greater than 5.



4.  $f_s$  is the initial shear traction that may be transmitted across a smeared crack plane. The shear traction is limited to be less than or equal to  $f_s(1 - \beta)(1 - \exp[-H\alpha])$ , through the use of two orthogonal shear damage surfaces. Note that the shear degradation is coupled to the tensile degradation through the internal variable alpha which measures the intensity of the crack field.  $\beta$  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.  $\beta$  is the shear retention factor. As the damage progresses the shear tractions allowed across the smeared crack plane asymptote to the product  $\beta f_s$ .
7.  $\eta$  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.  $\sigma_y$  is a uniaxial compressive yield stress. A check on compressive stresses is made using the J2 yield function  $\mathbf{s}:\mathbf{s} - \sqrt{\frac{2}{3}}\sigma_y \leq 0$ , 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  $E=3.15 \times 10^6$  psi,  $f_n=450$  psi,  $f_s=2100$  psi,  $\nu = 0.2$ ,  $g_c = 0.8$  lbs/in,  $\beta = 0.03$ ,  $\eta = 0.0$  psi-sec,  $\sigma_y = 4200$  psi. For stability, values of  $\eta$  between 104 to 106 psi/sec are recommended. Our limited experience thus far has shown that many problems require nonzero values of  $\eta$  to run to avoid error terminations.

Remark: Various other internal variables such as crack orientations and degraded stiffness tensors are internally calculated but currently not available for output.

**\*MAT\_SPOTWELD**

This is Material Type 100. The 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 constraint contact, \*CONTACT\_TIED\_SHELL\_EDGE\_TO\_SURFACE, 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, see \*CONSTRAINED\_EXTRA\_NODES\_OPTION. In the same way rigid bodies may also be tied together with this spotweld option. This weld option should not be used with rigid body switching.

**It is advisable to include all spotwelds, which provide the slave nodes, and spot welded materials, which define the master segments, within a single \*CONTACT\_TIED\_SHELL\_EDGE\_TO\_SURFACE interface.** As a constraint method these interfaces are treated independently which can lead to significant problems if such interfaces share common nodal points. The offset option, OFFSET, should not be used with spotwelds.

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	E	PR	SIGY	ET	DT	TFAIL
Type	I	F	F	F	F	F	F	F

Card 2            1            2            3            4            5            6            7            8

Variable	EFAIL	NRR	NRS	NRT	MRR	MSS	MTT	NF
Type	F	F	F	F	F	F	F	F

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
E	Young's modulus

<u>VARIABLE</u>	<u>DESCRIPTION</u>
PR	Poisson's ratio
SIGY	Initial yield stress
ET	Hardening modulus, $E_t$
DT	Time step size for mass scaling, $\Delta t$
TFAIL	Failure time if nonzero. If zero this option is ignored.
EFAIL	Effective plastic strain at failure.
NRR	Force resultant $N_{rr_F}$ at failure. If zero, failure due to this component is not considered.
NRS	Force resultant $N_{rs_F}$ at failure. If zero, failure due to this component is not considered.
NRT	Force resultant $N_{rt_F}$ at failure. If zero, failure due to this component is not considered.
MRR	Moment resultant $M_{rr_F}$ at failure. If zero, failure due to this component is not considered.
MSS	Moment resultant $M_{ss_F}$ at failure. If zero, failure due to this component is not considered.
MTT	Moment resultant $T_{rr_F}$ at failure. If zero, failure due to this component is not considered.
NF	Number of force vectors stored for filtering. The default value is set to zero which is generally recommended unless oscillatory resultant forces are observed in the time history databases. Even though these welds should not oscillate significantly, this option was added for consistency with the other spot weld options. NF affects the storage since it is necessary to store the resultant forces as history variables. When NF is nonzero, the resultants in the output databases are filtered.

### **Remarks:**

The weld material is modeled with isotropic hardening plasticity coupled to two failure models. The first model specifies a failure strain which fails each integration point in the spot weld independently. 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 NF is nonzero the resultants are filtered before failure is checked.

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  $\Delta t$ . 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  $\Delta t$ . If the value of  $\Delta t$  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.

Spotweld force history data is written into the SWFORC ascii file. In this database the resultant moments are not available, but they are in the binary time history database.

\*MAT\_ANISOTROPIC\_VISCOPLASTIC

This is Material Type 103. This anisotropic-viscoplastic material model applies to shell and brick elements. The material constants may be fit directly or, if desired, stress versus strain data may be input and a least squares fit will be performed by LS-DYNA to determine the constants. Kinematic or isotropic or a combination of kinematic and isotropic hardening may be used.. A detailed description of this model can be found in the following references: Berstad, Langseth, and Hopperstad [1994]; Hopperstad and Remseth [1995]; and Berstad [1996].

Card Format

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	E	PR	SIGY	FLAG	LCSS	ALPHA
Type	I	F	F	F	F	F	F	F

Card 2            1            2            3            4            5            6            7            8

Variable	QR1	CR1	QR2	CR2	QX1	CX1	QX2	CX2
Type	F	F	F	F	F	F	F	F

Card 3            1            2            3            4            5            6            7            8

Variable	VK	VM	R00 or F	R45 or G	R90 or H	L	M	N
Type	F	F	F	F	F	F	F	F

Card 4            1            2            3            4            5            6            7            8

Variable	AOPT							
Type	F							

Card 5

Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card 6

Variable	V1	V2	V3	D1	D2	D3	BETA	
Type	F	F	F	F	F	F	F	

**VARIABLE****DESCRIPTION**

MID	Material identification. A unique number has to be chosen.
RO	Mass density.
E	Young's modulus
PR	Poisson's ratio
SIGY	Initial yield stress
FLAG	Flag EQ.0 Give all material parameters EQ.1 Material parameters are fit in LS-DYNA to Load curve or Table given below. The parameters $Q_{r1}$ , $C_{r1}$ , $Q_{r2}$ , and $C_{r2}$ for isotropic hardening are determined by the fit and those for kinematic hardening are found by scaling those for isotropic hardening by $(1 - \alpha)$ where $\alpha$ is defined below in columns 51-60.
LCSS	Load curve ID or Table ID. The load curve ID defines effective stress versus effective plastic strain. Card 2 is ignored with this option. The table ID, see Figure 20.7, defines for each strain rate value a load curve ID giving the stress versus effective plastic strain for that rate. If the load curve only is used, then the coefficients $V_k$ and $V_m$ must be given if viscoplastic behavior is desired. If a Table ID is given these coefficients are determined internally during initialization.
ALPHA	$\alpha$ distribution of hardening used in the curve-fitting. $\alpha = 0$ pure kinematic hardening and $\alpha = 1$ provides pure isotropic hardening

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
QR1	Isotropic hardening parameter $Q_{r1}$
CR1	Isotropic hardening parameter $C_{r1}$
QR2	Isotropic hardening parameter $Q_{r2}$
CR2	Isotropic hardening parameter $C_{r2}$
QX1	Kinematic hardening parameter $Q_{\chi1}$
CX1	Kinematic hardening parameter $C_{\chi1}$
QX2	Kinematic hardening parameter $Q_{\chi2}$
CX2	Kinematic hardening parameter $C_{\chi2}$
VK	Viscous material parameter $V_k$
VM	Viscous material parameter $V_m$
R00	$R_{00}$ for shell (Default=1.0)
R45	$R_{45}$ for shell (Default=1.0)
R90	$R_{90}$ for shell (Default=1.0)
F	$F$ for brick (Default =1/2)
G	$G$ for brick (Default =1/2)
H	$H$ for brick (Default =1/2)
L	$L$ for brick (Default =3/2)
M	$M$ for brick (Default =3/2)
N	$N$ for brick (Default =3/2)
AOPT	Material axes option, see Figure 20.1:  EQ. 0.0: locally orthotropic with material axes determined by element nodes as shown in Figure 20.1. Nodes 1, 2, and 4 of an element are identical to the Nodes used for the definition of a coordinate system as by *DEFINE_COORDINATE_NODES.

---

**VARIABLE**

**DESCRIPTION**

EQ. 1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center; this is the a-direction.

EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with \*DEFINE\_COORDINATE\_VECTOR.

EQ. 3.0: applicable to shell elements only. This option determines locally orthotropic material axes by offsetting the material axes by an angle to be specified from a line in the plane of the shell determined by taking the cross product of the vector **v** defined below with the shell normal vector.

EQ. 4.0: locally orthotropic in cylindrical coordinate system with the material axes determined by a vector **v** and an originating point, P. They define the axis of symmetry.

XP,YP,ZP	x <sub>p</sub> y <sub>p</sub> z <sub>p</sub> , define coordinates of point <b>p</b> for AOPT = 1 and 4.
A1,A2,A3	a <sub>1</sub> a <sub>2</sub> a <sub>3</sub> , define components of vector <b>a</b> for AOPT = 2.
D1,D2,D3	d <sub>1</sub> d <sub>2</sub> d <sub>3</sub> , define components of vector <b>d</b> for AOPT = 2.
V1,V2,V3	v <sub>1</sub> v <sub>2</sub> v <sub>3</sub> , define components of vector <b>v</b> for AOPT = 3 and 4.
BETA	Material angle for AOPT = 3, which may be overridden on the element card, see *ELEMENT_SHELL.

**Remarks:**

The uniaxial stress-strain curve is given on the following form

$$\begin{aligned} \sigma(\epsilon_{eff}^p, \dot{\epsilon}_{eff}^p) = & \sigma_0 + Q_{r1}(1 - \exp(-C_{r1}\epsilon_{eff}^p)) + Q_{r2}(1 - \exp(-C_{r2}\epsilon_{eff}^p)) \\ & + Q_{\chi1}(1 - \exp(-C_{\chi1}\epsilon_{eff}^p)) + Q_{\chi2}(1 - \exp(-C_{\chi2}\epsilon_{eff}^p)) \\ & + V_k \dot{\epsilon}_{eff}^p V_m \end{aligned}$$

For bricks the following yield criteria is used

$$\begin{aligned} F(\sigma_{22} - \sigma_{33})^2 + G(\sigma_{33} - \sigma_{11})^2 + H(\sigma_{11} - \sigma_{22})^2 \\ + 2L\sigma_{23}^2 + 2M\sigma_{31}^2 + 2N\sigma_{12}^2 = \sigma(\epsilon_{eff}^p, \dot{\epsilon}_{eff}^p) \end{aligned}$$

where  $\epsilon_{eff}^p$  is the effective plastic strain and  $\dot{\epsilon}_{eff}^p$  is the effective plastic strain rate. For shells the anisotropic behavior is given by  $R_{00}$ ,  $R_{45}$  and  $R_{90}$ . The model will work when the three first parameters in card 3 are given values. When  $V_k = 0$  the material will behave elasto-plastically. Default values are given by:



$$F = G = H = \frac{1}{2}$$

$$L = M = N = \frac{3}{2}$$

$$R_{00} = R_{45} = R_{90} = 1$$

Strain rate of accounted for using the Cowper and Symonds model which, e.g., model 3, scales the yield stress with the factor:

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

To convert these constants set the viscoelastic constants,  $V_k$  and  $V_m$ , to the following values:

$$V_k = \sigma \left( \frac{1}{C} \right)^{1/p}$$

$$V_m = \frac{1}{p}$$

This model properly treats rate effects and should provide superior results to models 3 and 24.

**\*MAT\_DAMAGE\_1**

This is Material Type 104. This is a continuum damage mechanics (CDM) model which includes anisotropy and viscoplasticity. The CDM model applies to shell, thick shell, and brick elements. A more detailed description of this model can be found in the paper by Berstad, Hopperstad, Lademo, and Malo[1999].

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	E	PR	SIGY			
Type	I	F	F	F	F			

Card 2            1            2            3            4            5            6            7            8

Variable	Q1	C1	Q2	C2	EPSD	S	DC	FLAG
Type	F	F	F	F	F	F	F	F

Card 3            1            2            3            4            5            6            7            8

Variable	VK	VM	R00 or F	R45 or G	R90 or H	L	M	N
Type	F	F	F	F	F	F	F	F

Card 4            1            2            3            4            5            6            7            8

Variable	AOPT							
Type	F							

Card 5

Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card 6

Variable	V1	V2	V3	D1	D2	D3	BETA	
Type	F	F	F	F	F	F	F	

**VARIABLE**

**DESCRIPTION**

MID	Material identification. A unique number has to be chosen.
RO	Mass density.
E	Young's modulus
PR	Poisson's ratio
SIGY	Initial yield stress, $\sigma_0$ .
Q1	Isotropic hardening parameter $Q_1$
C1	Isotropic hardening parameter $C_1$
Q2	Isotropic hardening parameter $Q_2$
C2	Isotropic hardening parameter $C_2$
EPSD	Damage threshold $r_d$ Damage effective plastic strain when material softening begin.(Default=0.0)
S	Damage material constant $S$ . (Default= $\frac{\sigma_0}{200}$ )
DC	Critical damage value $D_c$ . When the damage value $D$ reaches this value, the element is deleted from the calculation. (Default=0.5)

<b>VARIABLE</b>	<b>DESCRIPTION</b>
FLAG	Flag EQ.0.No calculation of localization EQ.1.The model flags element where strain localization occur
VK	Viscous material parameter $V_k$
VM	Viscous material parameter $V_m$
R00	$R_{00}$ for shell (Default=1.0)
R45	$R_{45}$ for shell (Default=1.0)
R90	$R_{90}$ for shell (Default=1.0)
F	$F$ for brick (Default =1/2)
G	$G$ for brick (Default =1/2)
H	$H$ for brick (Default =1/2)
L	$L$ for brick (Default =3/2)
M	$M$ for brick (Default =3/2)
N	$N$ for brick (Default =3/2)
AOPT	Material axes option, see Figure 20.1:  EQ. 0.0: locally orthotropic with material axes determined by element nodes as shown in Figure 20.1. Nodes 1, 2, and 4 of an element are identical to the Nodes used for the definition of a coordinate system as by *DEFINE_COORDINATE_NODES.  EQ. 1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center; this is the a-direction.  EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR.  EQ. 3.0: applicable to shell elements only. This option determines locally orthotropic material axes by offsetting the material axes by an angle to be specified from a line in the plane of the shell determined by taking the cross product of the vector $\mathbf{v}$ defined below with the shell normal vector.  EQ. 4.0: locally orthotropic in cylindrical coordinate system with the material axes determined by a vector $\mathbf{v}$ and an originating point, P. They define the axis of symmetry.

VARIABLE	DESCRIPTION
XP,YP,ZP	$x_p, y_p, z_p$ , define coordinates of point <b>p</b> for AOPT = 1 and 4.
A1,A2,A3	$a_1, a_2, a_3$ , define components of vector <b>a</b> for AOPT = 2.
D1,D2,D3	$d_1, d_2, d_3$ , define components of vector <b>d</b> for AOPT = 2.
V1,V2,V3	$v_1, v_2, v_3$ , define components of vector <b>v</b> for AOPT = 3 and 4.
BETA	Material angle for AOPT = 3, which may be overridden on the element card, see *ELEMENT_SHELL.

### Remarks:

The Continuum Damage Mechanics (CDM) model is based on a CDM model proposed by Lemaitre [1992]. The effective stress  $\tilde{\sigma}$ , which is the stress calculated over the section that effectively resist the forces and reads.

$$\tilde{\sigma} = \frac{\sigma}{1-D}$$

where  $D$  is the damage variable. The evolution equation for the damage variable is defined as

$$\dot{D} = \begin{cases} 0 & \text{for } r \leq r_D \\ \frac{Y}{S(1-D)} \dot{r} & \text{for } r > r_D \text{ and } \sigma_1 > 0 \end{cases}$$

where  $r_D$  is the damage threshold,  $S$  is a positive material constant,  $Y$  is the so-called strain energy release rate and  $\sigma_1$  is the maximum principal stress. The strain energy density release rate is

$$Y = \frac{1}{2} \mathbf{e}_e : \mathbf{C} : \mathbf{e}_e = \frac{\sigma_{vm}^2 R_v}{2E(1-D)^2}$$

where  $\sigma_{vm}$  is the equivalent von Mises stress. The triaxiality function  $R_v$  is defined as

$$R_v = \frac{2}{3}(1+\nu) + 3(1-2\nu) \left( \frac{\sigma_H}{\sigma_{vm}} \right)^2$$

The uniaxial stress-strain curve is given in the following form

$$\sigma(r, \dot{\epsilon}_{eff}^p) = \sigma_0 + Q_1(1 - \exp(-C_1 r)) + Q_2(1 - \exp(-C_2 r)) + V_k \dot{\epsilon}_{eff}^p V_m$$

where  $r$  is the damage accumulated plastic strain, which can be calculated by

$$\dot{r} = \dot{\epsilon}_{eff}^p (1 - D)$$

For bricks the following yield criteria is used

$$F(\tilde{\sigma}_{22} - \tilde{\sigma}_{33})^2 + G(\tilde{\sigma}_{33} - \tilde{\sigma}_{11})^2 + H(\tilde{\sigma}_{11} - \tilde{\sigma}_{22})^2 + 2L\tilde{\sigma}_{23}^2 + 2M\tilde{\sigma}_{31}^2 + 2N\tilde{\sigma}_{12}^2 = \sigma(r, \dot{\epsilon}_{eff}^p)$$

where  $r$  is the damage effective viscoplastic strain and  $\dot{\epsilon}_{eff}^p$  is the effective viscoplastic strain rate. For shells the anisotropic behavior is given by the R-values:  $R_{00}$ ,  $R_{45}$ , and  $R_{90}$ . When  $V_k = 0$  the material will behave as an elastoplastic material without rate effects. Default values for the anisotropic constants are given by:

$$F = G = H = \frac{1}{2}$$

$$L = M = N = \frac{3}{2}$$

$$R_{00} = R_{45} = R_{90} = 1$$

so that isotropic behavior is obtained.

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

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

To convert these constants, set the viscoelastic constants,  $V_k$  and  $V_m$ , to the following values:

$$V_k = \sigma \left( \frac{1}{C} \right)^{1/p}$$

$$V_m = \frac{1}{p}$$

\*MAT\_DAMAGE\_2

This is Material Type 105. This is a elastic viscoplastic material model combined with continuum damage mechanincs (CDM). This material model applies to shell, thick shell, and brick elements. The elastoplastic behavior is described in the description of material model #24. A more detailed description of the CDM model is given in the description of material model #104 above.

Card Format

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	E	PR	SIGY	ETAN	FAIL	TDEL
Type	I	F	F	F	F	F	F	F
Default	none	none	none	none	none	0.0	10.E+20	0

Card 2

Variable	C	P	LCSS	LCSR				
Type	F	F	F	F				
Default	0	0	0	0				

Card 3

Variable	EPSD	S	DC					
Type	F	F	F					
Default	none	none	none					

Card 4

Variable	EPS1	EPS2	EPS3	EPS4	EPS5	EPS6	EPS7	EPS8
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

Card 5

Variable	ES1	ES2	ES3	ES4	ES5	ES6	ES7	ES8
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.
SIGY	Yield stress.
ETAN	Tangent modulus, ignored if (LCSS.GT.0) is defined.
FAIL	Failure flag. EQ.0.0: Failure due to plastic strain is not considered. GT.0.0: Plastic strain to failure. When the plastic strain reaches this value, the element is deleted from the calculation.
TDEL	Minimum time step size for automatic element deletion.



<u>VARIABLE</u>	<u>DESCRIPTION</u>
C	Strain rate parameter, C, see formula below.
P	Strain rate parameter, P, see formula below.
LCSS	Load curve ID or Table ID. Load curve ID defining effective stress versus effective plastic strain. If defined EPS1-EPS8 and ES1-ES8 are ignored. The table ID defines for each strain rate value a load curve ID giving the stress versus effective plastic strain for that rate, See Figure 20.7. The stress versus effective plastic strain curve for the lowest value of strain rate is used if the strain rate falls below the minimum value. Likewise, the stress versus effective plastic strain curve for the highest value of strain rate is used if the strain rate exceeds the maximum value. The strain rate parameters: C and P; the curve ID, LCSR; EPS1-EPS8 and ES1-ES8 are ignored if a Table ID is defined.
LCSR	Load curve ID defining strain rate scaling effect on yield stress.
EPSD	Damage threshold $r_d$ Damage effective plastic strain when material softening begin.(Default=0.0)
S	Damage material constant $S$ . (Default= $\frac{\sigma_0}{200}$ )
DC	Critical damage value $D_c$ . When the damage value $D$ reaches this value, the element is deleted from the calculation. (Default=0.5)
EPS1-EPS8	Effective plastic strain values (optional if SIGY is defined). At least 2 points should be defined.
ES1-ES8	Corresponding yield stress values to EPS1 - EPS8.

### **Remarks:**

The stress-strain behavior may be treated by a bilinear curve by defining the tangent modulus, ETAN. Alternately, a curve similar to that shown in Figure 20.4 is expected to be defined by (EPS1,ES1) - (EPS8,ES8); however, an effective stress versus effective plastic strain curve ID (LCSS) may be input instead if eight points are insufficient. The cost is roughly the same for either approach. The most general approach is to use the table definition with table ID, LCSR, discussed below.

Three options to account for strain rate effects are possible.

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

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

where  $\dot{\epsilon}$  is the strain rate.  $\dot{\epsilon} = \sqrt{\dot{\epsilon}_{ij} \dot{\epsilon}_{ij}}$

- II. For complete generality a load curve (LCSR) to scale the yield stress may be input instead. In this curve the scale factor versus strain rate is defined.
- III. If different stress versus strain curves can be provided for various strain rates, the option using the reference to a table (LCSS) can be used. Then the table input in \*DEFINE\_TABLE has to be used, see Figure 20.7.

A fully viscoplastic formulation is used in this model.

**\*MAT\_MODIFIED\_PIECEWISE\_LINEAR\_PLASTICITY**

This is Material Type 123. An elasto-plastic material with an arbitrary stress versus strain curve and arbitrary strain rate dependency can be defined. This model is currently available for shell elements only. Another model, MAT\_PIECEWISE\_LINEAR\_PLASTICITY, is similar but lacks the enhanced failure criteria. Failure is based on effective plastic strain, plastic thinning, the major principal in plane strain component, or a minimum time step size. See the discussion under the model description for MAT\_PIECEWISE\_LINEAR\_PLASTICITY if more information is desired.

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	E	PR	SIGY	ETAN	FAIL	TDEL
Type	I	F	F	F	F	F	F	F
Default	none	none	none	none	none	0.0	10.E+20	0

Card 2

Variable	C	P	LCSS	LCSR	VP	EPSTHIN	EPSMAJ	NUMINT
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

Card 3

Variable	EPS1	EPS2	EPS3	EPS4	EPS5	EPS6	EPS7	EPS8
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

Card 4

Variable	ES1	ES2	ES3	ES4	ES5	ES6	ES7	ES8
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

**VARIABLE****DESCRIPTION**

MID	Material identification. A unique number has to be chosen.
RO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.
SIGY	Yield stress.
ETAN	Tangent modulus, ignored if (LCSS.GT.0) is defined.
FAIL	Failure flag. LT.0.0: User defined failure subroutine is called to determine failure EQ.0.0: Failure is not considered. This option is recommended if failure is not of interest since many calculations will be saved. GT.0.0: Plastic strain to failure. When the plastic strain reaches this value, the element is deleted from the calculation.
TDEL	Minimum time step size for automatic element deletion.
C	Strain rate parameter, C, see formula below.
P	Strain rate parameter, P, see formula below.
LCSS	Load curve ID or Table ID. Load curve ID defining effective stress versus effective plastic strain. If defined EPS1-EPS8 and ES1-ES8 are ignored. The table ID defines for each strain rate value a load curve ID giving the stress versus effective plastic strain for that rate, See Figure 20.7. The stress versus effective plastic strain curve for the lowest value of strain rate is used if the strain rate falls below the minimum value. Likewise, the stress versus effective plastic strain curve for the highest value of strain rate is used if the strain rate exceeds the maximum value. The strain rate parameters: C and P;

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
	the curve ID, LCSR; EPS1-EPS8 and ES1-ES8 are ignored if a Table ID is defined.
LCSR	Load curve ID defining strain rate scaling effect on yield stress.
VP	Formulation for rate effects (Currently not used with this model)
EPSTHIN	Thinning plastic strain at failure. This number should be given as a positive number.
EPSMAJ	Major in plane strain at failure.
NUMINT	Number of through thickness integration points which must fail before the element is deleted. (If zero, all points must fail.)
EPS1-EPS8	Effective plastic strain values (optional if SIGY is defined). At least 2 points should be defined. The first point must be zero corresponding to the initial yield stress. <b>WARNING:</b> If the first point is nonzero the yield stress is extrapolated to determine the initial yield. If this option is used SIGY and ETAN are ignored and may be input as zero.
ES1-ES8	Corresponding yield stress values to EPS1 - EPS8.

# \*MAT

## \*MAT\_PLASTICITY\_COMPRESSION\_TENSION

---

### \*MAT\_PLASTICITY\_COMPRESSION\_TENSION

This is Material Type 124. An isotropic elastic-plastic material where unique yield stress versus plastic strain curves can be defined for compression and tension.. Also, failure can occur based on a plastic strain or a minimum time step size. Rate effects are modelled by using the Cowper-Symonds strain rate model.

#### Card Format

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	E	PR	C	P	FAIL	TDEL
Type	I	F	F	F	F	F	F	F
Default	none	none	none	none	0	0	10.E+20	0

Card 2

Variable	LCIDC	LCIDT						
Type	I	I						
Default	0	0						

Card 3

Variable	PC	PT						
Type	F	F						
Default	0	0						

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.
C	Strain rate parameter, C, see formula below.
P	Strain rate parameter, P, see formula below.
FAIL	Failure flag. LT.0.0: User defined failure subroutine is called to determine failure EQ.0.0: Failure is not considered. This option is recommended if failure is not of interest since many calculations will be saved. GT.0.0: Plastic strain to failure. When the plastic strain reaches this value, the element is deleted from the calculation.
TDEL	Minimum time step size for automatic element deletion.
LCIDC	Load curve ID defining yield stress versus effective plastic strain in compression.
LCIDT	Load curve ID defining yield stress versus effective plastic strain in tension.
PC	Compressive mean stress (pressure) at which the yield stress follows load curve ID, LCIDC. If the pressure falls between PC and PT a weighted average of the two load curves is used.
PT	Tensile mean stress at which the yield stress follows load curve ID, LCIDT.

**Remarks:**

The stress strain behavior follows a different curve in compression than it does in tension. Compression. Tension is determined by the sign of the mean stress where a positive mean stress (i.e., a negative pressure) is indicative of tension. Two curves must be defined giving the yield stress versus effective plastic strain for both the tension and compression regimes.

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

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

where  $\dot{\epsilon}$  is the strain rate.  $\dot{\epsilon} = \sqrt{\dot{\epsilon}_{ij} \dot{\epsilon}_{ij}}$

**\*MAT\_MODIFIED\_HONEYCOMB**

This is Material Type 126. The major use of this material model is for honeycomb and foam materials with real anisotropic behavior. A nonlinear elastoplastic material behavior can be defined separately for all normal and shear stresses. These are considered to be fully uncoupled. See notes below.

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	E	PR	SIGY	VF	MU	BULK
Type	I	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	.05	0.0

Card 2

Variable	LCA	LCB	LCC	LCS	LCAB	LCBC	LCCA	LCSR
Type	F	F	F	F	F	F	F	F
Default	none	LCA	LCA	LCA	LCS	LCS	LCS	optional

Card 3            1            2            3            4            5            6            7            8

Variable	EAAU	EBBU	ECCU	GABU	GBCU	GCAU	AOPT	
Type	F	F	F	F	F	F		

Card 4

Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		



Card 5

Variable	D1	D2	D3	TSEF	SSEF			
Type	F	F	F	F	F			

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
E	Young's modulus for compacted honeycomb material.
PR	Poisson's ratio for compacted honeycomb material.
SIGY	Yield stress for fully compacted honeycomb.
VF	Relative volume at which the honeycomb is fully compacted.
MU	$\mu$ , material viscosity coefficient. (default=.05) Recommended.
BULK	Bulk viscosity flag: EQ.0.0: bulk viscosity is not used. This is recommended. EQ.1.0: bulk viscosity is active and $\mu=0$ This will give results identical to previous versions of LS-DYNA.
LCA	Load curve ID, see *DEFINE_CURVE, for sigma-aa versus normal strain component aa. For the corotational solid elements, types 0 and 9, engineering strain is expected, but for all other solid element formulations a logarithmic strain is expected. See notes below.
LCB	Load curve ID, see *DEFINE_CURVE, for sigma-bb versus normal strain component bb. For the corotational solid elements, types 0 and 9, engineering strain is expected, but for all other solid element formulations a logarithmic strain is expected. Default LCB=LCA. See notes below.
LCC	Load curve ID, see *DEFINE_CURVE, for sigma-cc versus normal strain component cc. For the corotational solid elements, types 0 and 9, engineering strain is expected, but for all other solid element formulations a logarithmic strain is expected. Default LCC=LCA. See notes below.

<b>VARIABLE</b>	<b>DESCRIPTION</b>
LCS	Load curve ID, see *DEFINE_CURVE, for shear stress versus shear strain. For the corotational solid elements, types 0 and 9, engineering strain is expected, but for all other solid element formulations a shear strain based on the deformed configuration is used.. Default LCS=LCA. Each component of shear stress may have its own load curve. See notes below.
LCAB	Load curve ID, see *DEFINE_CURVE, for sigma-ab versus shear strain-ab. For the corotational solid elements, types 0 and 9, engineering strain is expected, but for all other solid element formulations a shear strain based on the deformed configuration is used.. Default LCAB=LCS. See notes below.
LCBC	Load curve ID, see *DEFINE_CURVE, for sigma-bc versus shear strain-bc. For the corotational solid elements, types 0 and 9, engineering strain is expected, but for all other solid element formulations a shear strain based on the deformed configuration is used.. Default LCBC=LCS. See notes below.
LCCA	Load curve ID, see *DEFINE_CURVE, or sigma-ca versus shear strain-ca. For the corotational solid elements, types 0 and 9, engineering strain is expected, but for all other solid element formulations a shear strain based on the deformed configuration is used.. Default LCCA=LCS. See notes below.
LCRS	Load curve ID, see *DEFINE_CURVE, for strain-rate effects defining the scale factor versus strain rate $\dot{\epsilon} = \sqrt{\quad}$ . This is optional. The curves defined above are scaled using this curve.
EAAU	Elastic modulus $E_{aaU}$ in uncompressed configuration.
EBBU	Elastic modulus $E_{bbU}$ in uncompressed configuration.
ECCU	Elastic modulus $E_{ccU}$ in uncompressed configuration.
GABU	Shear modulus $G_{abU}$ in uncompressed configuration.
GBCU	Shear modulus $G_{bcU}$ in uncompressed configuration.
GCAU	Shear modulus $G_{caU}$ in uncompressed configuration.
AOPT	Material axes option: EQ. 0.0: locally orthotropic with material axes determined by element nodes as shown in Figure 20.1. Nodes 1, 2, and 4 of an element are identical to the Nodes used for the definition of a coordinate system as by *DEFINE_COORDINATE_NODES. EQ. 1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center, this is the a-direction. EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR.

VARIABLE	DESCRIPTION
	EQ. 3.0: This option determines locally orthotropic material axes by offsetting the material axes by an angle to be specified from a line in the plane of the 1-2-3-4 segment determined by taking the cross product of the vector $\mathbf{v}$ defined below with the segment normal vector.
XP YP ZP	Coordinates of point p for AOPT = 1.
A1 A2 A3	Components of vector a for AOPT = 2.
D1 D2 D3	Components of vector d for AOPT = 2.
TSEF	Tensile strain at element failure (element will erode).
SSEF	Shear strain at element failure (element will erode).

**Remarks:**

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

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

$$E_{aa} = E_{aau} + \beta(E - E_{aau}) \quad G_{ab} = G_{abu} + \beta(G - G_{abu})$$

$$E_{bb} = E_{bbu} + \beta(E - E_{bbu}) \quad G_{bc} = G_{bcu} + \beta(G - G_{bcu})$$

$$E_{cc} = E_{ccu} + \beta(E - E_{ccu}) \quad G_{ca} = G_{cau} + \beta(G - G_{cau})$$

where

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

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

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

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 moduli according to:

$$\begin{aligned}\sigma_{aa}^{n+1^{trial}} &= \sigma_{aa}^n + E_{aa} \Delta \varepsilon_{aa} & \sigma_{ab}^{n+1^{trial}} &= \sigma_{ab}^n + 2G_{ab} \Delta \varepsilon_{ab} \\ \sigma_{bb}^{n+1^{trial}} &= \sigma_{bb}^n + E_{bb} \Delta \varepsilon_{bb} & \sigma_{bc}^{n+1^{trial}} &= \sigma_{bc}^n + 2G_{bc} \Delta \varepsilon_{bc} \\ \sigma_{cc}^{n+1^{trial}} &= \sigma_{cc}^n + E_{cc} \Delta \varepsilon_{cc} & \sigma_{ca}^{n+1^{trial}} &= \sigma_{ca}^n + 2G_{ca} \Delta \varepsilon_{ca}\end{aligned}$$

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  $\lambda$  is either unity or a value taken from the load curve number, LCSR, that defines  $\lambda$  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

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

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

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

the effective trial stress to the yield stress,  $\sigma_y$  (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

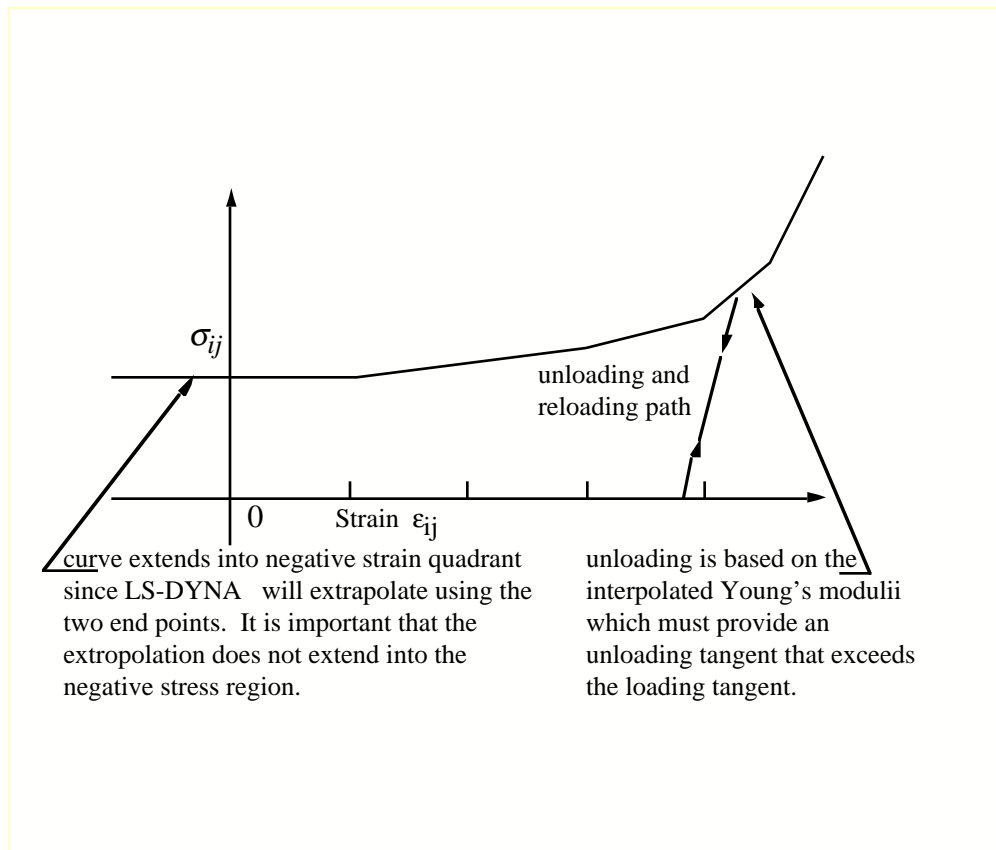
$$p^{n+1} = p^n - K \Delta \epsilon_{kk}^{n+1/2}$$

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

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 20.31.** 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.

**\*MAT\_ARRUDA\_BOYCE\_RUBBER**

This is Material Type 127. This material model provides a hyperelastic rubber model, see [Arruda and Boyce, 1993] combined optionally with linear viscoelasticity as outlined by [Christensen 1980].

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	K	G	N			
Type	I	F	F	F	I			

Card 2            1            2            3            4            5            6            7            8

Variable	LCID	TRAMP	NT					
Type	F	F	F					

**Card Format for Viscoelastic Constants. Up to 6 cards may be input. A keyword card (with a “\*” in column 1) terminates this input if less than 6 cards are used.**

Optional Cards            1            2            3            4            5            6            7            8

Variable	GI	BETAI						
Type	F	F						

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MID	Material identification. A unique number has to be chosen.
RO	Mass density
K	Bulk modulus
G	Shear modulus
N	Number of statistical links
LCID	Optional load curve ID of relaxation curve If constants $\beta t$ are determined via a least squares fit. This relaxation curve is shown in Figure 20.23. This model ignores the constant stress.
TRAMP	Optional ramp time for loading.
NT	Number of Prony series terms in optional fit. If zero, the default is 6. Currently, the maximum number is 6. Values less than 6, possibly 3-5 are recommended, since each term used adds significantly to the cost. Caution should be exercised when taking the results from the fit. Always check the results of the fit in the output file. Preferably, all generated coefficients should be positive. Negative values may lead to unstable results. Once a satisfactory fit has been achieved it is recommended that the coefficients which are written into the output file be input in future runs.
GI	Optional shear relaxation modulus for the <i>i</i> th term
BETAI	Optional decay constant if <i>i</i> th term

**Remarks:**

Rubber is generally considered to be fully incompressible since the bulk modulus greatly exceeds the shear modulus in magnitude. To model the rubber as an unconstrained material a hydrostatic work term,  $W_H(J)$ , is included in the strain energy functional which is function of the relative volume,  $J$ , [Ogden, 1984]:

$$W(J_1, J_2, J) = nk\theta \left[ \frac{1}{2}(J_1 - 3) + \frac{1}{20N}(J_1^2 - 9) + \frac{11}{1050N^2}(J_1^3 - 27) \right] \\ + nk\theta \left[ \frac{19}{7000N^3}(J_1^4 - 81) + \frac{519}{673750N^4}(J_1^5 - 243) \right] + W_H(J)$$

$$J_1 = I_1 J^{-2/3}$$

$$J_2 = I_2 J$$

where the hydrostatic work term is in terms of the bulk modulus,  $K$ , and the third invariant  $J$ , as:

$$W_H(J) = \frac{K}{2}(J - 1)^2$$

Rate effects are taken into account through linear viscoelasticity by a convolution integral of the form:

$$\sigma_{ij} = \int_0^t g_{ijkl}(t - \tau) \frac{\partial \epsilon_{kl}}{\partial \tau} d\tau$$

or in terms of the second Piola-Kirchhoff stress,  $S_{ij}$ , and Green's strain tensor,  $E_{ij}$ ,

$$S_{ij} = \int_0^t G_{ijkl}(t - \tau) \frac{\partial E_{kl}}{\partial \tau} d\tau$$

where  $g_{ijkl}(t - \tau)$  and  $G_{ijkl}(t - \tau)$  are the relaxation functions for the different stress measures. This stress is added to the stress tensor determined from the strain energy functional.

If we wish to include only simple rate effects, the relaxation function is represented by six terms from the Prony series:

$$g(t) = \alpha_0 + \sum_{m=1}^N \alpha_m e^{-\beta t}$$

given by,

$$g(t) = \sum_{i=1}^n G_i e^{-\beta_i t}$$

This model is effectively a Maxwell fluid which consists of dampers and springs in series. We characterize this in the input by shear moduli,  $G_i$ , and decay constants,  $\beta_i$ . The viscoelastic behavior is optional and an arbitrary number of terms may be used.



\*MAT\_HEART\_TISSUE

This is Material Type 128. This material model provides a heart tissue model described in the paper by Guccione, McCulloch, and Waldman [1991]. This model is transversely anisotropic.

Card Format

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	C	B1	B2	B3	P	
Type	I	F	F	F	F	F	F	

Card 2            1            2            3            4            5            6            7            8

Variable	AOPT							
Type	F							

Card 3

Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card 4

Variable	V1	V2	V3	D1	D2	D3	BETA	
Type	F	F	F	F	F	F	F	

<b>VARIABLE</b>	<b>DESCRIPTION</b>
C	Material coefficient.
B1	$b_1$ ,material coefficient.
B2	$b_2$ ,material coefficient.
B3	$b_3$ ,material coefficient.
P	Pressure in the muscle tissue.
AOPT	Material axes option, see Figure 20.1:  <p>EQ. 0.0: locally orthotropic with material axes determined by element nodes as shown in Figure 20.1. Nodes 1, 2, and 4 of an element are identical to the Nodes used for the definition of a coordinate system as by *DEFINE_COORDINATE_NODES.</p> <p>EQ. 1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center; this is the a-direction.</p> <p>EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR.</p> <p>EQ. 3.0: applicable to shell elements only. This option determines locally orthotropic material axes by offsetting the material axes by an angle to be specified from a line in the plane of the shell determined by taking the cross product of the vector <math>\mathbf{v}</math> defined below with the shell normal vector.</p> <p>EQ. 4.0: locally orthotropic in cylindrical coordinate system with the material axes determined by a vector <math>\mathbf{v}</math> and an originating point, P. They define the axis of symmetry.</p>
XP,YP,ZP	$x_p y_p z_p$ , define coordinates of point $\mathbf{p}$ for AOPT = 1 and 4.
A1,A2,A3	$a_1 a_2 a_3$ , define components of vector $\mathbf{a}$ for AOPT = 2.
D1,D2,D3	$d_1 d_2 d_3$ , define components of vector $\mathbf{d}$ for AOPT = 2.
V1,V2,V3	$v_1 v_2 v_3$ , define components of vector $\mathbf{v}$ for AOPT = 3 and 4.
BETA	Material angle in degrees for AOPT = 3, which may be overridden on the element card, see *ELEMENT_SHELL.

**Remarks:**

The tissue model is described in terms of the energy functional in terms of the Green strain components,  $E_{ij}$ ,

$$W(E) = \frac{c}{2}(e^Q - 1) + \frac{1}{2}P(I_3 - 1)$$

$$Q = b_1 E_{11}^2 + b_2 (E_{22}^2 + E_{33}^2 + E_{23}^2 + E_{32}^2) + b_3 (E_{12}^2 + E_{21}^2 + E_{13}^2 + E_{31}^2)$$

The Green components are modified to eliminate any effects of volumetric work following the procedures of Ogden.

**\*MAT\_LUNG\_TISSUE**

This is Material Type 129. This material model provides a hyperelastic model for heart tissue, see [Vawter, 1980] combined optionally with linear viscoelasticity as outlined by [Christensen 1980].

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	K	C	DELTA	ALPHA	BETA	
Type	I	F	F	F	I			

Card 2            1            2            3            4            5            6            7            8

Variable	C1	C2	LCID	TRAMP	NT			
Type	F	F	F	F	F			

**Card Format for Viscoelastic Constants. Up to 6 cards may be input. A keyword card (with a "\*" in column 1) terminates this input if less than 6 cards are used.**

Optional Cards            1            2            3            4            5            6            7            8

Variable	GI	BETAI						
Type	F	F						

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density
K	Bulk modulus
C	Material coefficient.
DELTA	$\Delta$ , material coefficient.
ALPHA	$\alpha$ , material coefficient.
BETA	$\beta$ , material coefficient.
C1	Material coefficient.
C2	Material coefficient.
LCID	Optional load curve ID of relaxation curve If constants $\beta t$ are determined via a least squares fit. This relaxation curve is shown in Figure 20.23. This model ignores the constant stress.
TRAMP	Optional ramp time for loading.
NT	Number of Prony series terms in optional fit. If zero, the default is 6. Currently, the maximum number is 6. Values less than 6, possibly 3-5 are recommended, since each term used adds significantly to the cost. Caution should be exercised when taking the results from the fit. Always check the results of the fit in the output file. Preferably, all generated coefficients should be positive. Negative values may lead to unstable results. Once a satisfactory fit has been achieved it is recommended that the coefficients which are written into the output file be input in future runs.
GI	Optional shear relaxation modulus for the ith term
BETAI	Optional decay constant if ith term

### **Remarks:**

The material is described by a strain energy functional expressed in terms of the invariants of the Green Strain:

$$W(I_1, I_2) = \frac{C}{2\Delta} e^{(\alpha I_1^2 + \beta I_2)} + \frac{12C_1}{\Delta(1+C_2)} [A^{(1+C_2)} - 1]$$

$$A^2 = \frac{4}{3}(I_1 + I_2) - 1$$

where the hydrostatic work term is in terms of the bulk modulus,  $K$ , and the third invariant  $J$ , as:

$$W_H(J) = \frac{K}{2}(J-1)^2$$

Rate effects are taken into account through linear viscoelasticity by a convolution integral of the form:

$$\sigma_{ij} = \int_0^t g_{ijkl}(t-\tau) \frac{\partial \varepsilon_{kl}}{\partial \tau} d\tau$$

or in terms of the second Piola-Kirchhoff stress,  $S_{ij}$ , and Green's strain tensor,  $E_{ij}$ ,

$$S_{ij} = \int_0^t G_{ijkl}(t-\tau) \frac{\partial E_{kl}}{\partial \tau} d\tau$$

where  $g_{ijkl}(t-\tau)$  and  $G_{ijkl}(t-\tau)$  are the relaxation functions for the different stress measures. This stress is added to the stress tensor determined from the strain energy functional.

If we wish to include only simple rate effects, the relaxation function is represented by six terms from the Prony series:

$$g(t) = \alpha_0 + \sum_{m=1}^N \alpha_m e^{-\beta t}$$

given by,

$$g(t) = \sum_{i=1}^n G_i e^{-\beta_i t}$$

This model is effectively a Maxwell fluid which consists of dampers and springs in series. We characterize this in the input by shear moduli,  $G_i$ , and decay constants,  $\beta_i$ . The viscoelastic behavior is optional and an arbitrary number of terms may be used.

\*MAT\_SPECIAL\_ORTHOTOPIC

This is Material Type 130. This model is available the Belytschko-Tsay and the C0 triangular shell elements and is based on a resultant stress formulation. In-plane behavior is treated separately from bending in order to model perforated materials such as television shadow masks. If other shell formulations are specified, the formulation will be automatically switched to Belyschko-Tsay.

Card Formats

Card 1            1            2            3            4            5            6            7            8

Variable	MID	RO	YS	EP				
Type	I	F	F	F				

Card 2

Variable	E11P	E22P	V12P	V21P	G12P	G23P	G31P	
Type	F	F	F	F	F	F	F	

Card 3

Variable	E11B	E22B	V12B	V21B	G12B	AOPT		
Type	F	F	F	F	F	F		

Card 4

Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card 5

Variable	V1	V2	V3	D1	D2	D3	BETA	
Type	F	F	F	F	F	F	F	

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
YS	Yield stress. This parameter is optional and is approximates the yield condition. Set to zero if the behavior is elastic.
EP	Plastic hardening modulus.
E11P	$E_{11p}$ , for in plane behavior.
E22P	$E_{22p}$ , for in plane behavior.
V12P	$\nu_{12p}$ , for in plane behavior.
V11P	$\nu_{21p}$ , for in plane behavior.
G12P	$G_{12p}$ , for in plane behavior.
G23P	$G_{23p}$ , for in plane behavior.
G31P	$G_{31p}$ , for in plane behavior.
E11B	$E_{11b}$ , for bending behavior.
E22B	$E_{22b}$ , for bending behavior.
V12B	$\nu_{12b}$ , for bending behavior.
V21B	$\nu_{21b}$ , for bending behavior.
G12B	$G_{12b}$ , for bending behavior.



VARIABLE	DESCRIPTION
AOPT	<p>Material axes option, see Figure 20.1:</p> <p>EQ. 0.0: locally orthotropic with material axes determined by element nodes as shown in Figure 20.1. Nodes 1, 2, and 4 of an element are identical to the Nodes used for the definition of a coordinate system as by *DEFINE_COORDINATE_NODES.</p> <p>EQ. 1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center; this is the a-direction.</p> <p>EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR.</p> <p>EQ. 3.0: applicable to shell elements only. This option determines locally orthotropic material axes by offsetting the material axes by an angle to be specified from a line in the plane of the shell determined by taking the cross product of the vector <math>\mathbf{v}</math> defined below with the shell normal vector.</p> <p>EQ. 4.0: locally orthotropic in cylindrical coordinate system with the material axes determined by a vector <math>\mathbf{v}</math> and an originating point, P. They define the axis of symmetry.</p>
XP,YP,ZP	$x_p$ $y_p$ $z_p$ , define coordinates of point $\mathbf{p}$ for AOPT = 1 and 4.
A1,A2,A3	$a_1$ $a_2$ $a_3$ , define components of vector $\mathbf{a}$ for AOPT = 2.
D1,D2,D3	$d_1$ $d_2$ $d_3$ , define components of vector $\mathbf{d}$ for AOPT = 2.
V1,V2,V3	$v_1$ $v_2$ $v_3$ , define components of vector $\mathbf{v}$ for AOPT = 3 and 4.
BETA	Material angle in degrees for AOPT = 3, which may be overridden on the element card, see *ELEMENT_SHELL.

**Remarks:**

The in-plane elastic matrix for in-plane, plane stress behavior is given by:

$$C_{in\ plane} = \begin{bmatrix} Q_{11p} & Q_{12p} & 0 & 0 & 0 \\ Q_{12p} & Q_{22p} & 0 & 0 & 0 \\ 0 & 0 & Q_{44p} & 0 & 0 \\ 0 & 0 & 0 & Q_{55p} & 0 \\ 0 & 0 & 0 & 0 & Q_{66p} \end{bmatrix}$$

The terms  $Q_{ijp}$  are defined as:

$$Q_{11p} = \frac{E_{11p}}{1 - \nu_{12p}\nu_{21p}}$$

$$Q_{22p} = \frac{E_{22p}}{1 - \nu_{12p}\nu_{21p}}$$

$$Q_{12p} = \frac{\nu_{12p}E_{11p}}{1 - \nu_{12p}\nu_{21p}}$$

$$Q_{44p} = G_{12p}$$

$$Q_{55p} = G_{23p}$$

$$Q_{66p} = G_{31p}$$

The elastic matrix for bending behavior is given by:

$$C_{bending} = \begin{bmatrix} Q_{11b} & Q_{12b} & 0 \\ Q_{12b} & Q_{22b} & 0 \\ 0 & 0 & Q_{44b} \end{bmatrix}$$

The terms  $Q_{ijp}$  are similarly defined.

**\*MAT\_SPRING\_ELASTIC**

This provides a translational or rotational elastic spring located between two nodes. Only one degree of freedom is connected.

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	MID	K						
Type	I	F						

**VARIABLE**

**DESCRIPTION**

- MID            Material ID. A unique number has to be chosen.
- K              Elastic stiffness (force/displacement) or (moment/rotation).

**\*MAT\_DAMPER\_VISCOUS**

This material provides a linear translational or rotational damper located between two nodes. Only one degree of freedom is then connected.

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	MID	DC							
Type	I	F							

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
-----------------	--------------------

---

MID            Material ID. A unique number has to be chosen.

DC            Damping constant (force/displacement rate) or (moment/rotation rate).

**\*MAT\_SPRING\_ELASTOPLASTIC**

This material provides an elastoplastic translational or rotational spring with isotropic hardening located between two nodes. Only one degree of freedom is connected.

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	MID	K	KT	FY				
Type	I	F	F	F				

**VARIABLE**

**DESCRIPTION**

- MID            Material number. A unique number has to be chosen.
- K              Elastic stiffness (force/displacement) or (moment/rotation).
- KT             Tangent stiffness (force/displacement) or (moment/rotation).
- FY             Yield (force) or (moment).

**\*MAT\_SPRING\_NONLINEAR\_ELASTIC**

This material provides a nonlinear elastic translational and rotational spring with arbitrary force versus displacement and moment versus rotation, respectively. Optionally, strain rate effects can be considered through a velocity dependent scale factor. With the spring located between two nodes, only one degree of freedom is connected.

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	MID	LCD	LCR						
Type	I	I	I						

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MID	Material number. A unique number has to be chosen.
LCD	Load curve ID describing force versus displacement or moment versus rotation relationship
LCR	Optional load curve describing scale factor on force or moment as a function of relative velocity or rotational velocity, respectively. <u>The load curve must define the response in the negative and positive quadrants and pass through point (0,0).</u>

\*MAT\_DAMPER\_NONLINEAR\_VISCOUS

This material provides a viscous translational damper with an arbitrary force versus velocity dependency, or a rotational damper with an arbitrary moment versus rotational velocity dependency. With the damper located between two nodes, only one degree of freedom is connected.

Card Format

Card 1            1            2            3            4            5            6            7            8

Variable	MID	LCDR						
Type	I	I						

VARIABLE

DESCRIPTION

MID

Material identification. A unique number has to be chosen.

LCDR

Load curve identification describing force versus rate-of-displacement relationship or a moment versus rate-of-rotation relationship. The load curve must define the response in the negative and positive quadrants and pass through point (0,0).

**\*MAT\_SPRING\_GENERAL\_NONLINEAR**

This material provides a general nonlinear translational or rotational spring with arbitrary loading and unloading definitions. Optionally, hardening or softening can be defined. With the spring located between two nodes, only one degree of freedom is connected.

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	MID	LCDL	LCDU	BETA	TYI	CYI		
Type	I	I	I	F	F	F		

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MID	Material identification. A unique number has to be chosen.
LCDL	Load curve identification describing force versus displacement resp. moment versus rotation relationship for loading, see Figure 20.32.
LCDU	Load curve identification describing force versus displacement resp. moment versus rotation relationship for unloading, see Figure 20.32.
BETA	Hardening parameter, $\beta$ : EQ.0.0: tensile and compressive yield with strain softening (negative or zero slope allowed in the force versus disp. load curves), NE.0.0: kinematic hardening <b>without strain softening</b> , EQ.1.0: isotropic <b>hardening without strain softening</b> .
TYI	Initial yield force in tension (> 0)
CYI	Initial yield force in compression (< 0)

**Remarks:**

Load curve points are in the format (displacement, force or rotation, moment). The points must be in order starting with the most negative (compressive) displacement resp. rotation and ending with the most positive (tensile) value. The curves need not be symmetrical.

The displacement origin of the “unloading” curve is arbitrary, since it will be shifted as necessary as the element extends and contracts. On reverse yielding the “loading” curve will also be shifted along the displacement resp. rotation axis. The initial tensile and compressive yield forces (TYI and CYI) define a range within which the element remains elastic (i.e. the “loading” curve is used for both loading and unloading). If at any time the force in the element exceeds this range, the element is deemed to have yielded, and at all subsequent times the “unloading” curve is used for unloading.



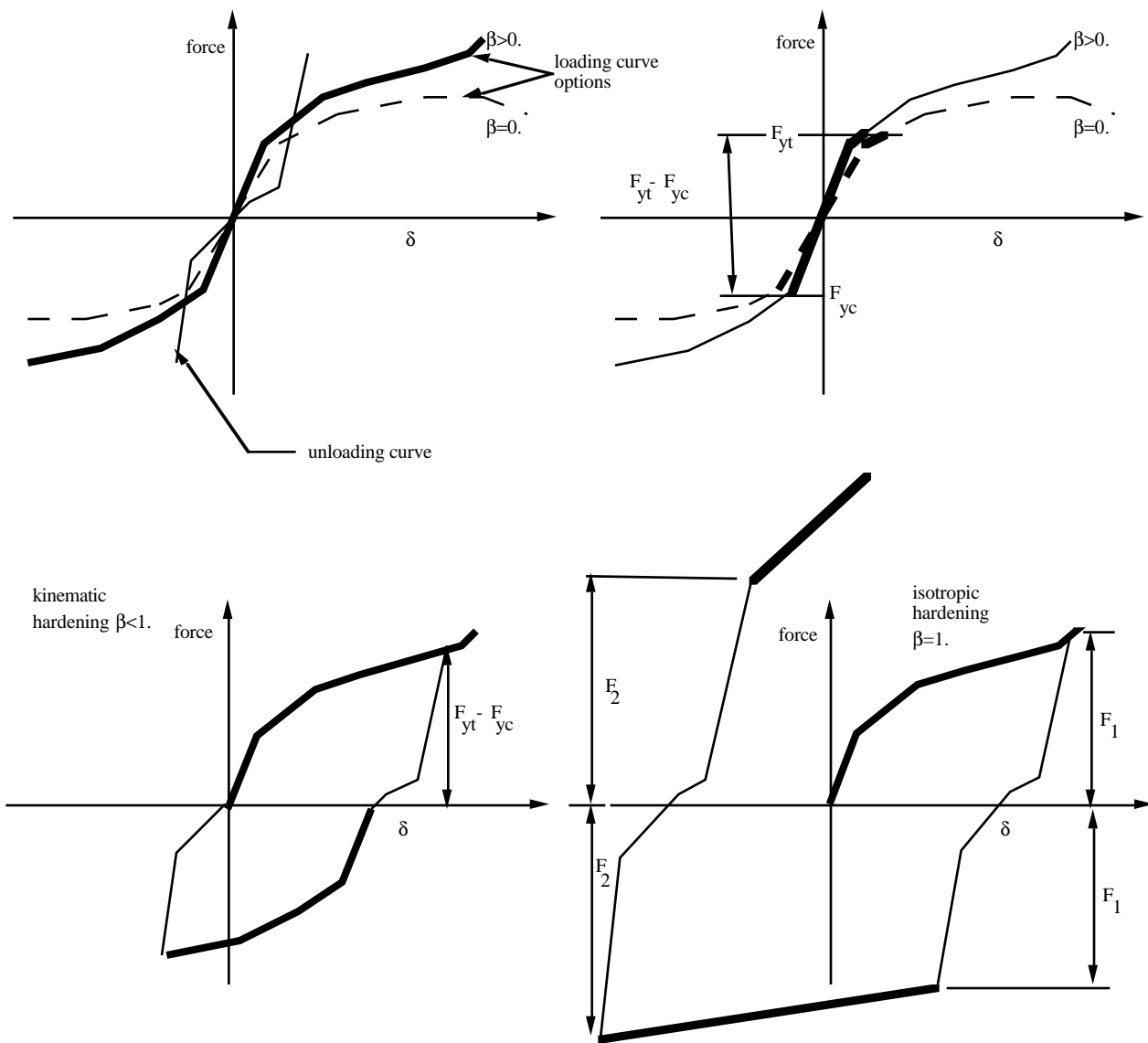


Figure 20.32. General nonlinear material for discrete elements.

**\*MAT\_SPRING\_MAXWELL**

This material provides a three Parameter Maxwell Viscoelastic translational or rotational spring. Optionally, a cutoff time with a remaining constant force/moment can be defined.

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	MID	K0	KI	BETA	TC	FC	COPT	
Type	I	F	F	F	F	F	F	
Default	---	---	---	---	10 <sup>20</sup>	0	0	

**VARIABLE****DESCRIPTION**

MID	Material identification. A unique number has to be chosen.
K0	$K_0$ , short time stiffness
KI	$K_\infty$ , long time stiffness
BETA	Decay parameter.
TC	Cut off time. After this time a constant force/moment is transmitted.
FC	Force/moment after cutoff time
COPT	Time implementation option: EQ.0: incremental time change, NE.0: continuous time change.

**Remarks:**

The time varying stiffness  $K(t)$  may be described in terms of the input parameters as

$$K(t) = K_\infty + (K_0 - K_\infty)e^{-\beta t} .$$

This equation was implemented by Schwer [60] as either a continuous function of time or incrementally following the approach of Herrmann and Peterson [61]. The continuous function of time implementation has the disadvantage of the energy absorber's resistance decaying with increasing time even without deformation. The advantage of the incremental implementation is that an energy absorber must undergo some deformation before its resistance decays, i.e., there is no

decay until impact, even in delayed impacts. The disadvantage of the incremental implementation is that very rapid decreases in resistance cannot be easily matched.

**\*MAT\_SPRING\_INELASTIC**

This material provides an inelastic tension or compression only, translational or rotational spring. Optionally, a user-specified unloading stiffness can be taken instead of the maximum loading stiffness.

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	MID	LCFD	KU	CTF				
Type	I	I	F	F				

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MID	Material identification. A unique number has to be chosen.
LCFD	Load curve identification 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.
KU	Unloading stiffness (optional). The maximum of KU and the maximum loading stiffness in the force/displacement or the moment/twist curve is used for unloading.
CTF	Flag for compression/tension: EQ.-1.0: tension only, EQ.0.0: default is set to 1.0, EQ.1.0: compression only.

\*MAT\_SPRING\_MUSCLE

This material is a Hill-type muscle model with activation. It is for use with discrete elements. The LS-DYNA implementation is due to Dr. J.A. Weiss.

Card Format

Card 1      1            2            3            4            5            6            7            8

Variable	MID	<i>Lo</i>	<i>Vmax</i>	<i>Sv</i>	<i>A</i>	<i>Fmax</i>	<i>TL</i>	<i>TV</i>
Type	I	F	F	F	F	F	F	F
Default		1.0		1.0			1.0	1.0

Card 2      1            2            3            4            5            6            7            8

Variable	<i>Fpe</i>	<i>Lmax</i>	<i>Ksh</i>					
Type	F	F	F					
Default	0.0							

**VARIABLE**

**DESCRIPTION**

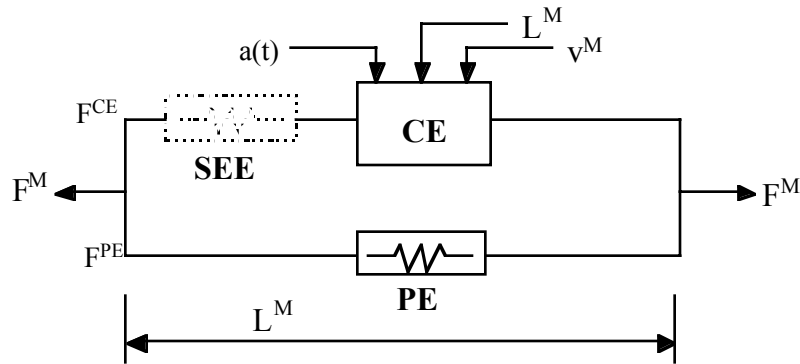
- Lo*            Initial muscle length.
- Vmax*        Maximum CE shortening velocity.
- Sv*            Scale factor for *Vmax* vs. active state.  
                  .LT.0: absolute value gives load curve ID  
                  .GE.0: constant value of 1.0 is used
- A*             Activation level vs. time function.  
                  .LT.0: absolute value gives load curve ID  
                  .GE.0: constant value of *A* is used
- Fmax*        Peak isometric force.

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
<i>TL</i>	Active tension vs. length function. .LT.0: absolute value gives load curve ID .GE.0: constant value of 1.0 is used
<i>TV</i>	Active tension vs. velocity function. .LT.0: absolute value gives load curve ID .GE.0: constant value of 1.0 is used
<i>Fpe</i>	Force vs. length function for parallel elastic element. .LT.0: absolute value gives load curve ID .EQ.0: exponential function is used (see below) .GT.0: constant value of 0.0 is used
<i>Lmax</i>	Relative length when <i>Fpe</i> reaches <i>Fmax</i> . Required if <i>Fpe</i> =0 above.
<i>Ksh</i>	Constant governing the exponential rise of <i>Fpe</i> . Required if <i>Fpe</i> =0 above.

**Remarks:**

The material behavior of the muscle model is adapted from the original model proposed by Hill (1938). Reviews of this model and extensions can be found in Winters (1990) and Zajac (1989). The most basic Hill-type muscle model consists of a contractile element (CE) and a parallel elastic element (PE) (Figure 20.33). An additional series elastic element (SEE) can be added to represent tendon compliance. The main assumptions of the Hill model are that the contractile element is entirely stress free and freely distensible in the resting state, and is described exactly by Hill's equation (or some variation). When the muscle is activated, the series and parallel elements are elastic, and the whole muscle is a simple combination of identical sarcomeres in series and parallel. The main criticism of Hill's model is that the division of forces between the parallel elements and the division of extensions between the series elements is arbitrary, and cannot be made without introducing auxiliary hypotheses. However, these criticisms apply to *any* discrete element model. Despite these limitations, the Hill model has become extremely useful for modeling musculoskeletal dynamics, as illustrated by its widespread use today.



**Figure 20.33** Discrete model for muscle contraction dynamics, based on a Hill-type representation. The total force is the sum of passive force  $F^{PE}$  and active force  $F^{CE}$ . The passive element (PE) represents energy storage from muscle elasticity, while the contractile element (CE) represents force generation by the muscle. The series elastic element (SEE), shown in dashed lines, is often neglected when a series tendon compliance is included. Here,  $a(t)$  is the activation level,  $L^M$  is the length of the muscle, and  $v^M$  is the shortening velocity of the muscle.

When the contractile element (CE) of the Hill model is inactive, the entire resistance to elongation is provided by the PE element and the tendon load-elongation behavior. As activation is increased, force then passes through the CE side of the parallel Hill model, providing the contractile dynamics. The original Hill model accommodated only full activation - this limitation is circumvented in the present implementation by using the modification suggested by Winters (1990). The main features of his approach were to realize that the CE force-velocity input force equals the CE tension-length output force. This yields a three-dimensional curve to describe the force-velocity-length relationship of the CE. If the force-velocity y-intercept scales with activation, then given the activation, length and velocity, the CE force can be determined.

Without the SEE, the total force in the muscle  $F^M$  is the sum of the force in the CE and the PE because they are in parallel:

$$F^M = F^{PE} + F^{CE}$$

The relationships defining the force generated by the CE and PE as a function of  $L^M$ ,  $v^M$  and  $a(t)$  are often scaled by  $F_{max}$ , the peak isometric force (p. 80, Winters 1990),  $L_0$ , the initial length of the muscle (p. 81, Winters 1990), and  $V_{max}$ , the maximum unloaded CE shortening velocity (p. 80, Winters 1990). From these, dimensionless length and velocity can be defined:

$$L = \frac{L^M}{L_0},$$

$$V = \frac{v^M}{V_{max} * S_v(a(t))}$$

Here,  $S_v$  scales the maximum CE shortening velocity  $V_{max}$  and changes with activation level  $a(t)$ . This has been suggested by several researchers, i.e. Winters and Stark (1985). The activation

level specifies the level of muscle stimulation as a function of time. Both have values between 0 and 1. The functions  $S_V(a(t))$  and  $a(t)$  are specified via load curves in LS-DYNA, or default values of  $S_V=1$  and  $a(t)=0$  are used. Note that  $L$  is always positive and that  $V$  is positive for lengthening and negative for shortening.

The relationship between  $F^{CE}$ ,  $V$  and  $L$  was proposed by Bahler et al. (1967). A three-dimensional relationship between these quantities is now considered standard for computer implementations of Hill-type muscle models (i.e., eqn 5.16, p. 81, Winters 1990). It can be written in dimensionless form as:

$$F^{CE} = a(t) * F_{max} * f_{TL}(L) * f_{TV}(V)$$

Here,  $f_{TL}$  and  $f_{TV}$  are the tension-length and tension-velocity functions for active skeletal muscle. Thus, if current values of  $L^M$ ,  $V^M$ , and  $a(t)$  are known, then  $F^{CE}$  can be determined (Figure 20.33).

The force in the parallel elastic element  $F^{PE}$  is determined directly from the current length of the muscle using an exponential relationship (eqn 5.5, p. 73, Winters 1990):

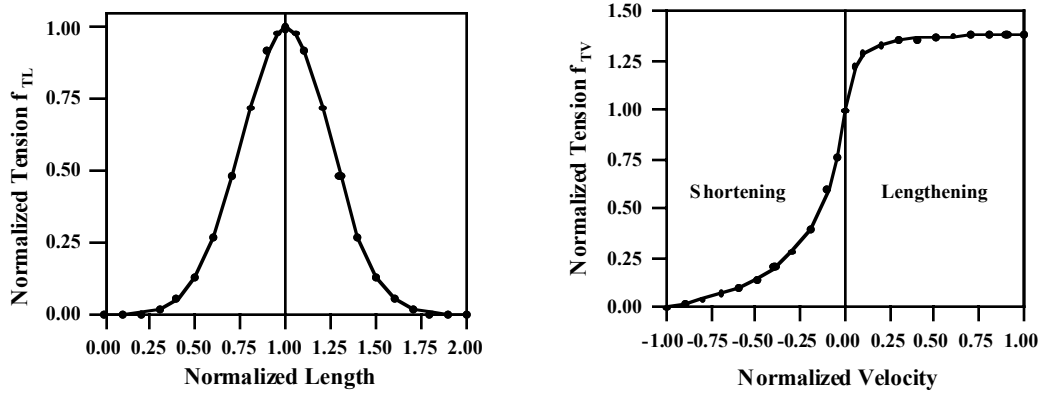
$$f_{PE} = \frac{F^{PE}}{F_{MAX}} = 0, \quad L \leq 1$$

$$f_{PE} = \frac{F^{PE}}{F_{MAX}} = \frac{1}{\exp(K_{sh}) - 1} \left[ \exp\left(\frac{K_{sh}}{L_{max}}(L - 1)\right) - 1 \right], \quad L > 1$$

Here,  $L_{max}$  is the relative length at which the force  $F_{max}$  occurs, and  $K_{sh}$  is a dimensionless shape parameter controlling the rate of rise of the exponential. Alternatively, the user can define a custom  $f_{PE}$  curve giving tabular values of normalized force versus dimensionless length as a load curve.

For computation of the total force developed in the muscle  $F^M$ , the functions for the tension-length  $f_{TL}$  and force-velocity  $f_{TV}$  relationships used in the Hill element must be defined. These relationships have been available for over 50 years, but have been refined to allow for behavior such as active lengthening. The active tension-length curve  $f_{TL}$  describes the fact that isometric muscle force development is a function of length, with the maximum force occurring at an optimal length. According to Winters, this optimal length is typically around  $L=1.05$ , and the force drops off for shorter or longer lengths, approaching zero force for  $L=0.4$  and  $L=1.5$ . Thus the curve has a bell-shape. Because of the variability in this curve between muscles, the user must specify the function  $f_{TL}$  via a load curve, specifying pairs of points representing the normalized force (with values between 0 and 1) and normalized length  $L$  (Figure 20.34).





**Figure 20.34** Typical normalized tension-length (TL) and tension-velocity (TV) curves for skeletal muscle.

The active tension-velocity relationship  $f_{TV}$  used in the muscle model is mainly due to the original work of Hill. Note that the dimensionless velocity  $V$  is used. When  $V=0$ , the normalized tension is typically chosen to have a value of 1.0. When  $V$  is greater than or equal to 0, muscle lengthening occurs. As  $V$  increases, the function is typically designed so that the force increases from a value of 1.0 and asymptotes towards a value near 1.4. When  $V$  is less than zero, muscle shortening occurs and the classic Hill equation hyperbola is used to drop the normalized tension to 0 (Figure 20.32). The user must specify the function  $f_{TV}$  via a load curve, specifying pairs of points representing the normalized tension (with values between 0 and 1) and normalized velocity  $V$ .

**\*MAT\_SEATBELT**

Purpose: Define a seat belt material. See notes below.

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	MID	MPUL	LLCID	ULCID	LMIN			
Type	I	F	I	I	F			
Default	0	0.	0	0	0.0			

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MID	Belt material number. A unique number has to be chosen.
MPUL	Mass per unit length
LLCID	Load curve identification for loading (strain/force with engineering strain).
ULCID	Load curve identification for unloading (strain/force with engineering strain).
LMIN	Minimum length (for elements connected to slip rings and retractors), see notes below.

**Remarks:**

Each belt material defines stretch characteristics and mass properties for a set of belt elements. The user enters a load curve for loading, the points of which are (Strain, Force). Strain is defined as engineering strain, i.e.

$$Strain = \frac{current\ length}{initial\ length} - 1.$$

Another similar curve is entered to describe the unloading behavior. Both 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 force opposes the relative motion of the nodes and is limited by stability:

$$D = \frac{.1 \times mass \times relative \ velocity}{time \ step \ size}$$

In addition, the magnitude of the damping force is limited to one-tenth of the force calculated from the force-strain relationship and is zero when the belt is slack. Damping forces are not applied to elements attached to slings 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 slings or are absorbed into the retractors. Onetenth of a typical initial element length is usually a good choice.

**\*MAT\_THERMAL\_OPTION**

The \*MAT\_THERMAL\_ cards allow thermal properties to be defined in coupled structural/thermal and thermal only analyses, see \*CONTROL\_SOLUTION. Thermal properties must be defined for all solid and shell elements in such analyses. Thermal properties need not be defined for beam or discrete elements as these elements are not accounted for in the thermal phase of the calculation. However dummy thermal properties will be echoed for these elements in the D3HSP file.

Thermal material properties are specified by a thermal material ID number (TMID), this number is independent of the material ID number (MID) defined on all other \*MAT\_.. property cards. In the same analysis identical TMID and MID numbers may exist. The TMID and MID numbers are related through the \*PART card.

\*MAT\_THERMAL\_ISOTROPIC

This is thermal material property type 1. It allows isotropic thermal properties to be defined.

Card Format (1 of 2)

	1	2	3	4	5	6	7	8
Variable	TMID	TRO	TGRLC	TGMULT				
Type	I	F	F	F				

Card Format (2 of 2)

	1	2	3	4	5	6	7	8
Variable	HC	TC						
Type	F	F						

<u>VARIABLE</u>	<u>DESCRIPTION</u>
TMID	Thermal material identification, a unique number has to be chosen.
TRO	Thermal density: EQ 0.0 default to structural density.
TGRLC	Thermal generation rate curve number, see *DEFINE_CURVE: GT.0: function versus time, EQ.0: use constant multiplier value, TGMULT, LT.0: function versus temperature.
TGMULT	Thermal generation rate multiplier: EQ.0.0: no heat generation.
HC	Heat capacity
TC	Thermal conductivity

**\*MAT\_THERMAL\_ORTHOTROPIC**

This is thermal material property type 2. It allows orthotropic thermal properties to be defined.

**Card Format (1 of 4)**

1            2            3            4            5            6            7            8

Variable	TMD	TRO	TGRLC	TGMULT	AOPT			
Type	I	F	F	F	F			

**Card Format (2 of 4)**

1            2            3            4            5            6            7            8

Variable	HC	K1	K2	K3				
Type	F	F	F	F				

**Card Format (3 of 4)**

1            2            3            4            5            6            7            8

Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card Format (4 of 4)

	1	2	3	4	5	6	7	8
Variable	D1	D2	D3					
Type	F	F	F					

VARIABLE	DESCRIPTION
T MID	Thermal material identification, a unique number has to be chosen.
T RO	Thermal density: EQ 0.0 default to structural density.
T GR LC	Thermal generation rate curve number, see *DEFINE_CURVE: GT.0: function versus time, EQ.0: use constant multiplier value, TGMULT, LT.0: function versus temperature.
T GM UL T	Thermal generation rate multiplier: EQ.0.0: no heat generation.
A OPT	Material axes definition: EQ.0: locally orthotropic with material axes by element nodes $N_1$ , $N_2$ and $N_4$ , EQ.1: locally orthotropic with material axes determined by a point in space and global location of element center, EQ.2: globally orthotropic with material axes determined by vectors.
H C	Heat capacity
K <sub>1</sub>	Thermal conductivity K <sub>1</sub> in local x-direction
K <sub>2</sub>	Thermal conductivity K <sub>2</sub> in local y-direction
K <sub>3</sub>	Thermal conductivity K <sub>3</sub> in local z-direction
X P, Y P, Z P	Define coordinate of point <b>p</b> for AOPT = 1
A 1, A 2, A 3	Define components of vector <b>a</b> for AOPT = 2
D 1, D 2, D 3	Define components of vector <b>v</b> for AOPT = 2

**\*MAT\_THERMAL\_ISOTROPIC\_TD**

This is thermal material property type 3. It allows temperature dependent isotropic properties to be defined. The temperature dependency is defined by specifying a minimum of two and a maximum of eight data points. The properties must be defined for the temperature range that the material will see in the analysis.

**Card Format (1 of 4)**

	1	2	3	4	5	6	7	8
Variable	TMD	TRO	TGRLC	TGMULT				
Type	I	F	F	F				

**Card Format (2 of 4)**

	1	2	3	4	5	6	7	8
Variable	T1	T2	T3	T4	T5	T6	T7	T8
Type	F	F	F	F	F	F	F	F

**Card Format (3 of 4)**

	1	2	3	4	5	6	7	8
Variable	C1	C2	C3	C4	C5	C6	C7	C8
Type	F	F	F	F	F	F	F	F



Card Format (4 of 4)

	1	2	3	4	5	6	7	8
Variable	K1	K2	K3	K4	K5	K6	K7	K8
Type	F	F	F	F	F	F	F	F

<u>VARIABLE</u>	<u>DESCRIPTION</u>
TMID	Thermal material identification, a unique number has to be chosen.
TRO	Thermal density: EQ 0.0 default to structural density.
TGRLC	Thermal generation rate curve number, see *DEFINE_CURVE: GT.0: function versus time, EQ.0: use constant multiplier value, TGMULT, LT.0: function versus temperature.
TGMULT	Thermal generation rate multiplier: EQ.0.0: no heat generation.
T1 ... T8	Temperatures (T1 ... T8)
C1 ... C8	Heat capacity at T1 ... T8
K1 ... K8	Thermal conductivity at T1 ... T8

**\*MAT\_THERMAL\_ORTHOTROPIC\_TD**

This is thermal material property type 4. It allows temperature dependent orthotropic properties to be defined. The temperature dependency is defined by specifying a minimum of two and a maximum of eight data points. The properties must be defined for the temperature range that the material will see in the analysis.

**Card Format (1 of 8)**

1            2            3            4            5            6            7            8

Variable	TMD	TRO	TGRLC	TGMULT	AOPT			
Type	I	F	F	F	F			

**Card Format (2 of 8)**

1            2            3            4            5            6            7            8

Variable	T1	T2	T3	T4	T5	T6	T7	T8
Type	F	F	F	F	F	F	F	F

**Card Format (3 of 8)**

1            2            3            4            5            6            7            8

Variable	C1	C2	C3	C4	C5	C6	C7	C8
Type	F	F	F	F	F	F	F	F

**Card Format (4 of 8)**

1 2 3 4 5 6 7 8

Variable	(K1) 1	(K1) 2	(K1) 3	(K1) 4	(K1) 5	(K1) 6	(K1) 7	(K1) 8
Type	F	F	F	F	F	F	F	F

**Card Format (5 of 8)**

1 2 3 4 5 6 7 8

Variable	(K2) 1	(K2) 2	(K2) 3	(K2) 4	(K2) 5	(K2) 6	(K2) 7	(K2) 8
Type	F	F	F	F	F	F	F	F

**Card Format (6 of 8)**

1 2 3 4 5 6 7 8

Variable	(K3) 1	(K3) 2	(K3) 3	(K3) 4	(K3) 5	(K3) 6	(K3) 7	(K3) 8
Type	F	F	F	F	F	F	F	F

**Card Format (7 of 8)**

1 2 3 4 5 6 7 8

Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

**Card Format (8 of 8)**

	1	2	3	4	5	6	7	8
Variable	D1	D2	D3					
Type	F	F	F					

<u>VARIABLE</u>	<u>DESCRIPTION</u>
TMID	Thermal material identification, a unique number has to be chosen.
TRO	Thermal density: EQ.0.0 default to structural density.
TGRLC	Thermal generation rate curve number, see *DEFINE_CURVE: GT.0: function versus time, EQ.0: use constant multiplier value, TGMULT, LT.0: function versus temperature.
TGMULT	Thermal generation rate multiplier: EQ.0.0: no heat generation.
AOPT	Material axes definition: EQ.0: locally orthotropic with material axes by element nodes $N_1$ , $N_2$ and $N_4$ , EQ.1: locally orthotropic with material axes determined by a point in space and global location of element center, EQ.2: globally orthotropic with material axes determined by vectors.
T1 ... T8	Temperatures (T1 ... T8)
C1 ... C8	Heat capacity at T1 ... T8
(K1) <sub>1</sub> ... (K1) <sub>8</sub>	Thermal conductivity $K_1$ in local x-direction at T1 ... T8
(K2) <sub>1</sub> ... (K2) <sub>8</sub>	Thermal conductivity $K_2$ in local y-direction at T1 ... T8
(K3) <sub>1</sub> ... (K3) <sub>8</sub>	Thermal conductivity $K_3$ in local z-direction at T1 ... T8
XP, YP, ZP	Define coordinate of point $\mathbf{p}$ for AOPT = 1
A1, A2, A3	Define components of vector $\mathbf{a}$ for AOPT = 2
D1, D2, D3	Define components of vector $\mathbf{v}$ for AOPT = 2

**\*MAT\_THERMAL\_ISOTROPIC\_PHASE\_CHANGE**

This is thermal material property type 9. It allows temperature dependent isotropic properties with phase change to be defined. The latent heat of the material is defined together with the solidus and liquidus temperatures. The temperature dependency is defined by specifying a minimum of two and a maximum of eight data points. The properties must be defined for the temperature range that the material will see in the analysis.

**Card Format (1 of 5)**

	1	2	3	4	5	6	7	8
Variable	TMID	TRO	TGRLC	TGMULT				
Type	I	F	F	F				

**Card Format (2 of 5)**

	1	2	3	4	5	6	7	8
Variable	T1	T2	T3	T4	T5	T6	T7	T8
Type	F	F	F	F	F	F	F	F

**Card Format (3 of 5)**

	1	2	3	4	5	6	7	8
Variable	C1	C2	C3	C4	C5	C6	C7	C8
Type	F	F	F	F	F	F	F	F

**Card Format (4 of 5)**

1            2            3            4            5            6            7            8

Variable	K1	K2	K3	K4	K5	K6	K7	K8
Type	F	F	F	F	F	F	F	F

**Card Format (5 of 5)**

1            2            3            4            5            6            7            8

Variable	SOLT	LIQT	LH					
Type	F	F	F					

**VARIABLE****DESCRIPTION**

TMID	Thermal material identification, a unique number has to be chosen.
TRO	Thermal density: EQ 0.0 default to structural density.
TGRLC	Thermal generation rate curve number, see *DEFINE_CURVE: GT.0: function versus time, EQ.0: use constant multiplier value, TGMULT, LT.0: function versus temperature.
TGMULT	Thermal generation rate multiplier: EQ.0.0: no heat generation.
T1 ... T8	Temperatures (T1 ... T8)
C1 ... C8	Heat capacity at T1 ... T8
K1 ... K8	Thermal conductivity at T1 ... T8
SOLT	Solidus temperature, $T_S$ (must be $< T_L$ )
LIQT	Liquidus temperature, $T_L$ (must be $> T_S$ )
LH	Latent heat

**Remarks:**

During phase change, that is between the solidus and liquidus temperatures, the heat capacity of the material will be enhanced to account for the latent heat as follows:

$$c(t) = m \left[ 1 - \cos 2\pi \left( \frac{T - T_s}{T_L - T_s} \right) \right] \quad T_s < T < T_L$$

Where

$T_L$  = liquidus temperature

$T_s$  = solidus temperature

$T$  = temperature

$m$  = multiplier such that  $\lambda = \int_{T_s}^{T_L} C(T) dT$

$\lambda$  = latent heat

$c$  = heat capacity

**\*MAT\_THERMAL\_ISOTROPIC\_TD\_LC**

This is thermal material property type 10. It allows isotropic thermal properties that are temperature dependent specified by load curves to be defined. The properties must be defined for the temperature range that the material will see in the analysis.

**Card Format (1 of 2)**

	1	2	3	4	5	6	7	8
Variable	TMID	TRO	TGRLC	TGMULT				
Type	I	F	F	F				

**Card Format (2 of 2)**

	1	2	3	4	5	6	7	8
Variable	HCLC	TCLC						
Type	F	F						

<u>VARIABLE</u>	<u>DESCRIPTION</u>
TMID	Thermal material identification, a unique number has to be chosen.
TRO	Thermal density: EQ 0.0 default to structural density.
TGRLC	Thermal generation rate curve number, see *DEFINE_CURVE: GT.0: function versus time, EQ.0: use constant multiplier value, TGMULT, LT.0: function versus temperature.
TGMULT	Thermal generation rate multiplier: EQ.0.0: no heat generation.
HCLC	Load curve ID specifying heat capacity vs. temperature.
TCLC	Load curve ID specifying thermal conductivity vs. temperature.



# \*NODE

## \*NODE

Purpose: Define a node and its coordinates in the global coordinate system. Also, the boundary conditions in global directions can be specified. Generally, nodes are assigned to elements; however, exceptions are possible, see remark 2 below.

### Card Format (I8,3E16.0,2F8.0)

Card 1            1            2            3            4            5            6            7            8            9            10

Variable	NID	X	Y	Z	TC	RC				
Type	I	F	F	F	F	F				
Default	none	0.	0.	0.	0.	0.				
Remarks					1	1				

---

**VARIABLE**

---

**DESCRIPTION**

NID	Node number
X	x coordinate
Y	y coordinate
Z	z coordinate
TC	Translational Constraint: EQ.0: no constraints, EQ.1: constrained x displacement, EQ.2: constrained y displacement, EQ.3: constrained z displacement, EQ.4: constrained x and y displacements, EQ.5: constrained y and z displacements, EQ.6: constrained z and x displacements, EQ.7: constrained x, y, and z displacements.

## \*NODE

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
RC	Rotational constraint: EQ.0: no constraints, EQ.1: constrained x rotation, EQ.2: constrained y rotation, EQ.3: constrained z rotation, EQ.4: constrained x and y rotations, EQ.5: constrained y and z rotations, EQ.6: constrained z and x rotations, EQ.7: constrained x, y, and z rotations.

### Remarks:

1. Boundary conditions can also be defined on nodal points in a local (or global) system by using the keyword **\*BOUNDARY\_SPC**. For other possibilities also see the **\*CONSTRAINED** keyword section of the manual.
2. A node without an element or a mass attached to it will be assigned a very small amount of mass and rotary inertia. Generally, massless nodes should not cause any problems but in rare cases may create stability problems if these massless nodes interact with the structure. Warning messages are printed when massless nodes are found. Also, massless nodes are used with rigid bodies to place joints, see **\*CONSTRAINED\_EXTRA\_NODES\_OPTION** and **\*CONSTRAINED\_NODAL\_RIGID\_BODY**.

# **\*PART**

Two keywords are used in this section.

**\*PART\_{*OPTION1*}\_{*OPTION2*}\_{*OPTION3*}**

**\*PART\_MOVE**

# \*PART

---

**\*PART\_{OPTION1}\_{OPTION2}\_{OPTION3}**

For *OPTION1* the available choices are

**<BLANK>**

**PART\_INERTIA**

**PART\_REPOSITION**

For *OPTION2* the available choices are

**<BLANK>**

**CONTACT**

For *OPTION3* the available choices are

**<BLANK>**

**PRINT**

The options may be specified in any order.

Purpose: Define parts, i.e., combine material and section information as well as hourglass control, thermal properties, and general specification for adaptivity.

The **INERTIA** option allows the inertial properties and initial conditions to be defined rather than calculated from the finite element mesh. This applies to rigid bodies, see **\*MAT\_RIGID**, only. The **REPOSITION** option applies to deformable materials and is used to reposition deformable materials attached to rigid dummy components whose motion is controlled by either **CAL3D** or **MADYMO**. At the beginning of the calculation each component controlled by **CAL3D/MADYMO** is automatically repositioned to be consistent with the **CAL3D/MADYMO** input. However, deformable materials attached to these component will not be repositioned unless this option is used.

The **CONTACT** option allows part based contact parameters to be used with the automatic contact types a3, 4, a5, a10, 13, a13, 15 and 26, that is

**\*CONTACT\_AUTOMATIC\_SURFACE\_TO\_SURFACE**

**\*CONTACT\_SINGLE\_SURFACE,**

**\*CONTACT\_AUTOMATIC\_NODES\_TO\_SURFACE,**

**\*CONTACT\_AUTOMATIC\_ONE\_WAY\_SURFACE\_TO\_SURFACE,**

- \*CONTACT\_AUTOMATIC\_SINGLE\_SURFACE,
- \*CONTACT\_AIRBAG\_SINGLE\_SURFACE,
- \*CONTACT\_ERODING\_SINGLE\_SURFACE,
- \*CONTACT\_AUTOMATIC\_GENERAL.

The default values to use for these contact parameters can be specified on the \*CONTACT input section.card.

The PRINT option allows user control over whether output data is written into the ASCII files MATSUM and RBDOUT. See \*DATABASE\_ASCII.

**Card Format**

Card 1

Variable	HEADING	
Type	C	
Default	none	
Remarks	1	

Card 2            1            2            3            4            5            6            7            8

Variable	PID	SECID	MID	EOSID	HGID	GRAV	ADPOPT	TMID
Type	I	I	I	I	I	I	I	I
Default	none	none	none	0	0	0	0	0

# \*PART

---

**Additional Cards are required for the INERTIA option. See remarks 3 and 4.**

Card 3            1            2            3            4            5            6            7            8

Variable	XC	YC	ZC	TM	IRCS			
Type	F	F	F	F	I			

Card 4            1            2            3            4            5            6            7            8

Variable	IXX	IXY	IXZ	IYY	IYZ	IZZ		
Type	F	F	F	F	F	F		

Card 5            1            2            3            4            5            6            7            8

Variable	VTX	VTY	VTZ	VRX	VRY	VRZ		
Type	F	F	F	F	F	F		

**Optional card required for IRCS=1. Define two local vectors or a local coordinate system ID.**

Card 6            1            2            3            4            5            6            7            8

Variable	XL	YL	ZL	XLIP	YLIP	ZLIP	CID	
Type	F	F	F	F	F	F	I	
Remark	2	2	2	2	2	2	none	

**Additional Card is required for the REPOSITION option.**

Optional            1            2            3            4            5            6            7            8

Variable	CMSN	MDEP	MOVOPT					
Type	I	I	I					

**Additional Card is required for the CONTACT option.**

**WARNING:** If FS, FD, DC, and VC are specified they will not be used unless FS is set to a negative value (-1.0) in the \*CONTACT section. These frictional coefficients apply only to contact types: SINGLE\_SURFACE, AUTOMATIC\_GENERAL, AUTOMATIC\_SINGLE\_SURFACE, AUTOMATIC\_NODES\_TO ..., AUTOMATIC\_SURFACE\_..., AUTOMATIC\_ONE\_WAY\_..., and ERODING\_SINGLE\_SURFACE. Default values are input via \*CONTROL\_CONTACT input.

Optional            1            2            3            4            5            6            7            8

Variable	FS	FD	DC	VC	OPTT	SFT	SSF	
Type	F	F	F	F	F	F	F	

**Additional Card is required for the PRINT option.**

Optional            1            2            3            4            5            6            7            8

Variable	PRBF							
Type	I							

**VARIABLE**

**DESCRIPTION**

HEADING	Heading for the part
PID	Part identification
SECID	Section identification defined in the *SECTION section

# \*PART

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MID	Material identification defined in the *MAT section
EOSID	Equation of state identification defined in the *EOS section. Nonzero only for solid elements using a an equation of state to compute pressure.
HGID	Hourglass/bulk viscosity identification defined in the *HOURGLASS Section: EQ.0: default values are used.
GRAV	Part initialization for gravity loading. This option initializes hydrostatic pressure in the part due to gravity acting on an overburden material. This option applies to brick elements only and must be used with the *LOAD_DENSITY_DEPTH option: EQ.0: all parts initialized, EQ.1: only current material initialized.
ADPOPT	Indicate if this part is adapted or not. see also *CONTROL_ADAPTIVITY: EQ.0: no adaptivity, EQ.1: H-adaptive for 3-D shells. EQ.2: R-adaptive remeshing for 2-D shells.
TMID	Thermal material property identification defined in the *MAT_THERMAL Section. Thermal properties must be specified for all solid, shell, and thick shell parts if a thermal or coupled thermal structural/analysis is being performed. Beams and discrete elements are not considered in thermal analyses. EQ.0: defaults to MID
XC	x-coordinate of center of mass
YC	y-coordinate of center of mass
ZC	z-coordinate of center of mass
TM	Translational mass
IRCS	Flag for inertia tensor reference coordinate system: EQ.0: global inertia tensor, EQ.1: principal moments of inertia with orientation vectors.
IXX	$I_{xx}$ , xx component of inertia tensor
IXY	$I_{xy}$ (set to zero if IRCS=1)
IXZ	$I_{xz}$ (set to zero if IRCS=1)
IYY	$I_{yy}$ , yy component of inertia tensor
IYZ	$I_{yz}$ (set to zero if IRCS=1)



---

<b>VARIABLE</b>	<b>DESCRIPTION</b>
I <sub>ZZ</sub>	I <sub>ZZ</sub> , zz component of inertia tensor
VTX	initial translational velocity of rigid body in x direction
VTY	initial translational velocity of rigid body in y direction
VTZ	initial translational velocity of rigid body in z direction
VRX	initial rotational velocity of rigid body about x axis
VR <sub>Y</sub>	initial rotational velocity of rigid body about y axis
VR <sub>Z</sub>	initial rotational velocity of rigid body about z axis
XL	x-coordinate of local x-axis. Origin lies at (0,0,0).
YL	y-coordinate of local x-axis
ZL	z-coordinate of local x-axis
XLIP	x-coordinate of vector in local x-y plane
YLIP	y-coordinate of vector in local x-y plane
ZLIP	z-coordinate of vecotr in local x-y plane
CID	Local coordinate system ID, see *DEFINE_COORDINATE_.... With this option leave fields 1-6 blank.
CMSN	CAL3D segment number/MADYMO system number. See the numbering in the corresponding program.
MDEP	MADYMO ellipse/plane number: GT.0: ellipse number, EQ.0: default, LT.0: absolute value is plane number.
MOVOPT	Flag to deactivate moving for merged rigid bodies, see *CONSTRAINED_RIGID_BODIES. This option allows a merged rigid body to be fixed in space while the nodes and elements of the generated CAL3D/MADYMO parts are repositioned: EQ.0: merged rigid body is repositioned, EQ.1: merged rigid body is not repositioned.
FS	Static coefficient of friction. The functional coefficient is assumed to be dependent on the relative velocity $v_{rel}$ of the surfaces in contact $\mu_c = FD + (FS - FD)e^{-DC v_{rel} }$

---

# \*PART

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
FD	Dynamic coefficient of friction. The functional coefficient is assumed to be dependent on the relative velocity $v_{rel}$ of the surfaces in contact $\mu_c = FD + (FS - FD)e^{-DC v_{rel} }$ .
DC	Exponential decay coefficient. The functional coefficient is assumed to be dependent on the relative velocity $v_{rel}$ of the surfaces in contact $\mu_c = FD + (FS - FD)e^{-DC v_{rel} }$ .
SFT	Optional thickness scale factor for PART ID in automatic contact (scales true thickness). This option applies only to contact with shell elements. True thickness is the element thickness of the shell elements.
SSF	Scale factor on default slave penalty stiffness for this PART ID whenever it appears in the contact definition. If zero, SSF is taken as unity.
PRBF	Print flag for RBDOUT and MATSUM files. EQ.0: default is taken from the keyword *CONTROL_OUTPUT, EQ.1: write data into RBDOUT file only EQ.2: write data into MATSUM file only EQ.3: do not write data into RBDOUT and MATSUM

## **Remarks:**

1. HEADING default is standard material description, e.g. Material Type 1. In case of SMUG post processing place PSHELL (or PBAR, or PSOLID) in columns 1-8 and Property name in columns 34-41.
2. The local cartesian coordinate system is defined as described in \*DEFINE\_COORDINATE\_VECTOR. The local z-axis vector is the vector cross product of the x axis and the in plane vector. The local y-axis vector is finally computed as the vector cross product of the z-axis vector and the x-axis vector. The local coordinate system defined by CID has the advantage that the local system can be defined by nodes in the rigid body which makes repositioning of the rigid body in a preprocessor much easier since the local system moves with the nodal points.
3. When specifying mass properties for a rigid body using the inertia option, the mass contributions of deformable bodies to nodes which are shared by the rigid body should be considered as part of the rigid body.
4. If the inertia option is used, all mass and inertia properties of the body must be specified for there are no default values.

**\*PART\_MOVE**

Purpose: Translate shell part by an increment. This option currently applies only to shell elements.

**Define one card. Card Format (I8,3E16.0)**

Card 1            1            2            3            4            5            6            7            8            9            10

Variable	PID	XMOV	YMOV	ZMOV						
Type	I	F	F	F						
Default	none	0.	0.	0.						

**VARIABLE****DESCRIPTION**

PID	Part identification
XMOV	Move shell part ID, PID, in the x-direction by the incremental distance, XMOV.
YMOV	Move shell part ID, PID, in the y-direction by the incremental distance, YMOV.
ZMOV	Move shell part ID, PID, in the z-direction by the incremental distance, ZMOV.



# **\*RIGIDWALL**

Two keywords are used in this section to define rigid surfaces:

**\*RIGIDWALL\_GEOMETRIC\_OPTION\_{OPTION}**

**\*RIGIDWALL\_PLANAR\_{OPTION}\_{OPTION}\_{OPTION}**

The RIGIDWALL option provides a simple way of treating contact between a rigid surface and nodal points of a deformable body, called slave nodes. Slave nodes which belong to rigid parts are not, in general, checked for contact with only one exception. The RIGIDWALL\_PLANAR option may be used with nodal points of rigid bodies if the planar wall defined by this option is fixed in space and the RWPNAL parameter is set to a positive nonzero value on the control card, \*CONTROL\_CONTACT.

When the rigid wall defined in this section moves with a prescribed motion, the equations of rigid body mechanics are not involved. For a general rigid body treatment with arbitrary surfaces and motion, refer to the \*CONTACT\_ENTITY definition. The \*CONTACT\_ENTITY option is for treating contact between rigid and deformable surfaces only.

# \*RIGIDWALL

---

\*RIGIDWALL\_GEOMETRIC\_OPTION\_{OPTION}

Available forms include (one is mandatory):

**RIGIDWALL\_GEOMETRIC\_FLAT**  
**RIGIDWALL\_GEOMETRIC\_PRISM**  
**RIGIDWALL\_GEOMETRIC\_CYLINDER**  
**RIGIDWALL\_GEOMETRIC\_SPHERE**

If prescribed motion is desired an additional option is available:

**MOTION**

One of the shape types [**FLAT, PRISM, CYLINDER, SPHERE**] must be specified, followed by the optional definition of **MOTION**, both on the same line with \***RIGIDWALL\_GEOMETRIC**

Purpose: Define a rigid wall with an analytically described form. Four forms are possible. A prescribed motion is optional. For general rigid bodies with arbitrary surfaces and motion, refer to the \*CONTACT\_ENTITY definition. This option is for treating contact between rigid and deformable surfaces only.

## Card Format:

- Cards 1 and 2 are required for all geometric shapes.
- Card 3 is required, but is dependent upon which shape is specified.
- Optional Card A is required if MOTION is specified.

**Card 1 - Required for all shape types**

.

Card 1            1            2            3            4            5            6            7            8

Variable	NSID	NSIDEX	BOXID					
Type	I	I	I					
Default	none	0	0					
Remarks								

**VARIABLE****DESCRIPTION**

NSID	Nodal set ID containing slave nodes, see *SET_NODE_OPTION: EQ.0: all nodes are slave to rigid wall.
NSIDEX	Nodal set ID containing nodes that exempted as slave nodes, see *SET_NODE_OPTION.
BOXID	If defined, only nodes in box are included as slave nodes to rigid wall.

# \*RIGIDWALL

---

## Card 2 - Required for all shape types.

Card 2            1            2            3            4            5            6            7            8

Variable	XT	YT	ZT	XH	YH	ZH	FRIC	
Type	F	F	F	F	F	F	F	
Default	0.	0.	0.	0.	0.	0.	0.	
Remarks								

### VARIABLE

### DESCRIPTION

XT	x-coordinate of tail of any outward drawn normal vector, <b>n</b> , originating on wall (tail) and terminating in space (head), see Figure 23.1.
YT	y-coordinate of tail of normal vector <b>n</b>
ZT	z-coordinate of tail of normal vector <b>n</b>
XH	x-coordinate of head of normal vector <b>n</b>
YH	y-coordinate of head of normal vector <b>n</b>
ZH	z-coordinate of head of normal vector <b>n</b>
FRIC	Interface friction: EQ.0.0: frictionless sliding after contact, EQ.1.0: stick condition after contact, 0.<FRIC<1.: Coulomb friction coefficient.



**Card 3 - Required if FLAT is specified after the keyword.**

A plane with a finite size or with an infinite size can be defined, see Figure 23.1. The vector **m** is computed as the vector cross product **n X l**. The origin, which is the tail of the normal vector, is the corner point of the finite size plane.

Card 3            1            2            3            4            5            6            7            8

Variable	XHEV	YHEV	ZHEV	LENL	LENM			
Type	F	F	F	F	F			
Default	0.	0.	0.	infinity	infinity			

**VARIABLE****DESCRIPTION**

XHEV	x-coordinate of head of edge vector <b>l</b> , see Figure 23.1.
YHEV	y-coordinate of head of edge vector <b>l</b>
ZHEV	z-coordinate of head of edge vector <b>l</b>
LENL	Length of <b>l</b> edge. A zero value defines an infinite size plane.
LENM	Length of <b>m</b> edge. A zero value defines an infinite size plane.

# \*RIGIDWALL

---

## Card 3 - Required if PRISM is specified after the keyword.

The description of the definition of a plane with finite size is enhanced by an additional length in the direction negative to **n**, see Figure 23.1.

Card 3            1            2            3            4            5            6            7            8

Variable	XHEV	YHEV	ZHEV	LENL	LENM	LENP		
Type	F	F	F	F	F	F		
Default	none	0.	0.	infinity	infinity	infinity		

<u>VARIABLE</u>	<u>DESCRIPTION</u>
-----------------	--------------------

XHEV	x-coordinate of head of edge vector <b>l</b> , see Figure 23.1.
YHEV	y-coordinate of head of edge vector <b>l</b>
ZHEV	z-coordinate of head of edge vector <b>l</b>
LENL	Length of <b>l</b> edge. A zero valure defines an infinite size plane.
LENM	Length of <b>m</b> edge. A zero valure defines an infinite size plane.
LENP	Length of prism in the direction negative to <b>n</b> , see Figure 23.1.

**Card 3 - Required if CYLINDER is specified after the keyword.**

The tail of **n** specifies the top plane of the cylinder. The length is defined in the direction negative to **n**, see Figure 23.1.

Card 3            1            2            3            4            5            6            7            8

Variable	RADCYL	LENCYL						
Type	F	F						
Default	none	infinity						

<u>VARIABLE</u>	<u>DESCRIPTION</u>
RADCYL	Radius of cylinder
LENCYL	Length of cylinder, see Figure 23.1. Only if a valure larger than zero is specified is a finite length assumed.

# \*RIGIDWALL

---

**Card 3 - Required if SPHERE is specified after the keyword.**

The center of the sphere is identical to the tail of **n**, see Figure 23.1.

Card 3            1            2            3            4            5            6            7            8

Variable	RADSPH							
Type	F							
Default	0.							

---

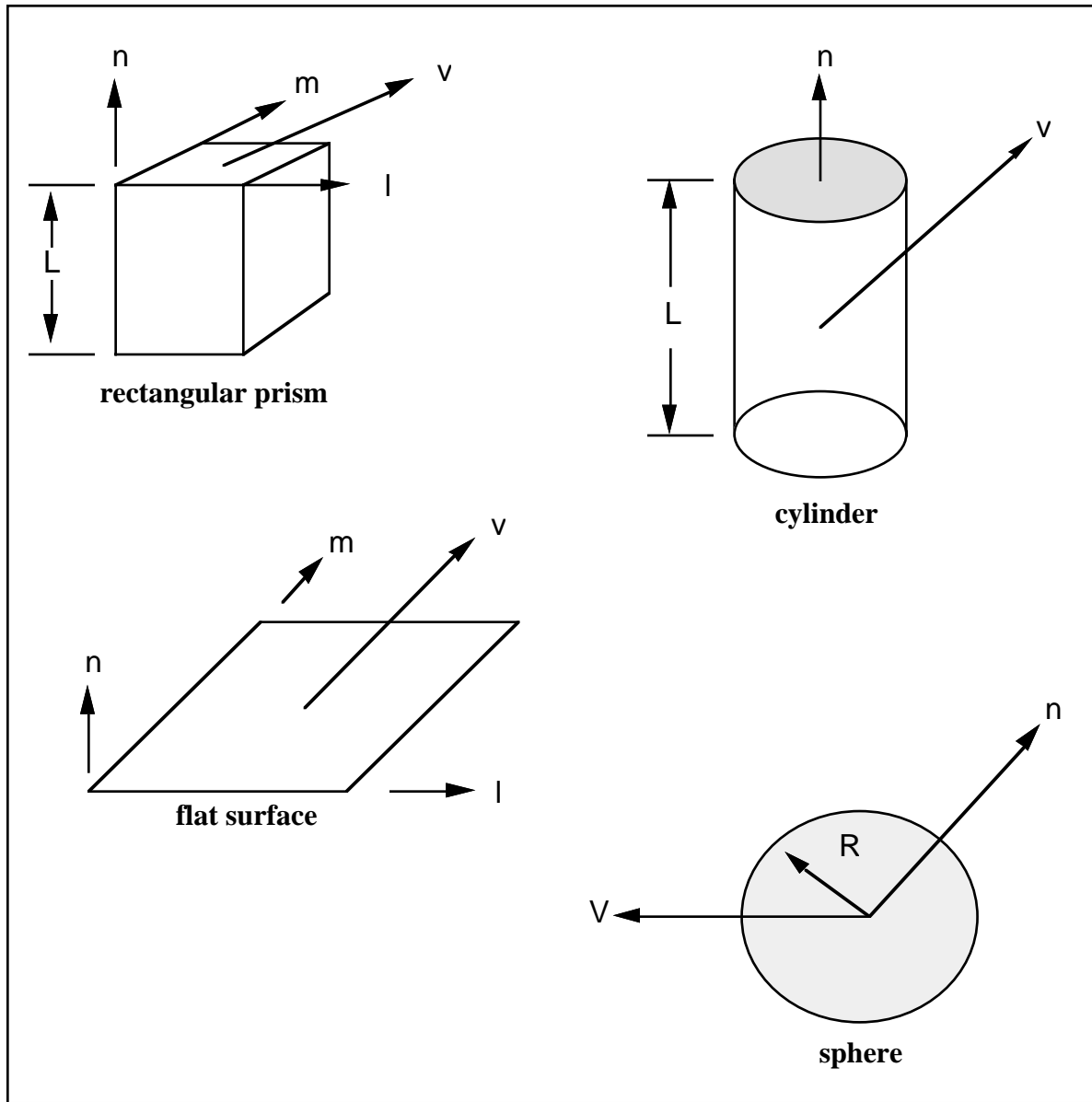
<u>VARIABLE</u>	<u>DESCRIPTION</u>
RADSPH	Radius of sphere

## Optional Card A - Required if MOTION is specified after the keyword.

Optional Card A	1	2	3	4	5	6	7	8
Variable	LCID	OPT	VX	VY	VZ			
Type	I	I	F	F	F			
Default	none	none	none	none	none			

<u>VARIABLE</u>	<u>DESCRIPTION</u>
LCID	Stonewall motion curve number, see *DEFINE_CURVE.
OPT	Type of motion: EQ.0: velocity specified, EQ.1: displacement specified.
VX	x-direction cosine of velocity/displacement vector
VY	y-direction cosine of velocity/displacement vector
VZ	z-direction cosine of velocity/displacement vector

# \*RIGIDWALL



**Figure 23.1.** Vector  $n$  determines the orientation of the generalized stonewalls. For the prescribed motion options the wall can be moved in the direction  $V$  as shown.



# \*RIGIDWALL

---

\*RIGIDWALL\_PLANAR\_{OPTION}\_{OPTION}\_{OPTION}

Available options include:

<BLANK>

**ORTHO**

**FINITE**

**MOVING**

**FORCES**

The ordering of the options in the input below must be observed but the ordering of the options on the command line is unimportant, i.e.; the **ORTHO** card is first, the **FINITE** definition card below must precede the **MOVING** definition card, and the **FORCES** definition card should be last. The **ORTHO** option does not apply if the **MOVING** option is used.

Purpose: Define planar rigid walls with either finite or infinite size (**FINITE**). Orthotropic friction can be defined (**ORTHO**). Also, the plane can possess a mass and an initial velocity (**MOVING**); otherwise, the wall is assumed to be stationary. The **FORCES** option allows the specification of segments on the rigid walls on which the contact forces are computed. In order to achieve a more physical reaction related to the force versus time curve, the SOFT value on the **FORCES** card can be specified.

## Card Format:

- Cards 1 and 2 are required.
- Optional Cards A and B are required if **ORTHO** is specified.
- Optional Card C is required if **FINITE** is specified.
- Optional Card D is required if **MOVING** is specified.
- Optional Card E is required if **FORCES** is specified.



# \*RIGIDWALL

## Card 1 - Required.

Card 1            1            2            3            4            5            6            7            8

Variable	NSID	NSIDEX	BOXID					
Type	I	I	I					
Default	none	0	0					

### VARIABLE

### DESCRIPTION

NSID	Nodal set ID containing slave nodes, see *SET_NODE_OPTION: EQ.0: all nodes are slave to rigid wall.
NSIDEX	Nodal set ID containing nodes that exempted as slave nodes, see *SET_NODE_OPTION.
BOXID	All nodes in box are included as slave nodes to rigid wall, see *DEFINE_BOX.

# \*RIGIDWALL

---

## Card 2 - Required.

Card 2            1            2            3            4            5            6            7            8

Variable	XT	YT	ZT	XH	YH	ZH	FRIC	WVEL
Type	F	F	F	F	F	F	F	F
Default	0.	0.	0.	0.	0.	0.	0.	0.

### VARIABLE

### DESCRIPTION

XT	x-coordinate of tail of any outward drawn normal vector, <b>n</b> , originating on wall (tail) and terminating in space (head), see Figure 23.3.
YT	y-coordinate of tail of normal vector <b>n</b>
ZT	z-coordinate of tail of normal vector <b>n</b>
XH	x-coordinate of head of normal vector <b>n</b>
YH	y-coordinate of head of normal vector <b>n</b>
ZH	z-coordinate of head of normal vector <b>n</b>
FRIC	Interface friction: EQ.0.0: frictionless sliding after contact, EQ.1.0: no sliding after contact, 0.<FRIC<1.: Coulomb friction coefficient. EQ.2.0: node is welded after contact with frictionless sliding. Welding occurs if and only if the normal value of the impact velocity exceeds the critical value specified by WVEL. EQ.3.0: node is welded after contact with no sliding. Welding occurs if and only if the normal value of the impact velocity exceeds the critical value specified by WVEL.
WVEL	Critical normal velocity at which nodes weld to wall (FRIC = 2 or 3).

**Optional Cards A and B - Required if ORTHO is specified after the keyword.**

See Figure 23.2 for the definition of orthotropic friction.

Optional Card A	1	2	3	4	5	6	7	8
Variable	SFRICA	SFRICB	DFRICA	DFRICB	DECAYA	DECAYB		
Type	F	F	F	F	F	F		
Default	0.	0.	0	0	0.	0.		

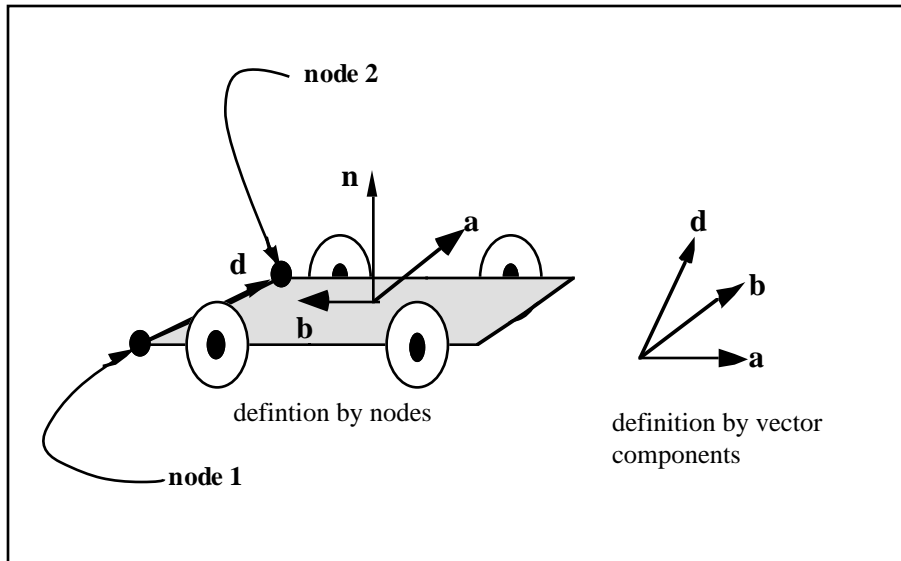
Optional Card B	1	2	3	4	5	6	7	8
Variable	NODE1	NODE2	D1	D2	D3			
Type	I	I	F	F	F			
Default	0.	0.	0	0	0.			

**VARIABLE****DESCRIPTION**

SFRICA	Static friction coefficient in local a-direction, $\mu_{sa}$ , see Figure 23.2
SFRICB	Static friction coefficient in local b-direction, $\mu_{sb}$
DFRICA	Dynamic friction coefficient in local a-direction, $\mu_{ka}$
DFRICB	Dynamic friction coefficient in local b-direction, $\mu_{kb}$
DECAYA	Decay constant in local a-direction, $d_{va}$
DECAYB	Decay constant in local b-direction, $d_{vb}$
NODE1	Node 1, alternative to definition with vector d below, see Figure 23.2. With the node definition the direction changes if the nodal pair rotates.
NODE2	Node 2

# \*RIGIDWALL

VARIABLE	DESCRIPTION
D1	$d_1$ , x-component of vector, alternative to definition with nodes above, see Figure 23.2. This vector is fixed as a function of time.
D2	$d_2$ , y-component of vector
D3	$d_3$ , z-component of vector



**Figure 23.2.** Definition of orthotropic friction vectors. The two methods of defining the vector,  $\mathbf{d}$  are shown. If vector  $\mathbf{d}$  is defined by nodes 1 and 2, the local coordinate system may rotate with the body which contains the nodes; otherwise,  $\mathbf{d}$  is fixed in space, thus on the rigid wall, and the local system is stationary.

## Remarks:

- 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

$$\mu_a = \mu_{ka} + (\mu_{sa} \mu_{ka}) e^{d_{va} V_{relative,a}}$$

$$\mu_b = \mu_{kb} + (\mu_{sb} \mu_{kb}) e^{d_{vb} V_{relative,b}}$$

- Orthotropic rigid walls can be used to model rolling objects on rigid walls where the frictional forces are substantially higher in a direction transverse to the rolling direction. To use this option define a vector  $\mathbf{d}$  to determine the local frictional directions via:

$$\tilde{b} = \tilde{n} \times \tilde{d} \text{ and that } \tilde{a} = \tilde{b} \times \tilde{n}$$

where  $\mathbf{n}$  is the normal vector to the rigid wall. If  $\mathbf{d}$  is in the plane of the rigid wall, then  $\mathbf{a}$  is identical to  $\mathbf{d}$ .

**Optional Card C - Required if FINITE is specified after the keyword.**

See Figure 23.3. The **m** vector is computed as the vector cross product  $\mathbf{m}=\mathbf{n} \times \mathbf{l}$ . The origin, the tail of the normal vector, is taken as the corner point of the finite size plane.

Optional Card C	1	2	3	4	5	6	7	8
Variable	XHEV	YHEV	ZHEV	LENL	LENM			
Type	F	F	F	F	F			
Default	0.	0.	0.	infinity	infinity			

**VARIABLE**

**DESCRIPTION**

XHEV	x-coordinate of head of edge vector <b>l</b> , see Figure 23.3.
YHEV	y-coordinate of head of edge vector <b>l</b>
ZHEV	z-coordinate of head of edge vector <b>l</b>
LENL	Length of <b>l</b> edge
LENM	Length of <b>m</b> edge

# \*RIGIDWALL

---

**Optional Card D - Required if MOVING is specified after keyword.**

**Note:** The MOVING option is not compatible with the ORTHO option.

Optional Card D	1	2	3	4	5	6	7	8
Variable	MASS	V0						
Type	F	F						
Default	none	0.						

---

**VARIABLE**

**DESCRIPTION**

---

MASS

Total mass of stonewall

V0

Initial velocity of stonewall in direction of defining vector, **n**

**Optional Card E - Required if FORCES is specified after the keyword.**

This option allows the force distribution to be monitored on the plane. Also four points can be defined for visualization of the rigid wall. A shell or membrane element must be defined with these four points as the connectivity for viewing in LS-POST.

Optional Card E	1	2	3	4	5	6	7	8
Variable	SOFT	SSID	N <sub>1</sub>	N <sub>2</sub>	N <sub>3</sub>	N <sub>4</sub>		
Type	I	I	I	I	I	I		
Default	0	0	0	0	0	0		
Remarks		1	2					

**VARIABLE****DESCRIPTION**

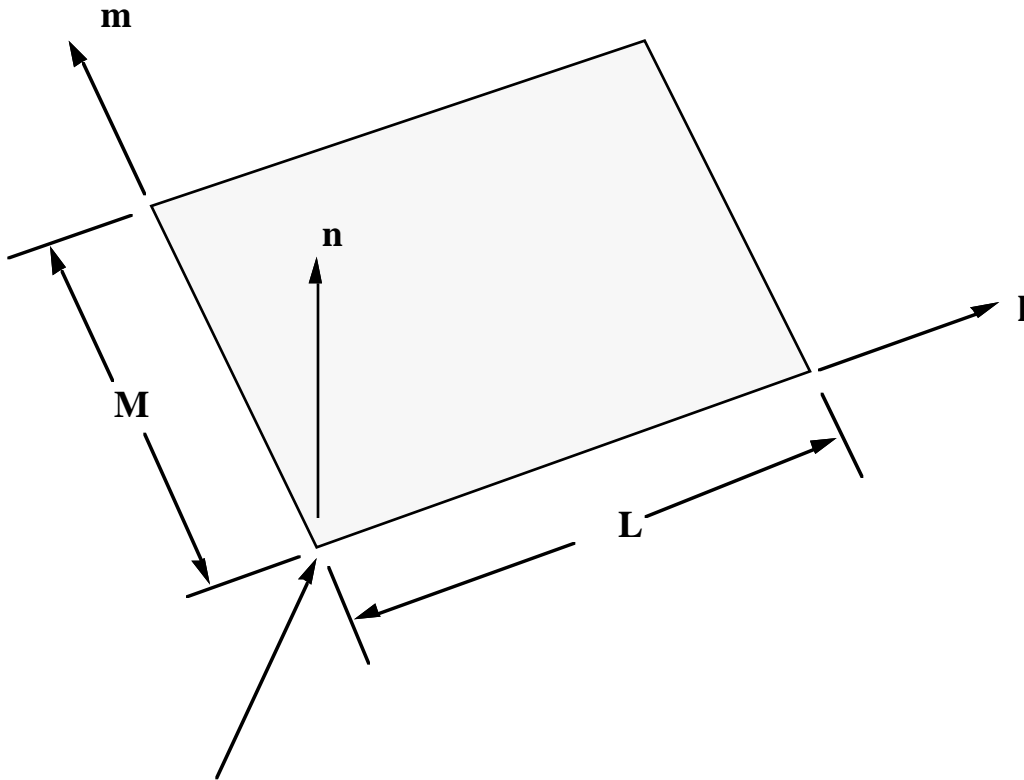
SOFT	Number of cycles to zero relative velocity to reduce force spike
SSID	Segment set identification number for defining areas for force output, see *SET_SEGMENT and remark 1 below.
N1	Optional nodal point for visualization in LS-DYNA database, see remark 2 below.
N2-N4	Optional nodal points for visualization

**Remarks:**

- 1 The segment set defines areas for computing resultant forces. These segments translate with the moving stonewall and allow the forced distribution to be determined. The resultant forces are written in file "RWFORC."
2. These four nodes are for visualizing the movement of the wall, i.e., they move with the wall. To view the wall in LS-POST it is necessary to define a single shell element with these four nodes as its connectivity. The single element must be deformable (non rigid) or else the segment will be treated as a rigid body and the nodes will have their motion modified independently of the stonewall.

## \*RIGIDWALL

---



Tail of normal vector is the origin and corner point if extent of stonewall is finite.

**Figure 23.3.** Vector  $\mathbf{n}$  is normal to the stonewall. An optional vector  $\mathbf{l}$  can be defined such that  $\mathbf{m} = \mathbf{n} \times \mathbf{l}$ . The extent of the stonewall is limited by defining  $\mathbf{L}$  (LENL) and  $\mathbf{M}$  (LENM). A zero value for either of these lengths indicates that the stonewall is infinite in that direction.



```

$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$ *RIGIDWALL_PLANAR_MOVING_FORCES
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$ Define a moving planar rigid wall:
$ - that is parallel to the y-z plane starting at x = 250 mm
$ - with an initial velocity of 8.94 mm/ms in the negative z-direction
$ - that has a mass of 800 kg
$ - which prevents all nodes in the model from penetrating the wall
$ - with a friction coefficient for nodes sliding along the wall of 0.1
$ - track the motion of the wall by creating a node (numbered 99999)
$ at the tail of the wall and assigning the node to move with the wall
$
*$RIGIDWALL_PLANAR_MOVING_FORCES
$.>...1.>...2.>...3.>...4.>...5.>...6.>...7.>...8
$ nsid nsidex boxid
$ 0 0 0
$
$ xt yt zt xh yh zh fric
$ 250.0 0.0 0.0 0.0 0.0 0.0 0.1
$
$ SW mass SW vel
$ 800.00 8.94
$
$ soft ssid node1 node2 node3 node4
$ 0 0 99999
$
$
*$NODE
$.>...1.>...2.>...3.>...4.>...5.>...6.>...7.>...8
$ nid x y z tc rc
$ 99999 250.0 0.0 0.0 0 0
$
$
*$DATABASE_HISTORY_NODE
$ Define nodes that output into nodout
$ id1 id2 id3
$.>...1.>...2.>...3.>...4.>...5.>...6.>...7.>...8
$ 99999
$
*$DATABASE_NODOUT
$ dt
$ 0.1
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$

```

# \*RIGIDWALL

---

# **\*SECTION**

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

**\*SECTION\_BEAM**  
**\*SECTION\_DISCRETE**  
**\*SECTION\_SEATBELT**  
**\*SECTION\_SHELL\_{OPTION}**  
**\*SECTION\_SOLID\_{OPTION}**  
**\*SECTION\_SPH**  
**\*SECTION\_TSHELL**

The location and order of these cards in the input file are arbitrary.

# \*SECTION

---

## \*SECTION\_BEAM

Purpose: Define cross sectional properties for beam, truss, discrete beam, and cable elements.

### Card Format

Card 1            1            2            3            4            5            6            7            8

Variable	SECID	ELFORM	SHRF	QR/IRID	CST	SCoor		
Type	I	I	F	F	F	F		
Default	none	1	1.0	2.0	0.0	0.0		

Card 2

Integrated spotweld	TS1	TS2	TT1	TT2	NSLOC	NTLOC		
Resultant	A	ISS	ITT	IRR	SA			
Discrete	VOL	INER	CID	CA	OFFSET	RRCON	SRCON	TRCON
Type	F	F	F	F	F	F	F	F

<u>VARIABLE</u>	<u>DESCRIPTION</u>
SECID	Section ID. SECID is referenced on the *PART card and must be unique.
ELFORM	Element formulation options: EQ.1: Hughes-Liu with cross section integration (default), EQ.2: Belytschko-Schwer resultant beam (resultant), EQ.3: truss (resultant), EQ.4: Belytschko-Schwer full cross-section integration, EQ.5: Belytschko-Schwer tubular beam with cross-section integration, EQ.6: discrete beam/cable, EQ.7: 2D plane strain shell element (xy plane), EQ.8: 2D axisymmetric volume weighted shell element (xy plane), EQ.9: spotweld beam, see *MAT_SPOTWELD.

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
	Note that the 2D and 3D element types must not be mixed, and different types of 2D elements must not be used together. For example, the plane strain element type must not be used with the axisymmetric element type. In 3D the different beam elements types, i.e., 1-6 and 9 can be freely mixed together.
SHRF	Shear factor. This factor is not needed for truss, resultant beam, discrete beam, and cable elements. The recommended value for rectangular sections is 5/6, the default is 1.0.
QR/IRID	Quadrature rule or rule number for user defined rule for integrated beams: EQ.1.0: one integration point, EQ.2.0: 2×2 Gauss quadrature (default beam), EQ.3.0: 3×3 Gauss quadrature, EQ.4.0: 3×3 Lobatto quadrature, EQ.5.0: 4×4 Gauss quadrature EQ.-n: where  n  is the number of the user defined rule. IRID integration rule n is defined using *INTEGRATION_BEAM card.
CST	Cross section type, not needed for truss, resultant beam, discrete beam, and cable elements: EQ.0.0: rectangular, EQ.1.0: tubular, EQ.2.0: arbitrary (user defined integration rule).
SCoor	Location of triad for tracking the rotation of the discrete beam element. The force and moment resultants in the output databases are referenced to this triad. 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.
TS1	Beam thickness (CST=0.0, 2.0) or outer diameter (CST = 1.0) in s direction at node n <sub>1</sub> . Note that the thickness defined on the *ELEMENT_BEAM_THICKNESS card overrides the definition give here.
TS2	Beam thickness (CST=0.0, 2.0) or outer diameter (CST = 1.0) in s direction at node n <sub>2</sub> .
TT1	Beam thickness (CST=0.0, 2.0) or inner diameter (CST = 1.0) in t direction at node n <sub>1</sub> .
TT2	Beam thickness (CST=0.0, 2.0) or inner diameter (CST = 1.0) in t direction at node n <sub>2</sub> .
NSLOC	Location of reference surface normal to s axis for Hughes-Liu beam elements only: EQ.1.0: side at s=1.0, EQ.0.0: center, EQ.-1.0: side at s = -1.0.

---

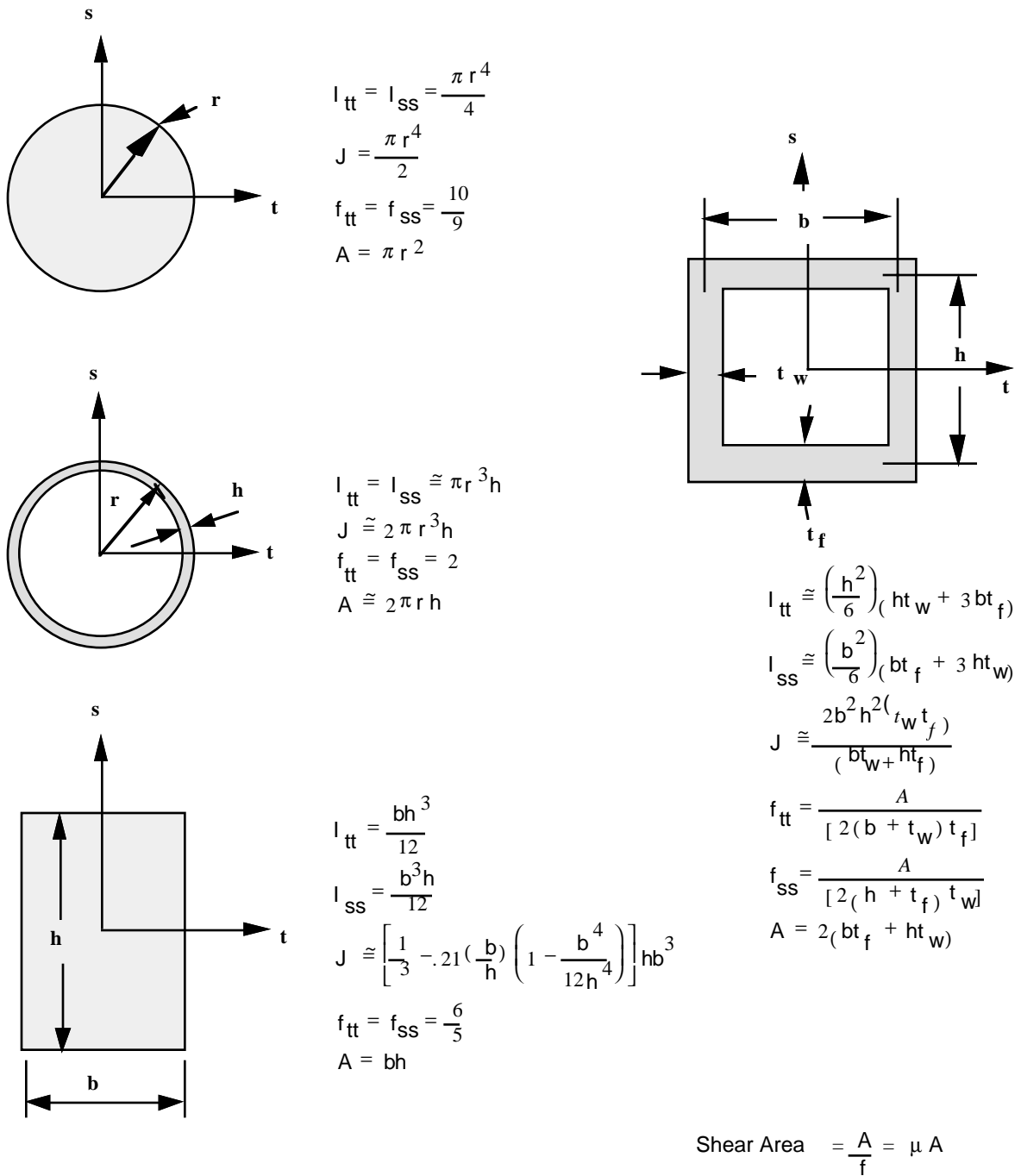
## \*SECTION

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
NTLOC	Location of reference surface normal to t axis for Hughes-Liu beam elements only: EQ.1.0: side at $t = 1.0$ , EQ.0.0: center, EQ.-1.0: side at $t = -1.0$ .
A	Cross-sectional area. The definition on *ELEMENT_BEAM_THICKNESS overrides the value defined here, see Figure 24.1.
ISS	$I_{ss}$ . The definition on *ELEMENT_BEAM_THICKNESS overrides the value defined here, see Figure 24.1.
ITT	$I_{tt}$ . The definition on *ELEMENT_BEAM_THICKNESS overrides the value defined here, see Figure 24.1.
IRR	$I_{rr}$ (J) polar inertia. The definition on *ELEMENT_BEAM_THICKNESS overrides the value defined here, see Figure 24.1.
SA	Shear area. The definition on *ELEMENT_BEAM_THICKNESS overrides the value defined here, see Figure 24.1.
VOL	Volume of discrete beam.
INER	I, lumped inertia of discrete beam which have six degrees of freedom. This lumped inertia is partitioned to the two nodes of the beam element.
CID	Coordinate system ID for orientation, materials type ID (67-69), see *DEFINE_COORDINATE_SYSTEM. This is not defined for cable elements.
CA	Cable area, materials type ID 71, *MAT_CABLE.
OFFSET	Offset for cable. For a definition see materials type ID 71, *MAT_CABLE.
RRCON	r-rotational constraint for local coordinate system EQ.0.0: Coordinate ID rotates about r axis with nodes. EQ.1.0: Rotation is constrained about the r-axis
SRCON	s-rotational constraint for local coordinate system EQ.0.0: Coordinate ID rotates about s axis with nodes. EQ.1.0: Rotation is constrained about the s-axis
TRCON	t-rotational constraint for local coordinate system EQ.0.0: Coordinate ID rotates about t axis with nodes. EQ.1.0: Rotation is constrained about the t-axis

### **Remarks:**

1. For the truss element, define the cross-sectional area, A, only.
2. 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 rotational motions.



**Figure 24.1.** Properties of beam cross section for several common cross sections.

# \*SECTION

---

```

$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$ $ *SECTION_BEAM
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$ Define a Belytschko-Schwer resultant beam (elform = 2) with the following
$ properties. This beam models the connection/stiffening beams of a medium
$ size roadside sign.
$
$ cross sectional area: a = 515.6 mm2
$ 2nd moment of area about s-axis: iss = 99,660.0 mm4
$ 2nd moment of area about t-axis: iss = 70,500.0 mm4
$ 2nd polar moment of area about beam axis: irr = 170,000.0 mm4
$
*SECTION_BEAM
$
$.>...1.>...2.>...3.>...4.>...5.>...6.>...7.>...8
$ sid elform shrf qr/irid cst
$ 111 2
$
$ a iss itt irr sa
$ 515.6 99660.0 70500.0 170000.0
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$

```



**\*SECTION\_DISCRETE**

Purpose: Defined spring and damper elements for translation and rotation. These definitions must correspond with the material type selection for the elements, i.e., \*MAT\_SPRING\_... and \*MAT\_DAMPER\_...

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	SECID	DRO	KD	V0	CL	FD		
Type	I	I	F	F	F	F		

Card 2            1            2            3            4            5            6            7            8

Variable	CDL	TDL						
Type	F	F						

---

**VARIABLE**

---

**DESCRIPTION**

SECID	Section ID. SECID is referenced on the *PART card and must be unique.
DRO	Displacement/Rotation Option: EQ.0: the material describes a translational spring/damper, EQ.1: the material describes a torsional spring/damper.
KD	Dynamic magnification factor. See remarks 1 and 2 below.
V0	Test velocity
CL	Clearance. See remark 3 below.
FD	Failure deflection (twist for DRO=1).
CDL	Deflection (twist for DRO=1) limit in compression. See remark 4 below.
TDL	Deflection (twist for DRO=1) limit in tension. See remark 4 below.

## \*SECTION

---

### Remarks:

1. The constants from KD to TDL are optional and do not need to be defined.
2. If  $k_d$  is nonzero, the forces computed from the spring elements are assumed to be the static values and are scaled by an amplification factor to obtain the dynamic value:

$$F_{dynamic} = \left( 1 + k_d \frac{V}{V_0} \right) F_{static}$$

where

$V$  = absolute value of the relative velocity between the nodes.

$V_0$  = dynamic test velocity.

For example, if it is known that a component shows a dynamic crush force at 15m/s equal to 2.5 times the static crush force, use  $k_d=1.5$  and  $V_0=15$ .

3. Here, “clearance” defines a compressive displacement which the spring sustains before beginning the force-displacement relation given by the load curve defined in the material selection. If a non-zero clearance is defined, the spring is compressive only.
4. The deflection limit in compression and tension is restricted in its application to no more than one spring per node subject to this limit, and to deformable bodies only. For example in the former case, if three springs are in series, either the center spring or the two end springs may be subject to a limit, but not all three. When the limiting deflection is reached, momentum conservation calculations are performed and a common acceleration is computed in the appropriate direction. An error termination will occur if a rigid body node is used in a spring definition where deflection is limited.



**\*SECTION**

**\*SECTION\_SEATBELT**

Purpose: Define section properties for the seat belt elements. This card is required for the \*PART Section. Currently, only the ID is required.

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	SECID								
Type	I								

**VARIABLE**

**DESCRIPTION**

SECID

Section ID

```
$  
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$  
$  
$$$ *SECTION_SEATBELT  
$  
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$  
$  
$ Define a seat belt section that is referenced by part 10. Nothing  
$ more than the sid is required.  
$  
*SECTION_SEATBELT  
$  
$...>...1...>...2...>...3...>...4...>...5...>...6...>...7...>...8  
$ sid  
 111  
$  
$  
*PART  
Seatbelt material  
$...>...1...>...2...>...3...>...4...>...5...>...6...>...7...>...8  
$ pid sid mid eosid hgid adpopt  
 10 111 220  
$  
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$  
$
```

**\*SECTION\_SHELL\_{OPTION}**

Options include:

**<BLANK>**  
**ALE**

such that the keyword cards appear:

**\*SECTION\_SHELL**  
**\*SECTION\_SHELL\_ALE**

Purpose: Define section properties for shell elements.

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	SECID	ELFORM	SHRF	NIP	PROPT	QR/IRID	ICOMP	SETYP
Type	I	I	F	F	F	F	I	I
Default	none	2	1.0	2	0.0	0.0	0	1

Card 2            1            2            3            4            5            6            7            8

Variable	T1	T2	T3	T4	NLOC			
Type	F	F	F	F	F			
Default	0.0	0.0	0.0	0.0	0.0			

# \*SECTION

---

**Optional Section Cards if ICOMP=1. Define NIP angles putting 8 on each card.**

Cards 3,4,..      1            2            3            4            5            6            7            8

Variable	B1	B2	B3	B4	B5	B6	B7	B8
Type	F	F	F	F	F	F	F	F

**Optional Section Card for ALE option.**

Also see \*CONTROL\_ALE and \*ALE\_SMOOTHING.

Card 3            1            2            3            4            5            6            7            8

Variable	AFAC	BFAC	CFAC	DFAC	EFAC	START	END	AAFAC
Type	F	F	F	F	F	F	F	F

<u>VARIABLE</u>	<u>DESCRIPTION</u>
SECID	Section ID. SECID is referenced on the *PART card and must be unique.
ELFORM	Element formulation options: EQ.1: Hughes-Liu, EQ.2: Belytschko-Tsay (default), EQ.3: BCIZ triangular shell, EQ.4: C <sub>0</sub> triangular shell, EQ.5: Belytschko-Tsay membrane, EQ.6: S/R Hughes-Liu , EQ.7: S/R co-rotational Hughes-Liu, EQ.8: Belytschko-Leviathan shell , EQ.9: Fully integrated Belytschko-Tsay membrane, EQ.10: Belytschko-Wong-Chiang, EQ.11: Fast (co-rotational) Hughes-Liu, EQ.12: Plane stress (x-y plane) , EQ.13: Plane strain (x-y plane) EQ.14: Axisymmetric solid (y-axis of symmetry) - area weighted, EQ.15: Axisymmetric solid (y-axis of symmetry) - volume weighted, EQ.16: Fully integrated shell element (very fast).

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
	Note that the 2D and 3D element types must not be mixed, and different types of 2D elements must not be used together. For example, 2D axisymmetric calculations can use either element types 14 or 15 but these element types must not be mixed together. Likewise, the plane strain element type must not be used with either the plane stress element or the axisymmetric element types. In 3D, the different shell elements types, i.e., 1-11 and 16, can be freely mixed together.
SHRF	Shear factor which scales the transverse shear stress. A suggested value is 5/6.
NIP	Number of through shell thickness integration points:  EQ.0: set to 2 integration points for shell elements. Through thickness integration for the two-dimensional elements (options 11-15 above) is not meaningful; consequently, the default is equal to 1 integration point. Fully integrated two-dimensional elements are available for options 13 and 15 by setting NIP equal to a value of 4 corresponding to a 2 by 2 Gaussian quadrature.
PROPT	Printout option: EQ.1.0: average resultants and fiber lengths, EQ.2.0: resultants at plan points and fiber lengths, EQ.3.0: resultants, stresses at all points, fiber lengths.
QR/IRID	Quadrature rule or Integration rule ID, see *INTEGRATION_SHELL: LT.0.0: absolute value is specified rule number, EQ.0.0: Gauss (up to five points are permitted), EQ.1.0: trapezoidal, <i>not recommend for accuracy reasons</i> .
ICOMP	Flag for orthotropic/anisotropic layered composite material model. This option applies to material types 22, 23, 33, 34, 36,41-50, 54-56, 58, 59, and 103: EQ.1: a material angle in degrees is defined for each through thickness integration point. Thus, each layer has one integration point.
SETYP	2D solid element type: Defined for ELFORM 13, 14, and 15. EQ.1: Lagrangian EQ.2: Eulerian (single material with voids) EQ.3: ALE
T1	Shell thickness at node $n_1$ , unless the thickness is defined on the *ELEMENT_SHELL_OPTION card.
T2	Shell thickness at node $n_2$ , see comment for T1 above.
T3	Shell thickness at node $n_3$ , see comment for T1 above.
T4	Shell thickness at node $n_4$ , see comment for T1 above.

---

# \*SECTION

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
NLOC	Location of reference surface (Hughes-Liu shell only): EQ.1.0: top surface, EQ.0.0: mid surface (default ), EQ.-1.0: bottom surface.
B1	$\beta_1$ , material angle at first integration point
B2	$\beta_2$ , material angle at second integration point
B3	$\beta_3$ , material angle at third integration point
.	.
.	.
.	.
B8	$\beta_8$ , material angle at eighth integration point
.	.
Bnip	$\beta_{nip}$ , material angle at niph integration point
AFAC	Smoothing weight factor - Simple average: EQ.-1: turn smoothing off.
BFAC	Smoothing weight factor - Volume weighting
CFAC	Smoothing weight factor - Isoparametric
DFAC	Smoothing weight factor - Equipotential
EFAC	Smoothing weight factor - Equilibrium
START	Start time for smoothing
END	End time for smoothing
AAFAC	ALE advection factor



```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$  *SECTION_SHELL
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$ Define a shell section that specifies the following:
$   elform = 10  Belytschko-Wong-Chiang shell element formulation.
$   nip = 3     Three through the shell thickness integration points.
$   t1 - t4 = 2.0 A shell thickness of 2 mm at all nodes.
$
*SECTION_SHELL
$
$...>...1...>...2...>...3...>...4...>...5...>...6...>...7...>...8
$   sid   elform   shrf      nip   propt  qr/irid   icode
$       1      10           3.0000
$
$   t1      t2      t3      t4      nloc
$   2.0     2.0     2.0     2.0
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

# \*SECTION

---

\*SECTION\_SOLID\_{OPTION}

Options include:

<BLANK>  
ALE

such that the keyword cards appear:

\*SECTION\_SOLID  
\*SECTION\_SOLID\_ALE

Purpose: Define section properties for solid continuum and fluid elements.

## Card 1 define for all options

Card 1            1            2            3            4            5            6            7            8

Variable	SECID	ELFORM	AET					
Type	I	I	I					

## Card 2 define only for the ALE option.

Also see \*ALE\_SMOOTHING for the smoothing definition.

Cards 2            1            2            3            4            5            6            7            8

Variable	AFAC	BFAC	CFAC	DFAC	START	END	AAFAC	
Type	F	F	F	F	F	F	F	

---

<b>VARIABLE</b>	<b>DESCRIPTION</b>
SECID	Section ID. SECID is referenced on the *PART card and must be unique.
ELFORM	Element formulation options: EQ.0: 1 point corotational for *MAT_MODIFIED_HONEYCOMB EQ.1: constant stress solid element (default), EQ.2: fully integrated S/R solid, EQ.3: fully integrated quadratic 8 node element with nodal rotations, EQ.4: S/R quadratic tetrahedron element with nodal rotations, EQ.5: 1 point ALE, EQ.6: 1 point Eulerian, EQ.7: 1 point Eulerian ambient, EQ.8: acoustic, EQ.9: 1 point corotational for *MAT_MODIFIED_HONEYCOMB EQ.10: 1 point tetrahedron. EQ.11: 1 point ALE multi-material element EQ.12: 1 point integration with single material and void.
AET	Ambient Element type: Can be defined for ELFORM 7, 11 and 12. EQ.1: temperature (not currently available), EQ.2: pressure and temperature (not currently available), EQ.3: pressure outflow, EQ.4: pressure inflow.(Default for ELFORM 7)
AFAC	Smoothing weight factor - Simple average: EQ.-1: turn smoothing off.
BFAC	Smoothing weight factor - Volume weighting
CFAC	Smoothing weight factor - Isoparametric
DFAC	Smoothing weight factor - Equipotential
START	Start time for smoothing
END	End time for smoothing
AAFAC	ALE advection factor

# \*SECTION

---

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$ *SECTION_SOLID
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$ A bolt modeled with solids was found to have excessive hourglassing.
$ Thus, the section (sid = 116) associated with the bolt part was used
$ to specify that a fully integrated Selectively-Reduced solid element
$ formulation be used to totally eliminate the hourglassing (elform = 2).
$
*SECTION_SOLID
$.>...1.>...2.>...3.>...4.>...5.>...6.>...7.>...8
$   sid   elform
$   116   2
$
*PART
bolts
$   pid   sid   mid   eosid   hgid   adpopt
$   17    116   5
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

**\*SECTION\_SPH**

Purpose: Define section properties for SPH particles.

**Card Format**

Card 1            1            2            3            4            5            6            7            8

Variable	SECID	CSLH						
Type	I	F						
Default	none	1.2						

**VARIABLE****DESCRIPTION**

SECID            Section ID. SECID is referenced on the \*PART card and must be unique.

CSLH            Constant applied to the smoothing length of the particles

# \*SECTION

---

## \*SECTION\_TSHELL

Purpose: Define section properties for thick shell elements.

### Card Format

Card 1            1            2            3            4            5            6            7            8

Variable	SECID	ELFORM	SHRF	NIP	PROPT	QR	ICOMP	
Type	I	I	F	F	F	F	I	
Default	none	1	1.0	2	1	0	0	

**Optional Section Cards if ICOMP=1 define NIP angles putting 8 on each card.**

Cards 2,3,..    1            2            3            4            5            6            7            8

Variable	B1	B2	B3	B4	B5	B6	B7	B8
Type	F	F	F	F	F	F	F	F

<u>VARIABLE</u>	<u>DESCRIPTION</u>
SECID	Section ID. SECID is referenced on the *PART card and must be unique.
ELFORM	Element formulation: EQ.1: one point reduced integration (default), EQ.2: selective reduced $2 \times 2$ in plane integration.
SHRF	Shear factor. A value of 5/6 is recommended.
NIP	Number of through shell thickness integration points: EQ.0: set to 2 integration points.
PROPT	Printout option: EQ.1.0: average resultants and fiber lengths, EQ.2.0: resultants at plan points and fiber lengths, EQ.3.0: resultants, stresses at all points, fiber lengths.

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
QR	Quadrature rule: LT.0.0: absolute value is specified rule number, EQ.0.0: Gauss (up to five points are permitted), EQ.1.0: trapezoidal, not recommended for accuracy reasons.
ICOMP	Flag for layered composite material mode: EQ.1: a material angle is defined for each through thickness integration point . For each layer one integration point is used.
B1	$\beta_1$ , material angle at first integration point. The same procedure for determining material directions is use for thick shells that is used for the 4 node quadrilateral shell.
B2	$\beta_2$ , material angle at second integration point
B3	$\beta_3$ , material angle at third integration point
.	.
.	.
.	.
B8	$\beta_8$ , material angle at eighth integration point
.	.
Bnip	$\beta_{nip}$ , material angle at niph integration point

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

# \*SECTION

---



# **\*SET**

The keyword **\*SET** provides a convenient way of defining groups of nodes, parts, elements, and segments. The sets can be used in the definitions of contact interfaces, loading conditions, boundary conditions, and other inputs. Each set type must have a unique numeric identification. The keyword control cards in this section are defined in alphabetical order:

**\*SET\_BEAM\_OPTION**

**\*SET\_DISCRETE\_{OPTION}**

**\*SET\_NODE\_OPTION**

**\*SET\_PART\_OPTION**

**\*SET\_SEGMENT**

**\*SET\_SHELL\_OPTION**

**\*SET\_SOLID\_{OPTION}**

**\*SET\_TSHELL\_{OPTION}**

# \*SET

---

## \*SET\_BEAM\_OPTION

Available options include:

### GENERATE

The last option, GENERATE, will generate a block of beam element ID's between a starting ID and an ending ID. An arbitrary number of blocks can be specified to define the set.

Purpose: Define a set of beam elements.

### Card Format

Card 1            1            2            3            4            5            6            7            8

Variable	SID							
Type	I							
Default	none							

**Cards 2, 3, 4, ... (OPTION=none) (The next "\*" card terminates the input.)**

1            2            3            4            5            6            7            8

Variable	K1	K2	K3	K4	K5	K6	K7	K8
Type	I	I	I	I	I	I	I	I

**Cards 2, 3, 4, ... (OPTION=GENERATE) (The next "\*" card terminates the input.)**

1            2            3            4            5            6            7            8

Variable	B1BEG	B1END	B2BEG	B2END	B3BEG	B3END	B4BEG	B4END
Type	I	I	I	I	I	I	I	I

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
SID	Set ID
K1	First beam element
K2	Second beam element
.	.
.	.
KNUM	Last beam element
BNBEG	First beam element ID in block N.
BNEND	Last beam element ID in block N. All defined ID's between and including BNBEG to BNEND are added to the set. These sets are generated after all input is read so that gaps in the element numbering are not a problem. BNBEG and BNEND may simply be limits on the ID's and not element ID's.

# \*SET

---

## \*SET\_DISCRETE\_{OPTION}

Available options include:

<BLANK>  
GENERATE

The last option, GENERATE, will generate a block of discrete element ID's between a starting ID and an ending ID. An arbitrary number of blocks can be specified to define the set.

Purpose: Define a set of discrete elements.

### Card Format

Card 1            1            2            3            4            5            6            7            8

Variable	SID							
Type	I							
Default	none							

### Cards 2, 3, 4, ... (OPTION=none) (The next "\*" card terminates the input.)

                  1            2            3            4            5            6            7            8

Variable	K1	K2	K3	K4	K5	K6	K7	K8
Type	I	I	I	I	I	I	I	I

### Cards 2, 3, 4, ... (OPTION=GENERATE) (The next "\*" card terminates the input.)

                  1            2            3            4            5            6            7            8

Variable	B1BEG	B1END	B2BEG	B2END	B3BEG	B3END	B4BEG	B4END
Type	I	I	I	I	I	I	I	I

---

<b>VARIABLE</b>	<b>DESCRIPTION</b>
SID	Set ID
K1	First discrete element
K2	Second discrete element
.	.
.	.
KNUM	Last discrete element
BNBEG	First discrete element ID in block N.
BNEND	Last discrete element ID in block N. All defined ID's between and including BNBEG to BNEND are added to the set. These sets are generated after all input is read so that gaps in the element numbering are not a problem. BNBEG and BNEND may simply be limits on the ID's and not element ID's.

# \*SET

---

## \*SET\_NODE\_OPTION

Available options include:

**LIST**  
**COLUMN**  
**LIST\_GENERATE**

The last option, LIST\_GENERATE, will generate a block of node ID's between a starting nodal ID number and an ending nodal ID number. An arbitrary number of blocks can be specified to define the set.

Purpose: Define a nodal set with some identical or unique attributes.

### Card Format

	1	2	3	4	5	6	7	8
Variable	SID	DA1	DA2	DA3	DA4			
Type	I	F	F	F	F			
Default	none	0.	0.	0.	0.			
Remark		1	1	1	1			

**Cards 2, 3, 4, ... (OPTION=LIST) (The next "\*" card terminates the input.)**

	1	2	3	4	5	6	7	8
Variable	NID1	NID2	NID3	NID4	NID5	NID6	NID7	NID8
Type	I	I	I	I	I	I	I	I

**Cards 2, 3, 4, ... (OPTION=COLUMN) (The next “\*” card terminates the input.)**

	1	2	3	4	5	6	7	8
Variable	NID	A1	A2	A3	A4			
Type	I	F	F	F	F			
Remark		2	2	2	2			

**Cards 2, 3, 4, ... (OPTION=LIST\_GENERATE) (The next “\*” card terminates the input.)**

	1	2	3	4	5	6	7	8
Variable	B1BEG	B1END	B2BEG	B2END	B3BEG	B3END	B4BEG	B4END
Type	I	I	I	I	I	I	I	I

**VARIABLE**

**DESCRIPTION**

SID	Set identification. All node sets should have a unique set ID.
DA1	First nodal attribute default value, see remark 1 below.
DA2	Second nodal attribute default value
DA3	Third nodal attribute default value
DA4	Fourth nodal attribute default value
NIDN	Node ID n
NID	Nodal ID
A1	First nodal attribute, see remark 2 below.
A2	Second nodal attribute
A3	Third nodal attribute
A4	Fourth nodal attribute

# \*SET

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
BNBEG	First node ID in block N.
BNEND	Last node ID in block N. All defined ID's between and including BNBEG to BNEND are added to the set. These sets are generated after all input is read so that gaps in the node numbering are not a problem. BNBEG and BNEND may simply be limits on the ID's and not nodal ID's.

## **Remarks:**

1. Nodal attributes can be assigned for some input types. For example, for contact option, \*CONTACT\_TIEBREAK\_NODES\_TO\_SURFACE the attributes are:  
  
DA1=NFLF     Normal failure force,  
DA2=NSFLF    Shear failure force,  
DA3=NNEN     Exponent for normal force,  
DA4=NMES     Exponent for shear force.
2. The default nodal attributes can be overridden on these cards; otherwise, A1=DA1, etc.



**\*SET\_PART\_OPTION**

Available options include:

- LIST**
- COLUMN**
- LIST\_GENERATE**

The last option will generate a block of part ID's between a starting part ID number and an ending part ID number. An arbitrary number of blocks can be specified to define the part set.

Purpose: Define a set of parts with optional attributes. For the column option, see \*AIRBAG or \*CONSTRAINED\_RIGID\_BODY\_STOPPERS.

**Card Format**

	1	2	3	4	5	6	7	8
Variable	SID	DA1	DA2	DA3	DA4			
Type	I	F	F	F	F			
Default	none	0.						
Remark		1	1	1	1			

**Card 2, 3, 4, ... (OPTION=LIST) (The next "\*" card terminates the input.)**

	1	2	3	4	5	6	7	8
Variable	PID1	PID2	PID3	PID4	PID5	PID6	PID7	PID8
Type	I	I	I	I	I	I	I	I

# \*SET

---

Card 2, 3, 4, ... (*OPTION=COLUMN*) (The next “\*” card terminates the input.)

	1	2	3	4	5	6	7	8
Variable	PID	A1	A2	A3	A4			
Type	I	F	F	F	F			
Remark		1	1	1	1			

Cards 2, 3, 4, ... (*OPTION=LIST\_GENERATE*) (The next “\*” card terminates the input.)

	1	2	3	4	5	6	7	8
Variable	B1BEG	B1END	B2BEG	B2END	B3BEG	B3END	B4BEG	B4END
Type	I	I	I	I	I	I	I	I

<u>VARIABLE</u>	<u>DESCRIPTION</u>
SID	Set ID. All part sets should have a unique set ID.
DA1	First attribute default value, see remark 1 below.
DA2	Second attribute default value
DA3	Third attribute default value
DA4	Fourth attribute default value
PID	Part ID
PID1	First part ID
PID2	Second part ID
.	.
A1	First part attribute, see remark 2 below.
A2	Second part attribute
A3	Third part attribute

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
A4	Fourth part attribute
BNBEG	First part ID in block N.
BNEND	Last part ID in block N. All defined ID's between and including BNBEG to BNEND are added to the set. These sets are generated after all input is read so that gaps in the part numbering are not a problem. BNBEG and BNEND may simply be limits on the ID's and not part ID's.

**Remarks:**

1. Part attributes can be assigned for some input types. For example, for airbags a time delay, DA1=T1, can be defined before pressure begins to act along with a time delay, DA2=T2, before full pressure is applied, (default T2=T1), and for the constraint option, \*CONSTRAINED\_RIGID\_BODY\_STOPPERS one attribute can be defined: DA1, the closure distance which activates the stopper constraint.
2. The default part attributes can be overridden on the part cards; otherwise, A1=DA1, etc.

# \*SET

---

## \*SET\_SEGMENT

Purpose: Define a set of quadrilateral and triangular segments with optional identical or unique attributes.

### Card Format

	1	2	3	4	5	6	7	8
Variable	SID	DA1	DA2	DA3	DA4			
Type	I	F	F	F	F			
Default	none	0.	0.	0.	0.			
Remarks		1	1	1	1			

Cards 2, 3, 4, ... (The next “\*” card terminates the input.)

	1	2	3	4	5	6	7	8
Variable	N1	N2	N3	N4	A1	A2	A3	A4
Type	I	I	I	I	F	F	F	F
Remarks				2	3	3	3	3

---

### VARIABLE

### DESCRIPTION

---

SID	Set ID. All segment sets should have a unique set ID.
DA1	First segment attribute default value, see remark 1 below.
DA2	Second segment attribute default value
DA3	Third segment attribute default value
DA4	Fourth segment attribute default value

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
N1	Nodal point n <sub>1</sub>
N2	Nodal point n <sub>2</sub>
N3	Nodal point n <sub>3</sub>
N4	Nodal point n <sub>4</sub> , see remark 2 below.
A1	First segment attribute, see remark 3 below.
A2	Second segment attribute
A3	Third segment attribute
A4	Fourth segment attribute
NFLS	Normal failure stress
SFLS	Shear failure stress. Failure criterion:

**Remarks:**

1. Segment attributes can be assigned for some input types. For example, for the contact options, the attributes for the SLAVE surface are:

DA1=NFLS Normal failure stress, \*CONTACT\_TIEBREAK\_SURFACE\_contact only,

DA2=SFLS Shear failure stress, \*CONTACT\_TIEBREAK\_SURFACE\_contact only,

DA3=FSF Coulomb friction scale factor,

DA4=VSF Viscous friction scale factor,

and the attributes for the MASTER surface are:

DA1=FSF Coulomb friction scale factor,

DA2=VSF Viscous friction scale factor.

For airbags, see \*AIRBAG, a time delay, DA1=T1, can be defined before pressure begins to act on a segment along with a time delay, DA2=T2, before full pressure is applied to the segment, (default T2=T1), and for the constraint option,

2. To define a triangular segment make n<sub>4</sub> equal to n<sub>3</sub>.
3. The default segment attributes can be overridden on these cards, otherwise, A1=DA1, etc.

# \*SET

---

## \*SET\_SHELL\_OPTION

Available options include:

**LIST**  
**COLUMN**  
**LIST\_GENERATE**

The last option will generate a block of shell ID's between a starting shell ID number and an ending ID number. An arbitrary number of blocks can be specified to define the shell set.

Purpose: Define a set of shell elements with optional identical or unique attributes.

### Card Format

	1	2	3	4	5	6	7	8
Variable	SID	DA1	DA2	DA3	DA4			
Type	I	F	F	F	F			
Default	none	0.	0.	0.	0.			
Remarks		1	1	1	1			

**Card 2, 3, 4, ... (OPTION=LIST) (The next “\*” card terminates the input.)**

	1	2	3	4	5	6	7	8
Variable	EID1	EID2	EID3	EID4	EID5	EID6	EID7	EID8
Type	I	I	I	I	I	I	I	I
Remarks	2	2	2	2	2	2	2	2

**Card 2, 3, 4, ... (OPTION=COLUMN) (The next “\*” card terminates the input.)**

	1	2	3	4	5	6	7	8
Variable	EID	A1	A2	A3	A4			
Type	I	F	F	F	F			
Remarks		3	3	3	3			

**Cards 2, 3, 4, ... (OPTION=LIST\_GENERATE) (The next “\*” card terminates the input.)**

	1	2	3	4	5	6	7	8
Variable	B1BEG	B1END	B2BEG	B2END	B3BEG	B3END	B4BEG	B4END
Type	I	I	I	I	I	I	I	I

**VARIABLE**

**DESCRIPTION**

SID	Set ID. All shell sets should have a unique set ID.
DA1	First attribute default value, see remark 1.
DA2	Second attribute default value
DA3	Third attribute default value
DA4	Fourth attribute default value
EID1	First shell element ID, see remark 2.
EID2	Second shell element ID
.	.
.	.
EID	Element ID
A1	First attribute
A2	Second attribute

# \*SET

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
A3	Third attribute
A4	Fourth attribute
BNBEG	First shell ID in shell block N.
BNEND	Last shell ID in block N. All defined ID's between and including BNBEG to BNEND are added to the set. These sets are generated after all input is read so that gaps in the element numbering are not a problem. BNBEG and BNEND may simply be limits on the ID's and not element ID's.

## **Remarks:**

1. Shell attributes can be assigned for some input types. For example, for the contact options, the attributes for the SLAVE surface are:  
  
DA1=NFLS      Normal failure stress, \*CONTACT\_TIEBREAK\_SURFACE\_contact only,  
DA2=SFLS      Shear failure stress, \*CONTACT\_TIEBREAK\_SURFACE\_contact only,  
DA3=FSF        Coulomb friction scale factor,  
DA4=VSF        Viscous friction scale factor,  
  
and the attributes for the MASTER surface are:  
  
DA1=FSF        Coulomb friction scale factor,  
DA2=VSF        Viscous friction scale factor.
2. The default attributes are taken.
3. The default shell attributes can be overridden on these cards; otherwise, A1=DA1, etc.



**\*SET\_SOLID\_{OPTION}**

Available options include:

<BLANK>

**GENERATE**

The last option, GENERATE, will generate a block of solid element ID's between a starting ID and an ending ID. An arbitrary number of blocks can be specified to define the set.

Purpose: Define a set of solid elements.

**Card Format**

1 2 3 4 5 6 7 8

Variable	SID							
Type	I							
Default	none							

**Cards 2, 3, 4, ... (OPTION=none) (The next "\*" card terminates the input.)**

1 2 3 4 5 6 7 8

Variable	K1	K2	K3	K4	K5	K6	K7	K8
Type	I	I	I	I	I	I	I	I

**Cards 2, 3, 4, ... (OPTION=GENERATE) (The next "\*" card terminates the input.)**

1 2 3 4 5 6 7 8

Variable	B1BEG	B1END	B2BEG	B2END	B3BEG	B3END	B4BEG	B4END
Type	I	I	I	I	I	I	I	I

# \*SET

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
SID	Set ID. All solid sets should have a unique set ID.
K1	First element ID
K2	Second element ID
.	.
.	.
K8	Eighth element ID
.	.
.	.
BNBEG	First solid element ID in block N.
BNEND	Last solid element ID in block N. All defined ID's between and including BNBEG to BNEND are added to the set. These sets are generated after all input is read so that gaps in the element numbering are not a problem. BNBEG and BNEND may simply be limits on the ID's and not element ID's.

**\*SET\_TSHELL\_{OPTION}**

Available options include:

<BLANK>

**GENERATE**

The last option, GENERATE, will generate a block of thick shell element ID's between a starting ID and an ending ID. An arbitrary number of blocks can be specified to define the set.

Purpose: Define a set of thick shell elements.

**Card Format**

1 2 3 4 5 6 7 8

Variable	SID							
Type	I							
Default	none							

**Cards 2, 3, 4, ... (OPTION=none) (The next "\*" card terminates the input.)**

1 2 3 4 5 6 7 8

Variable	K1	K2	K3	K4	K5	K6	K7	K8
Type	I	I	I	I	I	I	I	I

**Cards 2, 3, 4, ... (OPTION=GENERATE) (The next "\*" card terminates the input.)**

1 2 3 4 5 6 7 8

Variable	B1BEG	B1END	B2BEG	B2END	B3BEG	B3END	B4BEG	B4END
Type	I	I	I	I	I	I	I	I

# \*SET

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
SID	Set ID. All tshell sets should have a unique set ID.
K1	First thick shell element ID
K2	Second thick shell element ID
.	.
.	.
K8	Eighth thick shell element ID
.	.
.	.
BNBEG	First thick shell element ID in block N.
BNEND	Last thick shell element ID in block N. All defined ID's between and including BNBEG to BNEND are added to the set. These sets are generated after all input is read so that gaps in the element numbering are not a problem. BNBEG and BNEND may simply be limits on the ID's and not element ID's.

# **\*TERMINATION**

The keyword **\*TERMINATION** provides an alternative way of stopping the calculation before the termination time is reached. The termination time is specified on the **\*CONTROL\_TERMINATION** input and will terminate the calculation whether or not the options available in this section are active. Different types of termination may be defined:

**\*TERMINATION\_NODE**

**\*TERMINATION\_BODY**

**\*TERMINATION\_CONTACT**

# \*TERMINATION

---

## \*TERMINATION\_NODE

Purpose: Terminate calculation based on nodal point coordinates. The analysis terminates for \*TERMINATION\_NODE when the current position of the node specified reaches either the maximum or minimum value (stops 1, 2 or 3), or picks up force from any contact surface (stop 4). Termination by other means than \*TERMINATION is controlled by the \*CONTROL\_TERMINATION control card. Note that this type of termination is not active during dynamic relaxation.

### Card Format

	1	2	3	4	5	6	7	8
Variable	NID	STOP	MAXC	MINC				
Type	I	I	F	F				
Default	none	none	-	-				

### VARIABLE

### DESCRIPTION

NID	Node ID, see *NODE_OPTION.
STOP	Stop criterion: EQ.1: global x direction, EQ.2: global y direction, EQ.3: global z direction, EQ.4: stop if node touches contact surface.
MAXC	Maximum (most positive) coordinate (options 1, 2 and 3) above only.
MINC	Minimum (most negative) coordinate (options 1, 2 and 3) above only.

# \*TERMINATION

## \*TERMINATION\_BODY

Purpose: Terminate calculation based on rigid body displacements. For \*TERMINATION\_BODY the analysis terminates when the centre of mass displacement of the rigid body specified reaches either the maximum or minimum value (stops 1, 2 or 3) or the displacement magnitude of the centre of mass is exceeded (stop 4). If more than one condition is input, the analysis stops when any of the conditions is satisfied. Termination by other means than \*TERMINATION input is controlled by the \*CONTROL\_TERMINATION control card. Note that this type of termination is not active during dynamic relaxation.

### Card Format

	1	2	3	4	5	6	7	8
Variable	PID	STOP	MAXC	MINC				
Type	I	I	F	F				
Default	none	none	-	-				

### VARIABLE

### DESCRIPTION

PID	Part ID of rigid body, see *PART_OPTION.
STOP	Stop criterion: EQ.1: global x direction, EQ.2: global y direction, EQ.3: global z direction, EQ.4: stop if displacement magnitude is exceeded.
MAXC	Maximum (most positive) displacement, options 1, 2, 3 and 4: EQ:0.0. MAXC set to 1.0e21.
MINC	Minimum (most negative) displacement, options 1, 2 and 3 above only: EQ:0.0. MINC set to -1.0e21.

# \*TERMINATION

---

## \*TERMINATION\_CONTACT

Purpose: The analysis terminates when the magnitude of the contact interface resultant force is zero. If more than one contact condition is input, the analysis stops when any of the conditions is satisfied. Termination by other means than \*TERMINATION input is controlled by the \*CONTROL\_TERMINATION control card. Note that this type of termination is not active during dynamic relaxation.

### Card Format

1            2            3            4            5            6            7            8

Variable	CID	ACTIM	DUR					
Type	I	I	F					
Default	none	none	-	-				

### VARIABLE

### DESCRIPTION

CID            Contact ID. The contact ID is defined by the ordering of the contact input unless the TITLE option which allows the CID to be defined is used in the \*CONTACT section.

ACTIM        Activation time.

DUR            Time duration of null resultant force prior to termination. This time is tracked only after the activation time is reached and the contact resultant forces are zero.

EQ.0.0: Immediate termination after null force is detected.



---

# **\*TITLE**

**\*TITLE**

Purpose: Define job title.

**Card Format**

1            2            3            4            5            6            7            8

Variable	TITLE
Type	C
Default	LS-DYNA USER INPUT

**VARIABLE**

**DESCRIPTION**

TITLE            Heading to appear on output and in output files.

**\*TITLE**

---

# **\*TRANSLATE**

## **\*TRANSLATE\_ANSYS\_OPTION**

Available options include:

- 4**
- 5**

corresponding to ANSYS version numbers 4 and 5.

Purpose: Provide a convenient route to read in ANSYS input decks as part of the LS-DYNA keyword input. This keyword can appear more than once anywhere in the input. It is a direct interface to ANSYS file28 keyword files.

### **Card Format**

	1	2	3	4	5	6	7	8
Variable	FILE							
Type	A							
Default	none							

**VARIABLE**

**DESCRIPTION**

FILE	Filename of file created by ANSYS, see remarks below.
------	---

The supported options include:

<b><u>Version</u></b>	<b><u>ANSYS Keyword</u></b>	<b><u>LS-DYNA Keyword</u></b>
All	N,Type,NODE,Val1,Val2,Val3	*NODE
All	EN,Type,I1,I2,I3,I4,I5,I6,I7,I8	*ELEMENT
All	MPDATA, R5.0, LENGTH, Lab, MAT, STLOC, VAL1, VAL2, VAL3	*MAT_ELASTIC

# \*TRANSLATE

---

<b>Version</b>	<b>ANSYS Keyword</b>	<b>LS-DYNA Keyword</b>
All	ET, Type	*PART&*SECTION
All	R,R5.0,NSET,Type,STLOC,VAL1,VAL2,VAL3	*PART&*SECTION
5	DFLAB,NODF,LabD,LabF	
5	NDOF.eq.Ui,ROTi; LabD.eq.0	*BOUNDARY_SPC_OPTION
5	NODF.eq.Vi; LabD.eq.0	*INITIAL_VELOCITY_NODE
5	NODF.eq.Ui,ROTi,Ai,Vi,;LabD.eq.lcid; LabF.eq.val	*BOUNDARY_PRESCRIBED_ MOTION_NODE
5	NDOF.eq.Fi; LabF.eq.lcid	*LOAD_NODE_POINT
5	SFE,ELEM,LKEY,Lab,KEY,R5.0	
5	LKEY.eq.lcid; Lab.eq.pressure	*LOAD_SEGMENT

## **Remarks:**

1. Supported keywords as described in the SASI ANSYS Manual chapter on “Exporting a Finite Element Model.”
2. Solid elements and shell elements cannot have the same R value in reference to the ET and R ANSYS keywords.
3. Supported element types include: 63.eq.shells, 45.eq.solids, 73.eq.solids, 4.eq.beams, 16.eq.pipes, and 21.eq.lumped masses.

## \*TRANSLATE\_IDEAS\_{OPTION}

Available options include:

<BLANK>

**MASTER**

Purpose: Provide a convenient route to read in files created by IDEAS/SUPERTAB as part of the LS-DYNA keyword input. This keyword can appear more than once in the input. It is a direct interface to IDEAS universal files.

### Card Format

	1	2	3	4	5	6	7	8
Variable	FILE							
Type	A							
Default	none							

#### VARIABLE

#### DESCRIPTION

FILE

Filename of the IDEAS universal file.

The following table lists supported IDEAS keywords:

<u>Version</u>	<u>SDRC IDEAS Universal File</u>	<u>LS-DYNA Keyword</u>
All	N,Type,NODE,Val1,Val2,Val3	*NODE
All	EN,Type,I1,I2,I3,I4,I5,I6,I7,I8	*ELEMENT
5	781	*NODE
7	2411	*NODE
5	780	*ELEMENT
7	2412	*ELEMENT
5	773	*MAT_ELASTIC
5	772	*PART&*SECTION
6	788	*PART&*SECTION

# \*TRANSLATE

---

<u>Version</u>	<u>SDRC IDEAS Universal File</u>	<u>LS-DYNA Keyword</u>
7	2430	*PART&*SECTION
5	755	*BOUNDARY_SPC_NODE
7	791	
	time variation set.le.0.0	*BOUNDARY_SPC_NODE
	time variation set.gt.0.0	*BOUNDARY_PRESCRIBED_ MOTION_NODE
7	790	
	load type.eq.1	*LOAD_NODE

## \*TRANSLATE\_NASTRAN

Purpose: Provide a convenient route to read in NASTRAN input deck as part of the LSDYNA keyword input. This keyword can appear more than once anywhere in the input. Also, see remarks below.

### Card Format

	1	2	3	4	5	6	7	8
Variable	FILE							
Type	C							

#### VARIABLE

#### DESCRIPTION

FILE                      Filename of the NASTRAN input deck.

The following table lists supported NASTRAN keywords:

<u>Version</u>	<u>NASTRAN INPUT FILE</u>	<u>LS-DYNA Keyword</u>
All	N,Type,NODE,Val1,Val2,Val3	*NODE
All	EN,Type,I1,I2,I3,I4,I5,I6,I7,I8	*ELEMENT
All	BEGIN BULK	
All	GRID	*NODE
All	CORD2R	*DEFINE_COORDINATE_SYSTEM
All	CHEXA, CPENTA, CTETRA	*ELEMENT_SOLID
All	PSOLID	*PART and *SECTION_SOLID
All	CQUAD4, CTRIA3	*ELEMENT_SHELL
All	PSHELL	*PART and *SECTION_SHELL
All	CBAR, CBEAM	*ELEMENT_BEAM
All	CELAS1, CVISC, CDAMP1	*ELEMENT_DISCRETE
All	CONM2	*ELEMENT_MASS
All	MAT1	*MAT_ELASTIC
All	SPC, SPC1	*BOUNDARY_SPC_OPTION

# \*TRANSLATE

---

<u>Version</u>	<u>NASTRAN INPUT FILE</u>	<u>LS-DYNA Keyword</u>
All	RBE2	*CONSTRAINED_NODE_SET or *CONSTRAINED_NODAL_RIGID_BODY_
All	ENDDATA	*END

## Remarks:

- Both small and large field fixed NASTRAN formats are supported.
- The same keywords in LS-DYNA usually contain more options than the NASTRAN input. Therefore, to make it complete, we add some extra parameters to the NASTRAN keywords. For those extras we use the italics to distinguish from the standard ones. These additional parameters have to be added to the NASTRAN deck by the user to make the translation complete.

## **Card Format**

For further explanation see \*ELEMENT\_DISCRETE.

	1	2	3	4	5	6	7	8	9
CELAS1	EID	PID	<i>NI</i>	<i>DFG</i>	<i>N2</i>	<i>VID</i>	<i>S</i>	<i>PF</i>	
Type	I	I	I	F	I	F	F	F	

For further explanation see \*PART and \*SECTION\_SHELL.

PSHELL	PID	MID	<i>T</i>	<i>FORM</i>	<i>SHEAR</i>	<i>NIP</i>	<i>QR</i>	<i>HGID</i>	
Type	I	I	F	I	F	I	F	I	

- Current **NASTRAN** only supports shell element with constant thickness T.



## \*TRANSLATE

---

For further explanation see \*PART and \*SECTION\_SOLID.

PSOLID	PID	MID	SCID	EOSID	HGID				
Type	I	I	I	I	I				

4. The **THRU** command for SPC, SPC1 is not supported in the current translation.
5. For RBE2 keyword, if any of the rotational DOF (4,5,6) appears in the constraint, LS-DYNA will treat it as nodal rigid body constraint. Otherwise, LS-DYNA will use nodal constraints to treat this RBE2.

# **\*TRANSLATE**

---

---

# **\*USER**

## **\*USER\_INTERFACE\_OPTION**

Options include:

### **CONTROL FRICTION**

Purpose: Define user defined input and allocate storage for user defined subroutines for the contact algorithms. See also \*CONTROL\_CONTACT. The **CONTROL** option above allows the user to take information from the contact interface for further action, e.g., stopping the analysis. A sample user subroutine is provided in Appendix D.

The **FRICTION** option may be used to modify the Coulomb friction coefficients according to contact information or to use a friction coefficient database. A sample subroutine for treating the friction in contact is provided in Appendix E.

### **Card Format**

	1	2	3	4	5	6	7	8
Variable	IFID	NOC	NOCI					
Type	I	I	I					
Default	none	none	none					

# \*USER

---

**Card Format (Use as many cards as necessary to define NOCI variables)**

	1	2	3	4	5	6	7	8
Variable	UC1	UC2	UC3	UC4	UC5	UC6	UC7	UC8
Type	F	F	F	F	F	F	F	F
Default	0.	0.	0.	0.	0.	0.	0.	0.

<u>VARIABLE</u>	<u>DESCRIPTION</u>
IFID	Interface number
NOC	Number of history variables for interface. The number should not exceed the length of the array defined on *CONTROL_CONTACT.
NOCI	Initialize the first NOCI history variables in the input. NOCI must be smaller or equal to NOC.
UC1	First user defined input parameter
UC2	Second user defined input parameter
.	.
.	.
.	.
UCNOCI	Last user defined input parameter
.	.
.	.
.	.

**\*USER\_LOADING**

Purpose: Provide a means of applying pressure and force boundary conditions. The keyword \*USER\_LOADING activates this option. Input here is optional with the input being read until the next "\*" keyword appears. The data read here is to be stored in a common block provided in the user subroutine, LOADUD. This data is stored and retrieved from the restart files.

**Card Format (Insert as many cards as needed. The next \* card terminates input.)**

	1	2	3	4	5	6	7	8
Variable	PARM1	PARM2	PARM3	PARM4	PARM5	PARM6	PARM7	PARM8
Type	F	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	none	none

---

**VARIABLE**

---

**DESCRIPTION**PARM<sub>n</sub>

This is the nth user input parameter.



---

# RESTART INPUT DATA

In general three categories of restart actions are possible with LS-DYNA and are outlined in the following discussion:

- a) A simple restart occurs when LS-DYNA was interactively stopped before reaching the termination time. Then simply defining the `R=rtf` file on the execution line for LS-DYNA restarts the calculation from the termination point and the calculation will continue to the specified termination time-see INTRODUCTION, Execution Syntax. No additional input deck is required.
  
- b) If minor modifications are desired as, e.g.,
  - reset termination time,
  - reset output printing interval,
  - reset output plotting interval,
  - delete contact surfaces,
  - delete elements and parts,
  - switch deformable bodies to rigid,
  - switch rigid bodies to deformable,
  - change damping options.

This type of restart is called a small restart and the corresponding input deck a “small restart input deck.” All modifications to the problem made with the restart input deck will be reflected in subsequent restart dumps. All the members of the file families are consecutively numbered beginning from the last member. The small input deck replaces the standard input deck on the execution line which has at least the following contents:

LS-DYNA `I=restartinput R=D3DUMPnn`

where `D3DUMPnn` (or whatever name is chosen for the family member) is the  $n$  th restart file from the last run where the data is taken. LS-DYNA automatically detects that a small input deck is used since the `I=restartinput` file may contain the keywords:

**\*CHANGE\_OPTION**  
**\*CONTROL\_DYNAMIC\_RELAXATION**  
**\*CONTROL\_TERMINATION**  
**\*CONTROL\_TIMESTEP**

# RESTART

---

**\*DAMPING\_GLOBAL**  
**\*DATABASE\_OPTION**  
**\*DATABASE\_BINARY\_OPTION**  
**\*DELETE\_OPTION**  
**\*INTERFACE\_SPRINGBACK**  
**\*RIGID\_DEFORMABLE\_OPTION**  
**\*STRESS\_INITIALIZATION\_{OPTION}**  
**\*TERMINATION\_OPTION**  
**\*TITLE**  
**\*KEYWORD** (see INTRODUCTION, Execution Syntax)  
**\*CONTROL\_CPU**  
**\*DEFINE\_OPTION**  
**\*SET\_OPTION**

i.e., the keyword **\*STRESS\_INITIALIZATION** may not be used in the small restart. The user has to take care that nonphysical modifications to the input deck are avoided; otherwise, complete nonsense may be the result.

- c) If many modifications are desired a so called full restart may be the appropriate choice. Then the keyword **\*STRESS\_INITIALIZATION** has to be provided in the input. As also outlined in the INTRODUCTION, Restart Analysis, either all parts can be initialized with the restart data or some selection of parts can be made for the stress initialization. See **\*STRESS\_INITIALIZATION**. In a full deck restart, deleted elements in this section will be deleted in the full deck automatically even though they are defined. Likewise, if it is necessary to change the velocity field, that must also be performed in this section using the **CHANGE\_VELOCITY\_....** options. The velocity field in the full deck part of the input is ignored.



## **\*CHANGE\_OPTION**

Available options are:

**BOUNDARY\_CONDITION**  
**CONTACT\_SMALL\_PENETRATION**  
**CURVE\_DEFINITION**  
**RIGID\_BODY\_CONSTRAINT**  
**RIGID\_BODY\_STOPPER**  
**STATUS\_REPORT\_FREQUENCY**  
**THERMAL\_PARAMETERS**  
**VELOCITY**  
**VELOCITY\_NODE**  
**VELOCITY\_RIGID\_BODY**  
**VELOCITY\_ZERO**

Purpose: Change some solution options.

# RESTART

---

For **BOUNDARY\_CONDITION** option define an arbitrary number of cards giving the nodal ID and the additional translational displacement boundary condition code. Previous boundary condition codes will continue to be imposed, i.e., a fixed node cannot be freed with this option. This input terminates when the next “\*” card is encountered.

## Card Format

	1	2	3	4	5	6	7	8
Variable	NID	BCC						
Type	I	I						

<u>VARIABLE</u>	<u>DESCRIPTION</u>
NID	Nodal point ID, see also *NODE.
BCC	New translational boundary condition code: EQ.1: constrained x displacement, EQ.2: constrained y displacement, EQ.3: constrained z displacement, EQ.4: constrained x and y displacements, EQ.5: constrained y and z displacements, EQ.6: constrained z and x displacements, EQ.7: constrained x, y, and z displacements.

# RESTART

For **CONTACT\_SMALL\_PENETRATION** option define an arbitrary number of cards giving a list of contact surface ID numbers where the small penetration check is to be turned on. This input terminates when the next “\*” card is encountered. See the PENCHK variable on the \*CONTACT definition.

## Card Format

	1	2	3	4	5	6	7	8
Variable	ID1	ID2	ID3	ID4	ID5	ID6	ID7	ID8
Type	I	I	I	I	I	I	I	I

### VARIABLE

### DESCRIPTION

ID<sub>n</sub>

Contact ID for surface number *n*.

The **CURVE\_DEFINITION** option allows a load curve to be redefined. *The new load curve must contain the same number of points as the curve it replaces.* The curve should be defined in the DEFINE\_CURVE section of this manual. This input terminates when the next “\*” card is encountered. Any offsets and scale factors are ignored.

## Card Format

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

### VARIABLE

### DESCRIPTION

LCID

Load curve ID

# RESTART

---

The **RIGID\_BODY\_CONSTRAINT** option allows translational and rotational boundary conditions on a rigid body to be changed. This input terminates when the next “\*” card is encountered. Also, see \*CONSTRAINED\_RIGID\_BODIES.

## Card Format

	1	2	3	4	5	6	7	8
Variable	PID	TC	RT					
Type	I	I	I					

<u>VARIABLE</u>	<u>DESCRIPTION</u>
PID	Part ID, see *PART.
TC	Translational constraint: EQ.0: no constraints, EQ.1: constrained x displacement, EQ.2: constrained y displacement, EQ.3: constrained z displacement, EQ.4: constrained x and y displacements, EQ.5: constrained y and z displacements, EQ.6: constrained z and x displacements, EQ.7: constrained x, y, and z displacements.
RC	Rotational constraint: EQ.0: no constraints, EQ.1: constrained x rotation, EQ.2: constrained y rotation, EQ.3: constrained z rotation, EQ.4: constrained x and y rotations, EQ.5: constrained y and z rotations, EQ.6: constrained z and x rotations, EQ.7: constrained x, y, and z rotations.

# RESTART

The **RIGID\_BODY\_STOPPER** option allows existing stoppers to be redefined. This input terminates when the next “\*” card is encountered. See \*CONSTRAINED\_RIGID\_BODY\_STOPPERS.

New stopper definitions cannot be introduced in this section. Existing stoppers can be modified.

## Card Formats

Card 1            1            2            3            4            5            6            7            8

Variable	PID	LCMAX	LCMIN	PSIDMX	PSIDMN	LCVMNX	DIR	VID
Type	I	I	I	I	I	I	I	I
Default	required	0	0	0	0	0	required	0

Card 2            1            2            3            4            5            6            7            8

Variable	BIRTH	DEATH						
Type	F	F						
Default	0	10 <sup>28</sup>						

### VARIABLE

### DESCRIPTION

PID	Part ID of master rigid body, see *PART.
LCMAX	Load curve ID defining the maximum coordinate as a function of time: EQ.0: no limitation of the maximum displacement. New curves can be defined by the *DEFINE_CURVE within the present restart deck.
LCMIN	Load curve ID defining the minimum coordinate as a function of time: EQ.0: no limitation of the minimum displacement. New curves can be defined by the *DEFINE_CURVE within the present restart deck.
PSIDMX	Optional part set ID of rigid bodies that are slaved in the maximum coordinate direction to the master rigid body. This option requires additional input by the *SET_PART definition.

# RESTART

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
PSIDMN	Optional part set ID of rigid bodies that are slaved in the minimum coordinate direction to the master rigid body. This option requires additional input by the *SET_PART definition.
LCVMNX	Load curve ID which defines the maximum absolute value of the velocity that is allowed within the stopper: EQ.0: no limitation of the minimum displacement.
DIR	Direction stopper acts in: EQ.1: x-translation, EQ.2: y-translation, EQ.3: z-translation, EQ.4: arbitrary, defined by vector VID, EQ.5: x-axis rotation, EQ.6: y-axis rotation, EQ.7: z-axis rotation, EQ.8: arbitrary, defined by vector VID.
VID	Vector for arbitrary orientation of stopper. The vector must be defined by a *DEFINE_VECTOR within the present restart deck.
BIRTH	Time at which stopper is activated.
DEATH	Time at which stopper is deactivated.

## **Remarks:**

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

# RESTART

The **STATUS\_REPORT\_FREQUENCY** option allows the output status interval to be changed.

## Card Format

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

### VARIABLE

### DESCRIPTION

IKEDIT

Problem status report interval steps in the D3HSP output file:  
EQ.0: interval remains unchanged.

# RESTART

---

The **THERMAL\_PARAMETERS** option allows parameters used by a thermal or coupled structural/thermal analysis to be changed. These parameters were initially defined on the \*CONTROL\_THERMAL cards. Two cards are defined for this option.

## Card Format (Card 1 of 2)

	1	2	3	4	5	6	7	8
Variable	TS	DT	TMIN	TMAX	DTEMP	TSCP		
Type	I	F	F	F	F	F		

## Card Format (Card 2 of 2)

	1	2	3	4	5	6	7	8
Variable	REFMAX	TOL						
Type	I	F						

<u>VARIABLE</u>	<u>DESCRIPTION</u>
TS	Thermal time step code: EQ.0: No change, EQ.1: Fixed timestep, EQ.2: variable timestep.
DT	Thermal time step on restart: EQ.0: No change.
TMIN	Minimum thermal timestep: EQ.0: No change.
TMAX	Maximum thermal timestep: EQ.0: No change.
DTEMP	Maximum temperature change in a thermal timestep: EQ.0: No change.
TSCP	Time step control parameter (0.0 < TSCP < 1.0): EQ.0: No change.
REFMAX	Maximum number of reformations per thermal time step: EQ.0: No change.



<u>VARIABLE</u>	<u>DESCRIPTION</u>
TOL	Non-linear convergence tolerance: EQ.0: No change.

# RESTART

---

The **VELOCITY\_NODE** option allows the velocity of nodal points to be changed at restart. Termination of this input is when the next “\*” card is read.

## Card Format

	1	2	3	4	5	6	7	8
Variable	NID	VX	VY	VZ	VXR	VYR	VZR	
Type	I	F	F	F	F	F	F	
Default	none	0.	0.	0.	0.	0.	0.	

<u>VARIABLE</u>	<u>DESCRIPTION</u>
NID	Node ID
VX	Translational velocity in x-direction.
VY	Translational velocity in y-direction.
VZ	Translational velocity in z-direction.
VXR	Rotational velocity about the x-axis.
VYR	Rotational velocity about the y-axis.
VZR	Rotational velocity about the z-axis.

## **Remarks:**

1. If a node is initialized on more than one input card set, then the last set input will determine its velocity, unless it is specified on a \*CHANGE\_VELOCITY\_NODE card.
2. Undefined nodes will have their nodal velocities set to zero if a \*CHANGE\_VELOCITY definition is encountered in the restart deck.
3. If both \*CHANGE\_VELOCITY and \*CHANGE\_VELOCITY\_ZERO cards are defined then all velocities will be reset to zero.

# RESTART

The **VELOCITY** option allows a new velocity field to be imposed at restart. Termination of this input is when the next “\*” card is read.

## Card Format

Card 1            1            2            3            4            5            6            7            8

Variable	NSID							
Type	I							
Default	none							
Remark	1							

Card 2            1            2            3            4            5            6            7            8

Variable	VX	VY	VZ	VXR	VYR	VZR		
Type	F	F	F	F	F	F		
Default	0.	0.	0.	0.	0.	0.		

### VARIABLE

### DESCRIPTION

NSID	Nodal set ID containing nodes for initial velocity.
VX	Velocity in x-direction.
VY	Velocity in y-direction.
VZ	Velocity in z-direction.

# RESTART

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
VXR	Rotational velocity about the x-axis.
VYR	Rotational velocity about the y-axis.
VZR	Rotational velocity about the z-axis.

## **Remarks:**

1. If a node is initialized on more than one input card set, then the last set input will determine its velocity, unless it is specified on a \*CHANGE\_VELOCITY\_NODE card.
2. Undefined nodes will have their nodal velocities set to zero if a \*CHANGE\_VELOCITY definition is encountered in the restart deck.
3. If both \*CHANGE\_VELOCITY and \*CHANGE\_VELOCITY\_ZERO cards are defined then all velocities will be reset to zero.

# RESTART

The **VELOCITY\_RIGID\_BODY** option allows the velocity components of a rigid body to be changed at restart. Termination of this input is when the next “\*” card is read.

## Card Format

	1	2	3	4	5	6	7	8
Variable	PID	VX	VY	VZ	VXR	VYR	VZR	
Type	I	F	F	F	F	F	F	
Default	none	0.	0.	0.	0.	0.	0.	

### VARIABLE

### DESCRIPTION

PID	Part ID of rigid body.
VX	Translational velocity in x-direction.
VY	Translational velocity in y-direction.
VZ	Translational velocity in z-direction.
VXR	Rotational velocity about the x-axis.
VYR	Rotational velocity about the y-axis.
VZR	Rotational velocity about the z-axis.

### **Remarks:**

1. Rotational velocities are defined about the center of mass of the rigid body.
2. Rigid bodies not defined in this section will not have their velocities modified.

# RESTART

---

The **VELOCITY\_ZERO** option resets the velocities to zero at the start of the restart. Only the \*CHANGE\_VELOCITY\_ZERO card is required for this option without any further input.

**\*CONTROL\_DYNAMIC\_RELAXATION**

Purpose: Define controls for dynamic relaxation.

**Card Format**

	1	2	3	4	5	6	7	8
Variable	NRCYCK	DRTOL	DRFCTR	DRTERM	TSSFDR	IRELAL	EDTTL	IDRFLG
Type	I	F	F	F	F	I	F	I
Default	250	0.001	0.995	infinity	TSSFAC	0	0.0	0
Remarks	1	1	1	1	1			1

**VARIABLE****DESCRIPTION**

NRCYCK	Number of iterations between convergence checks, for dynamic relaxation option (default = 250).
DRTOL	Convergence tolerance for dynamic relaxation option (default = 0.001).
DRFCTR	Dynamic relaxation factor (default = .995).
DRTERM	Optional termination time for dynamic relaxation. Termination occurs at this time or when convergence is attained (default = infinity).
TSSFDR	Scale factor for computed time step during dynamic relaxation. If zero, the value is set to TSSFAC defined on *CONTROL_TERMINATION. After converging, the scale factor is reset to TSSFAC.
IRELAL	Automatic control for dynamic relaxation option based on algorithm of Papadrakakis [Papadrakakis 1981].
EDTTL	Convergence tolerance on automatic control of dynamic relaxation.
IDRFLG	Dynamic relaxation flag for stress initialization: EQ.0: not active, EQ.1: dynamic relaxation is activated.

# RESTART

---

## **Remarks:**

1. If a dynamic relaxation analysis is being restarted at a point before convergence was obtained, then NRCYCK, DRTOL, DRFCTR, DRTERM and TSSFDR will default to their previous values, and IDRFLG will be set to 1.
2. If dynamic relaxation is activated after a restart from a normal transient analysis LS-DYNA continues the output of data as it would without the dynamic relaxation being active. This is unlike the dynamic relaxation phase at the beginning of the calculation when a separate database is not used. Only load curves that are flagged for dynamic relaxation are applied after restarting.



## \*CONTROL\_TERMINATION

Purpose: Stop the job.

### Card Format

	1	2	3	4	5	6	7	8
Variable	ENDTIM	ENDCYC						
Type	F	I						

---

#### VARIABLE

#### DESCRIPTION

---

ENDTIM

Termination time:  
EQ:0.0 Termination time remains unchanged.

ENDCYC

Termination cycle. The termination cycle is optional and will be used if the specified cycle is reached before the termination time.  
EQ:0.0 Termination cycle remains unchanged.

This is a reduced version of the \*CONTROL\_TERMINATION card used in the initial input deck.

# RESTART

---

## \*CONTROL\_TIMESTEP

Purpose: Set time step size control using different options.

### Card Format

	1	2	3	4	5	6	7	8
Variable	DUMMY	TSSFAC	ISDO	DUMMY	DT2MS	LCTM		
Type	F	F	I	F	F	I		

<u>VARIABLE</u>	<u>DESCRIPTION</u>
DUMMY	Dummy field, see remark 1 below.
TSSFAC	Scale factor for computed time step. EQ:0.0. TSSFAC remains unchanged.
ISDO	Basis of time size calculation for 4-node shell elements, ISDO 3-node shells use the shortest altitude for options 0,1 and the shortest side for option 2. This option has no relevance to solid elements, which use a length based on the element volume divided by the largest surface area: EQ.0: characteristic length=area/(longest side), EQ.1: characteristic length=area/(longest diagonal), EQ.2: based on bar wave speed and MAX [shortest side, area/longest side]. THIS LAST OPTION CAN GIVE A MUCH LARGER TIME STEP SIZE THAT CAN LEAD TO INSTABILITIES IN SOME APPLICATIONS, ESPECIALLY WHEN TRIANGULAR ELEMENTS ARE USED.
DUMMY	Dummy field, see remark 1 below.
DT2MS	New time step for mass scaled calculations. Mass scaling must be active in the time zero analysis. EQ:0.0. DT2MS remains unchanged.
LCTM	Load curve ID that limits maximum time step size: EQ:0. LCTM remains unchanged.

### **Remark:**

1. This a reduced version of the \*CONTROL\_TIMESTEP used in the initial analysis. The dummy fields are included to maintain compatability. If using free format input then a 0.0 should be entered for the dummy values.

## \*DAMPING\_GLOBAL

Purpose: Define mass weighed nodal damping that applies globally to the deformable nodes.

### Card Format

	1	2	3	4	5	6	7	8
Variable	LCID	VALDMP						
Type	I	F						
Default	0	0.0						

### VARIABLE

### DESCRIPTION

LCID

Load curve ID which specifies node system damping:  
EQ.n: system damping is given by load curve n. The damping force applied to each node is  $f = -d(t)mv$ , where  $d(t)$  is defined by load curve n.

VALDMP

System damping constant, d (this option is bypassed if the load curve number defined above is nonzero).

# RESTART

---

## **\*DATABASE\_OPTION**

Options for ASCII files include. If a file is not specified in the restart deck then the output interval for the file will remain unchanged.

<b>SECFORC</b>	Cross section forces.
<b>RWFORC</b>	Wall forces.
<b>NODOUT</b>	Nodal point data.
<b>ELOUT</b>	Element data.
<b>GLSTAT</b>	Global data.
<b>DEFORC</b>	Discrete elements.
<b>MATSUM</b>	Material energies.
<b>NCFORC</b>	Nodal interface forces.
<b>RCFORC</b>	Resultant interface forces.
<b>DEFGEO</b>	Deformed geometry file
<b>SPCFORC</b>	Set dt for spc reaction forces.
<b>SWFORC</b>	Nodal constraint reaction forces (spotwelds and rivets).
<b>ABSTAT</b>	Set dt for airbag statistics.
<b>NODFOR</b>	Set dt for nodal force groups.
<b>BNDOUT</b>	Boundary condition forces and energy
<b>RBDOUT</b>	Set dt for rigid body data.
<b>GCEOUT</b>	Set dt for geometric contact entities.
<b>SLEOUT</b>	Set dt for sliding interface energy.
<b>JNTFORC</b>	Set dt for joint force file.
<b>SBTOUT</b>	Set dt for seat belt output file.
<b>AVSFLT</b>	Set dt for AVS database.
<b>MOVIE</b>	Set dt for MOVIE.
<b>MPGS</b>	Set dt for MPGS.
<b>TPRINT</b>	Set dt for thermal file.

## Card Format

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

VARIABLE

DESCRIPTION

DT

Time interval between outputs:  
EQ:0.0 output interval is unchanged.

To terminate output to a particular file set DT to a high value.

# RESTART

---

## \*DATABASE\_BINARY\_OPTION

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

- D3PLOT** Dt for complete output states.
- D3THDT** Dt for time history data for element subsets.
- D3DUMP** Binary output restart files. Define output frequency in cycles
- RUNRSF** Binary output restart file. Define output frequency in cycles.
- INTFOR** Dt for contact surface Interface database.

### Card Format

	1	2	3	4	5	6	7	8
Variable	DT/CYCL							
Type	F							

<u>VARIABLE</u>	<u>DESCRIPTION</u>
DT	Time interval between outputs. EQ:0.0. Time interval remains unchanged.
CYCL	Output interval in time steps. EQ:0.0. output interval remains unchanged.

## \*DELETE\_OPTION

Available options are:

**CONTACT**  
**CONTACT\_2DAUTO**  
**ENTITY**  
**PART**  
**ELEMENT\_BEAM**  
**ELEMENT\_SHELL**  
**ELEMENT\_SOLID**  
**ELEMENT\_TSHELL**

Purpose: Delete contact surfaces, parts, or elements by a list of IDs. There are two contact algorithms for two dimensional problems: the line-to-line contact and the automatic contact defined by part ID's. Each use their own numbering.

For **CONTACT**, **CONTACT\_2DAUTO**, **ENTITY**, or **PART** option.

### Card Format

	1	2	3	4	5	6	7	8
Variable	ID1	ID2	ID3	ID4	ID5	ID6	ID7	ID8
Type	I	I	I	I	I	I	I	I

---

#### VARIABLE

#### DESCRIPTION

IDI

Contact ID/Part ID

For \*DELETE\_CONTACT a negative ID implies that the absolute value gives the contact surface which is to be activated

# RESTART

---

For the four **ELEMENT** options. Termination of input is when the next “\*” card is read.

## Card Format

1            2            3            4            5            6            7            8

Variable	ESID							
Type	I							

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
ESID	Element set ID, see *SET_SOLID, *SET_BEAM, *SET_SHELL, *SET_TSHELL.



## \*INTERFACE\_SPRINGBACK

Purpose: Define a material subset for an implicit springback calculation in LS-NIKE3D and any nodal constraints to eliminate rigid body degrees-of-freedom. Generally, only the materials that make up the original blank are included in the springback calculation. After termination of the LS-DYNA3D computation, an input deck for LS-NIKE3D and a stress initialization file for LS-NIKE3D are written.

### Card Format

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

### VARIABLE

### DESCRIPTION

PSID

Part set ID for springback, see \*SET\_PART.

Define a list of nodal points that are constrained for the springback. This section is terminated by an "\*" indicating the next input section.

### Card Format

	1	2	3	4	5	6	7	8
Variable	NID	TC	RC					
Type	I	F	F					
Default	none	0.	0.					

# RESTART

---

<u>VARIABLE</u>	<u>DESCRIPTION</u>
NID	Node ID
TC	Tranlational constraint: EQ.0: no constraints, EQ.1: constrained x displacement, EQ.2: constrained y displacement, EQ.3: constrained z displacement, EQ.4: constrained x and y displacements, EQ.5: constrained y and z displacements, EQ.6: constrained z and x displacements, EQ.7: constrained x, y, and z displacements.
RC	Rotational constraint: EQ.0: no constraints, EQ.1: constrained x rotation, EQ.2: constrained y rotation, EQ.3: constrained z rotation, EQ.4: constrained x and y rotations, EQ.5: constrained y and z rotations, EQ.6: constrained z and x rotations, EQ.7: constrained x, y, and z rotations.

## **\*RIGID\_DEFORMABLE\_OPTION**

The *OPTIONS* available are:

### **CONTROL**

**D2R**            (Deformable to rigid part switch)

**R2D**            (Rigid to deformable part switch)

Purpose: Define parts to be switched from rigid to deformable and deformable to rigid in a restart. It is only possible to switch parts on a restart if part switching was activated in the time zero analysis. See \*DEFORMABLE\_TO\_RIGID for details of part switching.

# RESTART

---

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

## Card Format

	1	2	3	4	5	6	7	8
Variable	NRBF	NCSF	RWF	DTMAX				
Type	I	I	I	F				
Default	0	0	0	none				

<u>VARIABLE</u>	<u>DESCRIPTION</u>
NRBF	<p>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.</p>
NCSF	<p>Flag to delete or activate nodal constraint set.            If nodal constraint/spotweld definitions are active in the deformable bodies that are switched to rigid, then the definitions should be deleted to avoid instabilities:                EQ.0: no change,                EQ.1: delete,                EQ.2: activate.</p>
RWF	<p>Flag to delete or activate rigid walls:                EQ.0: no change,                EQ.1: delete,                EQ.2: activate.</p>
DTMAX	Maximum permitted time step size after restart.

# RESTART

For the **D2R** option define the following card. Termination of this input is when the next “\*” card is read.

## Card Format

	1	2	3	4	5	6	7	8
Variable	PID	MRB						
Type	I	I						
Default	none	0						

### VARIABLE

### DESCRIPTION

PID	Part ID of the part which is switched to a rigid material.
MRB	Part ID of the master rigid body to which the part is merged. If zero, the part becomes either an independent or master rigid body.

For the **R2D** option define the following card. Termination of this input is when the next “\*” card is read.

## Card Format

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

### VARIABLE

### DESCRIPTION

PID	Part ID of the part which is switched to a deformable material.
-----	---

# RESTART

---

**\*STRESS\_INITIALIZATION\_{*OPTION*}**

This keyword allows a full deck restart to be performed in LS-DYNA. For a full deck restart a complete input deck has to be included in the restart deck. The stress initialization feature allows all or a number of parts to be initialized on restart.

The options that are available with this keyyord are:

**<BLANK>**  
**DISCRETE**  
**SEATBELT**

## \*STRESS\_INITIALIZATION

If this card is specified without further input then all parts in the new analysis are initialized from the corresponding part of the old analysis. Further all seatbelt and discrete parts are initialized.

If only a subset of parts are to be initialized in the new analysis then define as many of the following cards as necessary. Termination of this input is when the next “\*” card is read.

### Card Format

Card 1            1            2            3            4            5            6            7            8

Variable	PIDO	PIDN						
Type	I	I						
Default	none	PIDO						

### VARIABLE

### DESCRIPTION

PIDO	Old part ID, see *PART.
PIDN	New part ID, see *PART: EQ:0. New part ID is the same as the old part ID.

### **Remarks:**

If one or more of the above cards are defined then discrete and seatbelt elements will not be initialized unless the additional option cards \*STRESS\_INITIALIZATION\_DISCRETE and \*STRESS\_INITIALIZATION\_SEATBELT are defined.

# RESTART

---

## **\*STRESS\_INITIALIZATION\_DISCRETE**

Initialize all discrete parts from the old parts. No further input is required with this card. This card is not required if \*STRESS\_INITIALIZATION is specified without further input.

## **\*STRESS\_INITIALIZATION\_SEATBELT**

Initialize all seatbelt parts from the old parts. No further input is required with this card. This card is not required if \*STRESS\_INITIALIZATION is specified without further input.



## \*TERMINATION\_OPTION

Purpose: Stop the job depending on some displacement conditions.

Available options include:

### NODE

### BODY

**Caution:** The inputs are different for the nodal and rigid body stop conditions. The nodal stop condition works on the global coordinate position, while the body stop condition works on the relative global translation. The number of termination conditions cannot exceed the maximum of 10 or the number specified in the original analysis.

The analysis terminates for \*TERMINATION\_NODE when the current position of the node specified reaches either the maximum or minimum value (stops 1, 2 or 3), or picks up force from any contact surface (stop 4). For \*TERMINATION\_BODY the analysis terminates when the center of mass displacement of the rigid body specified reaches either the maximum or minimum value (stops 1, 2 or 3) or the displacement magnitude of the center of mass is exceeded (stop 4). If more than one condition is input, the analysis stops when any of the conditions is satisfied. *This input completely overrides the existing termination conditions defined in the time zero run.*

Termination by other means is controlled by the \*CONTROL\_TERMINATION control card.

**For both options, the input is identical:**

### Card Format

	1	2	3	4	5	6	7	8
Variable	NID/PID	STOP	MAXC	MINC				
Type	I	I	F	F				
Default	none	none	-	-				

# RESTART

---

For the **NODE** option:

<u>VARIABLE</u>	<u>DESCRIPTION</u>
NID	Node ID
STOP	Stop criterion: EQ.1: global x direction, EQ.2: global y direction, EQ.3: global z direction, EQ.4: stop if node touches contact surface.
MAXC	Maximum (most positive) coordinate, options 1, 2 and 3 above only.
MINC	Minimum (most negative) coordinate, options 1, 2 and 3 above only.

For the **BODY** option:

<u>VARIABLE</u>	<u>DESCRIPTION</u>
PID	Part ID of rigid body
STOP	Stop criterion: EQ.1: global x direction, EQ.2: global y direction, EQ.3: global z direction, EQ.4: stop if displacement magnitude is exceeded.
MAXC	Maximum (most positive) displacement, options 1, 2, 3 and 4: EQ:0.0. MAXC set to 1.0e21
MINC	Minimum (most negative) displacement, options 1, 2 and 3 above only: EQ:0.0. MINC set to -1.0e21

## \*TITLE

Purpose: Define job title.

### Card Format

1            2            3            4            5            6            7            8

Variable	TITLE
Type	C
Default	LS-DYNA USER INPUT

---

#### VARIABLE

#### DESCRIPTION

---

TITLE

Heading to appear on output.

# RESTART

---

### REFERENCES

- Allman, D.J., "A Compatible Triangular Element Including Vertex Rotations for Plane Elasticity Analysis," *Comp. Struct.*, 19,1-8, (1984).
- 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).
- Bahler AS: The series elastic element of mammalian skeletal muscle. *Am J Physiol* 213:1560-1564, 1967.
- Baker, E.L., "An Explosives Products Thermodynamic Equation of State Appropriate for Material Acceleration and Overdriven Detonation: Theoretical Background and Formulation," Technical Report ARAED-TR-911013, U.S. Army Armament Research, Development and Engineering Center, Picatinney Arsenal, New Jersey, 1991.
- Baker, E.L. and J. Orosz, J., "Advanced Warheads Concepts: An Advanced Equation of State for Overdriven Detonation," Technical Report ARAED-TR-911007, U.S. Army Armament Research, Development and Engineering Center, Picatinney Arsenal, New Jersey, 1991.
- Baker, E.L. and L.I. Stiel, "Improved Quantitative Explosive Performance Prediction Using Jaguar," 1997 Insensitive Munitions and Energetic Materials Technology Symposium, Tampa, FL, (1997).
- Bammann, D.J. and E.C. Aifantis, "A Model for Finite-Deformation Plasticity," *Acta Mechanica*, 70, 1-13 (1987).
- Bammann, D.J. and G. Johnson, "On the Kinematics of Finite-Deformation Plasticity," *Acta Mechanica*, 69, 97-117 (1987).
- Bammann, D.J., "Modeling the Temperature and Strain Rate Dependent Large Deformation of Metals," Proceedings of the 11th US National Congress of Applied Mechanics, Tuscon, AZ, (1989).
- Bammann, D.J., M.L. Chiesa, A. McDonald, W.A. Kawahara, J.J. Dike, and V.D. Revelli, "Predictions of Ductile Failure in Metal Structures," in AMD-Vol. 107, Failure Criteria and Analysis in Dynamic Response, Edited by. H.E. Lindberg, 7-12, (1990).
- Bandak, F.A., private communications, U.S. Dept. of Trans., Division of Biomechanics Research, 400 7th St., S.W. Washington, D.C. 20590 (1991).
- Barlat, F. and J. Lian, "Plastic Behavior and Stretchability of Sheet Metals. Part I: A Yield Function for Orthotropic Sheets Under Plane Stress Conditions," *Int. J. of Plasticity*, Vol. 5, pp. 51-66 (1989).
- Barlat, F., D.J. Lege, and J.C. Brem, "A Six-Component Yield Function for Anisotropic Materials," *Int. J. of Plasticity*, 7, 693-712, (1991).
- Barlat, F., Y. Maeda, K. Chung, M. Yanagawa, J.C. Brem, Y. Hayashida, D.J. Lege, K. Matsui, S.J. Murtha, S. Hattori, R.C. Becker, and S. Makosey, "Yield Function Development for Aluminum Alloy Sheets", *J. Mech. Phys. Solids*, Vol. 45, No. 11-12, 1727-1763, (1997).

## REFERENCES

---

- Bazeley, G.P., W.K. Cheung, R.M. Irons, and O.C. Zienkiewicz, "Triangular Elements in Plate Bending-Confirming and Nonconforming Solutions in Matrix Methods and Structural Mechanics," Proc. Conf. on Matrix Methods in Structural Analysis, Rept. AFFDL-R-66-80, Wright Patterson AFB, 547-576 (1965).
- Belytschko, T. and Bindeman, L. P. "Assumed Strain Stabilization of the Eight Node Hexahedral Element," *Comp. Meth. Appl. Mech. Eng.* **105**, 225-260 (1993).
- Belytschko, T.B. and A.H. Marchertas, "Nonlinear Finite Element Method for Plates and its Application to the Dynamic Response of Reactor Fuel Subassemblies," *Trans. ASME J. Pressure Vessel Tech.*, 251-257 (1974).
- Belytschko, T.B. and C.S. Tsay, "Explicit Algorithms for Nonlinear Dynamics of Shells," AMD-Vol.48, ASME, 209-231 (1981).
- Belytschko, T.B. and C.S. Tsay, "Explicit Algorithms for Nonlinear Dynamics of Shells," *Comp. Meth. Appl. Mech. Eng.*, 43, 251-276, (1984).
- Belytschko, T.B. and C.S. Tsay, "A Stabilization Procedure for the Quadrilateral Plate Element with One-Point Quadrature," *Int. J. Num. Method. Eng.*, 19, 405-419 (1983).
- Belytschko, T.B., H. Stolarski, and N. Carpenter, "A C<sup>0</sup> Triangular Plate Element with One-Point Quadrature," *Int. J. Num. Meth. Eng.*, 20, 787-802 (1984).
- Belytschko, T.B., L. Schwer, and M.J. Klein, "Large Displacement Transient Analysis of Space Frames," *Int. J. Num. Eng.*, 11, 65-84 (1977).
- Benson, D.J. and J.O. Hallquist, "A Simple Rigid Body Algorithm for Structural Dynamics Programs," *Int. J. Numer. Meth. Eng.*, 22, (1986).
- Benson, D.J. and J.O. Hallquist, "A Single Surface Contact Algorithm for the Postbuckling Analysis of Shell Structures," *Comp. Meths. Appl. Mech. Eng.*, 78, 141-163 (1990).
- 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).
- Berstad, T., Hopperstad, O.S., Lademo, O.-G. and Malo, K.A., "Computational Model of Ductile Damage and Fracture in Shell Analysis", Second European LS-DYNA Conference, Gothenburg, Sweden, (1999).
- 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," RAND, RM-6301-DASA, DASA 2506, (1970).

## REFERENCES

---

- Brown, B.E. and J.O. Hallquist, "TAURUS: An Interactive Post-Processor for the Analysis Codes NIKE3D, DYNA3D, TACO3D, and GEMINI," University of California, Lawrence Livermore National Laboratory, Rept. UCID-19392 (1982) Rev. 1 (1984).
- 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).
- 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., E. Albright, and N. Alexander, "The Joy Computer Code," Lawrence Livermore National Laboratory, Internal Document Rept. UCID-19688, (January, 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).
- Dick, R.E., and W.H. Harris, "Full Automated Rezoning of Evolving Geometry Problems," Numerical Methods in Industrial Forming Processes, Chenot, Wood, and Zienkiewicz, Editors, Bulkema, Rotterdam, 243-248, (1992)
- Dilger, W.H., R. Koch, and R. Kowalczyk, "Ductility of Plain and Confined Concrete Under Different Strain Rates," ACI Journal, January-February, (1984).
- Dobratz, B.M., "LLNL Explosives Handbook, Properties of Chemical Explosives and Explosive Simulants," University of California, Lawrence Livermore National Laboratory, Rept. UCRL-52997 (1981).
- 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)

## REFERENCES

---

- 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 Exposition, 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., "Preliminary User's Manuals for DYNA3D and DYNAP (Nonlinear Dynamic Analysis of Solids in Three Dimension)," University of California, Lawrence Livermore National Laboratory, Rept. UCID-17268 (1976) and Rev. 1 (1979).[a]
- Hallquist, J.O., "A Procedure for the Solution of Finite Deformation Contact-Impact Problems by the Finite Element Method," University of California, Lawrence Livermore National Laboratory, Rept. UCRL-52066 (1976).
- Hallquist, J.O., "A Numerical Procedure for Three-Dimensional Impact Problems," American Society of Civil Engineering, Preprint 2956 (1977).
- Hallquist, J.O., "A Numerical Treatment of Sliding Interfaces and Impact," in: K.C. Park and D.K. Gartling (eds.) Computational Techniques for Interface Problems, AMD Vol. 30, ASME, New York (1978).
- Hallquist, J.O., "NIKE2D: An Implicit, Finite-Element Code for Analyzing the Static and Dynamic Response of Two-Dimensional Solids," University of California, Lawrence Livermore National Laboratory, Rept. UCRL-52678 (1979).[b]
- Hallquist, J.O., "User's Manual for DYNA2D – An Explicit Two-Dimensional Hydrodynamic Finite Element Code with Interactive Rezoning," University of California, Lawrence Livermore National Laboratory, Rept. UCID-18756 (1980).
- Hallquist, J.O., "User's Manual for DYNA3D and DYNAP (Nonlinear Dynamic Analysis of Solids in Three Dimensions)," University of California, Lawrence Livermore National Laboratory, Rept. UCID-19156 (1981).[a]



## REFERENCES

---

- Hallquist, J. O., "NIKE3D: An Implicit, Finite-Deformation, Finite-Element Code for Analyzing the Static and Dynamic Response of Three-Dimensional Solids," University of California, Lawrence Livermore National Laboratory, Rept. UCID-18822 (1981).[b]
- Hallquist, J.O., "DYNA3D User's Manual (Nonlinear Dynamic Analysis of Solids in Three Dimensions)," University of California, Lawrence Livermore National Laboratory, Rept. UCID-19156 (1982; Rev. 1: 1984; Rev. 2: 1986).
- Hallquist, J.O., "Theoretical Manual for DYNA3D," University of California, Lawrence Livermore National Laboratory, Rept. UCID-19501 (March, 1983).
- Hallquist, J.O., "DYNA3D User's Manual (Nonlinear Dynamic Analysis of Solids in Three Dimensions)," University of California, Lawrence Livermore National Laboratory, Rept. UCID-19156 (1988, Rev. 4).
- Hallquist, J.O., "LS-DYNA User's Manual (Nonlinear Dynamic Analysis of Solids in Three Dimensions)," Livermore Software Technology Corporation, Rept. 1007 (1990).
- Hallquist, J.O., D.J. Benson, and G.L. Goudreau, "Implementation of a Modified Hughes-Liu Shell into a Fully Vectorized Explicit Finite Element Code," Proceedings of the International Symposium on Finite Element Methods for Nonlinear Problems, University of Trondheim, Trondheim, Norway (1985).
- Hallquist, J.O. and D.J. Benson, "A Comparison of an Implicit and Explicit Implementation of the Hughes-Liu Shell," Finite Element Methods for Plate and Shell Structures, T.J.R. Hughes and E. Hinton, Editors, 394-431, Pineridge Press Int., Swansea, U.K. (1986).
- Hallquist, J.O. and D.J. Benson, "DYNA3D User's Manual (Nonlinear Dynamic Analysis of Solids in Three Dimensions)," University of California, Lawrence Livermore National Laboratory, Rept. UCID-19156 (Rev. 2: 1986; Rev. 3: 1987).
- Hallquist, J.O., D.W. Stillman, T.J.R. Hughes, C. and Tarver,"Modeling of Airbags Using MVMA/DYNA3D," LSTC Report (1990).
- Herrmann, L.R. and F.E. Peterson, "A Numerical Procedure for Viscoelastic Stress Analysis," Seventh Meeting of ICRPG Mechanical Behavior Working Group, Orlando, FL, CPIA Publication No. 177, 1968.
- Hill A.V., "The heat of shortening and the dynamic constants of muscle," Proc Roy Soc B126:136-195, (1938).
- Hill, R., "A Theory of the Yielding and Plastic Flow of Anisotropic Metals," Proceedings of the Royal Society of London, Series A., Vol. 193, 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).

## REFERENCES

---

- 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 E. Carnoy, "Nonlinear Finite Element Shell Formulation Accounting for Large Membrane strains," *AMD-Vol.48, ASME*, 193-208 (1981).
- Hughes, T.J.R. and W.K. Liu, "Nonlinear Finite Element Analysis of Shells: Part I. Three-Dimensional Shells." *Comp. Meths. Appl. Mechs.*, 27, 331-362 (1981a).
- Hughes, T.J.R. and W.K. Liu, "Nonlinear Finite Element Analysis of Shells: Part II. Two-Dimensional Shells." *Comp. Meths. Appl. Mechs.*, 27, 167-181 (1981b).
- Hughes, T.J.R., W.K. Liu, and I. Levit, "Nonlinear Dynamics Finite Element Analysis of Shells." *Nonlinear Finite Element Analysis in Struct. Mech.*, Eds. W. Wunderlich, E. Stein, and K.J. Bathe, Springer-Verlag, Berlin, 151- 168 (1981c).
- Johnson, G.C. and D.J. Bammann, "A discussion of stress rates in finite deformation problems," *Int. J. Solids Struct*, 20, 725-737 (1984).
- Johnson, G.R. and W.H. Cook, "A Constitutive Model and Data for Metals Subjected to Large Strains, High Strain Rates and High Temperatures." Presented at the Seventh International Symposium on Ballistics, The Hague, The Netherlands, April 1983.
- 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).
- Lemaitre, J., *A Course on Damage Mechanics*, Springer-Verlag, (1992).
- MADYMO3D USER'S MANUAL, Version 4.3, TNO Road-Vehicles Research Institute, Department of Injury Prevention, The Hague, The Netherlands, (1990).
- Maker, B.N., Private communication Lawrence Livermore National Laboratory, Dr. Maker programmed and implemented the compressible Mooney Rivlin rubber model (1987).
- Matzenmiller, A. and J. K.Schm , "Crashworthiness Considerations of Composite Structures– A First Step with Explicit Time Integration in Nonlinear Computational Mechanics–State-of-the-Art," Ed. P. Wriggers, W. Wagner, Springer Verlay, , 1991.

## REFERENCES

---

- Neilsen, M.K., H.S. Morgan, and R.D. Krieg, "A Phenomenological Constitutive Model for Low Density Polyurethane Foams," Rept. SAND86-2927, Sandia National Laboratories, Albuquerque, N.M., (1987)
- Nusholtz, G., W. Fong, and J. Wu, "Air Bag Wind Blast Phenomena Evaluation," Experimental Techniques, Nov.-Dec. (1991).
- Nusholtz, G., D. Wang, and E.B. Wylie, "Air Bag Momentum Force Including Aspiration," Preprint, Chrysler Corporation, (1996).
- Nusholz, private communication, (1996).
- Papadrakakis, M., "A Method for the Automatic Evaluation of the Dynamic Relaxation Parameters," Comp. Meth. Appl. Mech. Eng., Vol. 25, 1981, pp. 35-48.
- 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).
- 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., (in press) (1988b).
- 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 from  $10^{-4}$  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).
- 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.
- Taylor, L.M. and D.P. Flanagan, "PRONTO3D A Three-Dimensional Transient Solid Dynamics Program," Sandia Report: SAND87-1912, UC-32, (1989).

## REFERENCES

---

- Tsai, S.W. and E.M. Wu, "A General Theory of Strength for Anisotropic Materials," *J. Composite Materials*, 5, 1971, pp. 73-96.
- 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).
- VDA Richtlinier (Surface Interfaces), Version 20, Verband der Automobilindustrie e.v., Frankfurt, Main, Germany, (1987).
- Wang, J.T. and O.J. Nefske, "A New CAL3D Airbag Inflation Model," SAE paper 880654, 1988.
- Wang, J.T., "An Analytical Model for an Airbag with a Hybrid Inflator", Publication R&D 8332, General Motors Development Center, Warren, Mi. (1995).
- Wang, J.T., "An Analytical Model for an Airbag with a Hybrid Inflator", AMD-Vol. 210, BED-Vol. 30, ASME, pp 467-497, (1995).
- 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).
- Winters J.M., "Hill-based muscle models: A systems engineering perspective," In *Multiple Muscle Systems: Biomechanics and Movement Organization*, JM Winters and SL-Y Woo eds, Springer-Verlag,1990.
- Winters J.M. and Stark L., "Estimated mechanical properties of synergistic muscles involved in movements of a variety of human joints," *J Biomechanics* 21:1027-1042, 1988.
- Woodruff, J.P., "KOVEC User's Manual," University of California, Lawrence Livermore National Laboratory, Rept. UCRL-51079 (1973).
- Zajac F.E., "Muscle and tendon: Properties, models, scaling, and application to biomechanics and motor control," *CRC Critical Reviews in Biomedical Engineering* 17(4):359-411, 1989.

---

**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 in 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 solid elements only 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.

# Appendix A

---

```
      SUBROUTINE UMAT41 (CM, EPS, SIG, HISV, DT1, CAPA, ETYPE, TIME)
C*****
C      LIVERMORE SOFTWARE TECHNOLOGY CORPORATION (LSTC)
C      -----
C      COPYRIGHT 1987-1994, 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 INCREMENT
C      EPS(2)=LOCAL Y STRAIN INCREMENT
C      EPS(3)=LOCAL Z STRAIN INCREMENT
C      EPS(4)=LOCAL XY STRAIN INCREMENT
C      EPS(5)=LOCAL YZ STRAIN INCREMENT
C      EPS(6)=LOCAL ZX STRAIN INCREMENT
C      EPS(1)=LOCAL X STRAIN INCREMENT
C
C      SIG(1)=LOCAL X STRESS
C      SIG(2)=LOCAL Y STRESS
C      SIG(3)=LOCAL Z STRESS
C      SIG(4)=LOCAL XY STRESS
C      SIG(5)=LOCAL YZ STRESS
C      SIG(6)=LOCAL ZX STRESS
C
C      HISV(1)=1ST HISTORY VARIABLE
C      HISV(2)=2ND HISTORY VARIABLE
C      .
C      .
C      .
C      HISV(N)=NTH HISTORY VARIABLE--SHALL NOT EXCEED VALUE GIVEN IN
C      *MAT_USER_DEFINED_MATERIAL_MODELS
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      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 EXITING 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
C      CHARACTER*(*) ETYPE
C      DIMENSION CM(*),EPS(*),SIG(*),HISV(*)
C
C      COMPUTE SHEAR MODULUS, G
C
C      G2=CM(1)/(1.+CM(2))
C      G =.5*G
C
C      IF (ETYPE.EQ.'BRICK') THEN
C      DAVG=(-EPS(1)-EPS(2)-EPS(3))/3.
C      P=-DAVG*CM(1)/((1.-2.*CM(2)))
C      SIG(1)=SIG(1)+P+G2*(EPS(1)+DAVG)
C      SIG(2)=SIG(2)+P+G2*(EPS(2)+DAVG)
C      SIG(3)=SIG(3)+P+G2*(EPS(3)+DAVG)
C      SIG(4)=SIG(4)+G*EPS(4)
C      SIG(5)=SIG(5)+G*EPS(5)
C      SIG(6)=SIG(6)+G*EPS(6)
C
C      ELSEIF (ETYPE.EQ.'SHELL') THEN
C
C      GC =CAPA*G
C      Q1 =CM(1)*CM(2)/((1.0+CM(2))*(1.0-2.0*CM(2)))
C      Q3 =1./(Q1+G2)
C      EPS(3)=-Q1*(EPS(1)+EPS(2))*Q3
C      DAVG =(-EPS(1)-EPS(2)-EPS(3))/3.
C      P =-DAVG*CM(1)/((1.-2.*CM(2)))
C      SIG(1)=SIG(1)+P+G2*(EPS(1)+DAVG)
C      SIG(2)=SIG(2)+P+G2*(EPS(2)+DAVG)
C      SIG(3)=0.0
C      SIG(4)=SIG(4)+G *EPS(4)
C      SIG(5)=SIG(5)+GC*EPS(5)
C      SIG(6)=SIG(6)+GC*EPS(6)
C
C      ELSEIF (ETYPE.EQ.'BEAM') THEN
C      Q1 =CM(1)*CM(2)/((1.0+CM(2))*(1.0-2.0*CM(2)))
C      Q3 =Q1+2.0*G
C      GC =CAPA*G
C      DETI =1./(Q3*Q3-Q1*Q1)
C      C22I = Q3*DETI
C      C23I =-Q1*DETI
C      FAC =(C22I+C23I)*Q1
C      EPS(2)=-EPS(1)*FAC-SIG(2)*C22I-SIG(3)*C23I
C      EPS(3)=-EPS(1)*FAC-SIG(2)*C23I-SIG(3)*C22I
C      DAVG =(-EPS(1)-EPS(2)-EPS(3))/3.
C      P =-DAVG*CM(1)/(1.-2.*CM(2))
C      SIG(1)=SIG(1)+P+G2*(EPS(1)+DAVG)
C      SIG(2)=0.0
C      SIG(3)=0.0
C      SIG(4)=SIG(4)+GC*EPS(4)
C      SIG(5)=0.0
C      SIG(6)=SIG(6)+GC*EPS(6)
C      ENDIF
C
C      RETURN
C      END

```





## APPENDIX B: User Defined Airbag Sensor

The addition of a user sensor subroutine into LS-DYNA is relatively simple. The sensor is mounted on a rigid body which is attached to the structure. The motion of the sensor is provided in the local coordinate system defined for the rigid body in the definition of material model 20—the rigid material. When the user defined criterion is met for the deployment of the airbag, a flag is set and the deployment begins. All load curves relating to the mass flow rate versus time are then shifted by the initiation time. The user subroutine is given below with all the necessary information contained in the comment cards.

```

SUBROUTINE AIRUSR (RBU,RBV,RBA,TIME,DT1,DT2,PARAM,HIST,ITRNON,
. RBUG,RBVG,RBAG)
C*****
C | LIVERMORE SOFTWARE TECHNOLOGY CORPORATION (LSTC) |
C | ----- |
C | COPYRIGHT 1987, 1988, 1989 JOHN O. HALLQUIST, LSTC |
C | ALL RIGHTS RESERVED |
C*****
C
C USER SUBROUTINE TO INITIATE THE INFLATION OF THE AIRBAG
C
C VARIABLES
C
C DISPLACEMENTS ARE DEFINED AT TIME N+1 IN LOCAL SYSTEM
C VELOCITIES ARE DEFINED AT TIME N+1/2 IN LOCAL SYSTEM
C ACCELERATIONS ARE DEFINED AT TIME N IN LOCAL SYSTEM
C
C RBU(1-3) TOTAL DISPLACEMENTS IN THE LOCAL XYZ DIRECTIONS
C RBV(3-6) TOTAL ROTATIONS ABOUT THE LOCAL XYZ AXES
C RBV(1-3) VELOCITIES IN THE LOCAL XYZ DIRECTIONS
C RBV(3-6) ROTATIONAL VELOCITIES ABOUT THE LOCAL XYZ AXES
C RBA(1-3) ACCELERATIONS IN THE LOCAL XYZ DIRECTIONS
C RBA(3-6) ROTATIONAL ACCELERATIONS ABOUT THE LOCAL XYZ AXES
C TIME IS THE CURRENT TIME
C DT1 IS TIME STEP SIZE AT N-1/2
C DT2 IS TIME STEP SIZE AT N+1/2
C PARAM IS USER DEFINED INPUT PARAMETERS (MAX 25)
C HIST IS USER DEFINED HISTORY VARIABLES (MAX 25)
C ITRNON IS FLAG TO TURN ON THE AIRBAG INFLATION
C RBUG,RBVG,RBAG, ARE SIMILAR TO RBU,RBV,RBA BUT ARE DEFINED
C GLOBALLY.
C
C THE USER SUBROUTINE SETS THE VARIABLE ITRNON TO:
C
C ITRNON=0 BAG IS NOT INFLATED
C ITRNON=1 BAG INFLATION BEGINS AND THIS SUBROUTINE IN NOT
C CALLED AGAIN
C
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. This subroutine is call each time step and does not need any control card to operate.

```

SUBROUTINE UCTRL1 (NUMNP, NDOF, TIME, DT1, DT2, PRTC, PLTC, FRCI, PRTO,
. PLTO, FRCO, VT, VR, AT, AR, UT, UR, XMST, XMSR, IRBODY, RBDYN, USRHV,
. MESSAG, TOTALM, CYCL, IDRINT)
C*****
C | LIVERMORE SOFTWARE TECHNOLOGY CORPORATION (LSTC) |
C | ----- |
C | COPYRIGHT 1987, 1988, 1989 JOHN O. HALLQUIST, LSTC |
C | ALL RIGHTS RESERVED |
C*****
C
C CHARACTER*(*) MESSAG
C 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

```

# Appendix C

---

```
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
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
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
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/DEENG/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 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 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)

```

## Appendix C

---

```
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
C   RETURN
C   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
C   RETURN
C   END
```

## APPENDIX D: User Defined Interface Control

This subroutine may be provided by the user to turn the interfaces on and off. This option is activated by the \*USER\_INTERFACE\_CONTROL keyword. The arguments are defined in the listing provided below.

```

SUBROUTINE UCTRL2 (NSI,NTY,TIME,CYCLE,MSR,NMN,NSV,NSN,
1 THMR,THSV,VT,XI,UT,ISKIP,IDRINT,NUMNP,DT2,NINPUT,UA)
C*****
C | LIVERMORE SOFTWARE TECHNOLOGY CORPORATION (LSTC) |
C | ----- |
C | COPYRIGHT 1987, 1988, 1989 JOHN O. HALLQUIST, LSTC |
C | ALL RIGHTS RESERVED |
C*****
C
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

```

# Appendix D

---

```
C      SET FLAG FOR ACTIVE CONTACT
C      ISKIP=0 ACTIVE
C      ISKIP=1 INACTIVE
C
C*****
C      DIMENSION MSR(*),NSV(*),THMR(*),THSV(*),VT(3,*),XI(3,*),
C              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
C      IF (NTY.EQ.4) RETURN
C      DT2HLF=DT2/2.
C      XMIN= 1.E20
C      XMAX=-XMIN
C      YMIN= 1.E20
C      YMAX=-YMIN
C      ZMIN= 1.E20
C      ZMAX=-ZMIN
C      XMINM= 1.E20
C      XMAXM=-XMINM
C      YMINM= 1.E20
C      YMAXM=-YMINM
C      ZMINM= 1.E20
C      ZMAXM=-ZMINM
C      THKS=0.0
C      THKM=0.0
C      DO 10 I=1,NSN
C      DSP1=UT(1,NSV(I))+DT2HLF*VT(1,NSV(I))
C      DSP2=UT(2,NSV(I))+DT2HLF*VT(2,NSV(I))
C      DSP3=UT(3,NSV(I))+DT2HLF*VT(3,NSV(I))
C      X1=XI(1,NSV(I))+DSP1
C      X2=XI(2,NSV(I))+DSP2
C      X3=XI(3,NSV(I))+DSP3
C      THKS =MAX(THSV(I),THKS)
C      XMIN=MIN(XMIN,X1)
C      XMAX=MAX(XMAX,X1)
C      YMIN=MIN(YMIN,X2)
C      YMAX=MAX(YMAX,X2)
C      ZMIN=MIN(ZMIN,X3)
C      ZMAX=MAX(ZMAX,X3)
10 CONTINUE
C      DO 20 I=1,NMN
C      DSP1=UT(1,MSR(I))+DT2HLF*VT(1,MSR(I))
C      DSP2=UT(2,MSR(I))+DT2HLF*VT(2,MSR(I))
C      DSP3=UT(3,MSR(I))+DT2HLF*VT(3,MSR(I))
C      X1=XI(1,MSR(I))+DSP1
C      X2=XI(2,MSR(I))+DSP2
C      X3=XI(3,MSR(I))+DSP3
C      THKM =MAX(THMR(I),THKS)
C      XMIN=MIN(XMINM,X1)
C      XMAX=MAX(XMAXM,X1)
C      YMIN=MIN(YMINM,X2)
C      YMAX=MAX(YMAXM,X2)
C      ZMIN=MIN(ZMINM,X3)
C      ZMAX=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. This option is activated by the \*USER\_INTERFACE\_FRICTION keyword. 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
      USER SUBROUTINE FOR INTERFACE FRICTION CONTROL
C
      NOTE:  LS-DYNA USES AN INTERNAL NUMBERING SYSTEM TO
      ACCOMODATE ARBITRARY NODE NUMBERING.  TO ACCESS
      INFORMATION FOR USER NODE N, ADDRESS ARRAY LOCATION M,
      M=LQF(N,1).  TO OBTAIN USER NODE NUMBER, N,
      CORRESPONDING TO ARRAY ADDRESS M, SET N=LQFINV(M,1)
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
C                   MASTER SEGMENT

```

# Appendix E

---

```
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
                DEFINED BY USER.  THE LENGTH OF THIS
                ARRAY IS DEFINED ON CONTROL CARD 15.
                THIS ARRAY IS UNIQUE TO INTERFACE NSI.
C
C   SIDE       ='MASTER' FOR FIRST PASS.  THE MASTER
                SURFACE IS THE SURFACE DESIGNATED IN THE
                INPUT.
C
                ='SLAVE' FOR SECOND PASS AFTER SLAVE AND
                MASTER SURFACES HAVE BE SWITCHED FOR
                THE TYPE 3 SYMMETRIC INTERFACE TREATMENT
C
C*****
C
C   RETURN
C   END
```

---

<b>APPENDIX F: Occupant Simulation Including the Coupling to Programs CAL3D and MADYMO</b>
--

## INTRODUCTION

LS-DYNA is coupled to occupant simulation codes to generate solutions in automotive crashworthiness that include occupants interacting with the automotive structure. In such applications LS-DYNA provides the simulation of the structural and deformable aspects of the model and the OSP (Occupant Simulation Program) simulates the motion of the occupant. There is some overlap between the two programs which provides flexibility in the modeling approach. For example, both the OSP and LS-DYNA have the capability of modeling seat belts and other deformable restraints. The advantage of using the OSP is related to the considerable databases and expertise that have been developed in the past for simulating dummy behavior using these programs.

The development of the interface provided LSTC a number of possible approaches. The approach selected is consistent with the LSTC philosophy of providing the most flexible and useful interface possible. This is important because the field of non-linear mechanics is evolving rapidly and techniques which are used today are frequently rendered obsolete by improved methodologies and lower cost computing which allows more rigorous techniques to be used. This does make the learning somewhat more difficult as there is not any single procedure for performing a coupling.

One characteristic of LS-DYNA is the large number of capabilities, particularly those associated with rigid bodies. This creates both an opportunity and a difficulty: LSDYNA3D has many ways approximating different aspects of problems, but they are frequently not obvious to users without considerable experience. Therefore, in this Appendix we emphasize modeling methods rather than simply listing capabilities.

## THE LS-DYNA/OCCUPANT SIMULATION PROGRAM LINK

Coupling between the OSP and LS-DYNA is performed by combining the programs into a single executable. In the case of CAL3D, LS-DYNA calls CAL3D as a subroutine, but in the case of MADYMO, LS-DYNA is called as a subroutine. The two programs are then integrated in parallel with the results being passed between the two until a user defined termination time is reached.

The OSP and LS-DYNA have different approaches to the time integration schemes. The OSP time integrators are based on accurate implicit integrators which are valid for large time steps which are on the order of a millisecond for the particular applications of interest here. An iterative solution is used to insure that the problem remains in equilibrium. The implicit integrators are extremely good for smoothly varying loads, however, sharp nonlinear pulses can introduce considerable error. An

## Appendix F

---

automatic time step size control which decreases the time step size quickly restores the accuracy for such events. The LS-DYNA time integrator is based on an explicit central difference scheme. Stability requires that the time step size be less than the highest frequency in the system. For a coarse airbag mesh, this number is on the order of 100 microseconds while an actual car crash simulation is on the order of 1 microsecond. The smallest LS-DYNA models have at least 1,000 elements. Experience indicates that the cost of a single LS-DYNA time step for a small model is at least as great as the cost of a time step in the OSP. Therefore, in the coupling, the LS-DYNA time step is used to control the entire simulation including the OSP part. This approach has negligible cost penalties and avoids questions of stability and accuracy that would result by using a subcycling scheme between the two programs. Optionally, a subcycling scheme can be used, however, the results of the analysis have to be checked with care.

LS-DYNA has a highly developed rigid body capability which is used in different parts of automobile crash simulation. In particular, components such as the engine are routinely modeled with rigid bodies. These rigid bodies have been modified so that they form the basis of the coupling procedure in LS-DYNA to the OSP.

In LS-DYNA, the geometry of a model is broken down into nodal points which identify positions in space. These nodes are then connected by elements so that the volume of a structure is identified. Each element has a “material” associated with it. If the element is deformable, then the material will specify its characteristics such as density and Young’s Modulus. A crash model can consist of 100 or more separate materials which are each assigned a “material number,” and each material number has an associated “material type” which determines if it is elastic, plastic, viscoelastic, orthotropic, etc.

The material type may also specify that it is a rigid body. In this case, all elements of the same material number are treated as a single rigid body. These elements are integrated to determine the mass, centroid and moments of inertia for the group. This group is then treated as a rigid body with six degrees-of-freedom including three translations and three rotations. The positions of the rigid bodies are updated in LS-DYNA by a time integrator which works together with the central difference time integration.

There is an additional flag which specifies that the LS-DYNA rigid body is coupled to an OSP rigid body. This flag can be found in the description of the rigid body material \*MAT\_RIGID (formerly material type 20). In coupled updates, the OSP rigid body time integrator takes over control of the LS-DYNA rigid body and the normal LS-DYNA updates are bypassed. The time integration procedure is then as follows:

1. At the beginning of a step, LS-DYNA determines the locations and updates the positions of all of the rigid bodies which are coupled to the OSP. This information is obtained from common block information in the OSP.
2. Using the information on rigid body locations, LS-DYNA proceeds to update the stresses and history variables of all of the deformable structures and computes the resultant forces acting on all rigid bodies.
3. The resultant forces are stored into an OSP common block along with the current time step. Control is then returned to the OSP so that the step can be completed by the OSP determining the new positions of the rigid bodies based on the applied forces.

At the end of the calculation LS-DYNA terminates normally, closing its files, and then control is returned to OSP which will also terminate normally. The termination time for the coupled run is taken as the minimum of the termination time provided to LS-DYNA and the termination time provided to the OSP.

The executable for the coupling with MADYMO currently needs to be specially created at each site. TNO provides all of the appropriate load modules with their libraries, and the appropriate load modules for LS-DYNA may be obtained by the corporate contact point at the LS-DYNA distributor. A complete executable must then be made by linking the two libraries. A revised password file must be obtained from TNO prior to running the coupled code. Coupling with CAL3D requires special on-site modification of the client's CAL3D version to eliminate conflicting I/O unit numbers and to ensure that the common block lengths between the codes are consistent. LSTC does not distribute or support CAL3D.

To make the coupled program run, an input deck must be provided to both the OSP and LS-DYNA3D. The two input decks must be provided in the same set of consistent units. This can potentially require a major conversion to either the OSP input or the LS-DYNA input. With two legitimate and consistent input decks, the coupled program should run to completion with no problems. Additional inputs are required to make the models interact between the OSP and LS-DYNA3D portions of the run.

The simplest form of a coupled simulation is simply to include a single body in an OSP run. No special modifications are needed to the OSP input deck for use in the coupled simulation. Ellipsoids and planes in the OSP are usually attached to "segments" which correspond to LS-DYNA3D "rigid bodies." Because the coupling procedure works on the basis of shared information on LS-DYNA rigid bodies with the OSP segments, the ellipsoids/planes listed in the OSP section must correspond to the segments which are to be coupled. These ellipsoids and planes

may be actual geometry which is used for contact, or they may be simply artificial shapes to permit the data transfer between the OSP and LS-DYNA.

### DUMMY MODELING

The dummy is typically modeled entirely within the OSP. The coupling of the dummy into LS-DYNA requires the creation of a separate LS-DYNA rigid body material for each segment of the OSP. The easiest way to create a mesh for the model is to set the LS-DYNA rigid body coupling option to 2.0. This causes LS-DYNA to search all of the ellipsoids connected to the appropriate segment and generate meshes which are then slaved to the OSP dummy. Thus, with minimal input, a complete dummy may be generated and the kinematics may be traced in LS-DYNA3D and displayed in the LS-DYNA post-processor, LS-POST

Once the basic dummy coupling has been accomplished, the deformable finite element structure can be added. Assuming that an ellipsoid is available for the steering wheel, a flat airbag can be added in the proper location. One or more nodes must be attached to the steering wheel. This is done by identifying the attached nodes as “Extra Nodes for Rigid Body” which is input in LS-DYNA3D by *\*CONSTRAINED\_EXTRA\_NODES*\_Option. The nodes are slaved to the LS-DYNA3D material which has been coupled to the MADYMO steering wheel model. Contact must now be identified between the airbag and the steering wheel, the windshield, and the various body parts which may be affected. This requires the use of one geometric contact entity (see *\*CONTACT\_ENTITY*) for each plane or ellipsoid which may interact with the airbag. A control volume specifying inflation properties for the airbag must be specified (see *\*AIRBAG\_OPTION*) to complete the model.

### AIRBAG MODELING

Modeling of airbags is accomplished by use of shell or membrane elements in conjunction with a control volume (see *\*AIRBAG\_OPTION*) and possibly a single surface contact algorithm to eliminate interpenetrations during the inflation phase (see *\*CONTACT\_OPTION*). The contact types showing an “a” in front are most suited for airbag analysis. Current recommended material types for the airbags are:

*\*MAT\_ELASTIC* = Type 1. Elastic

*\*MAT\_COMPOSITE\_DAMAGE* = Type 22. Layered orthotropic elastic for composites

*\*MAT\_FABRIC* = Type 34. Fabric model for folded airbags



Model 34 is a “fabric” model which can be used for flat bags. As a user option this model may or may not support compression.

The elements which can be used are as follows:

Belytschko-Tsay quadrilateral with 1 point quadrature. This element behaves rather well for folded and unfolded cases with only a small tendency to hourglass. The element tends to be a little stiff. Stiffness form hourglass control is recommended.

Belytschko-Tsay membrane. This model is softer than the normal Belytschko-Tsay element and can hourglass quite badly. Stiffness form hourglass is recommended. As a better option, the fully integrated Belytschko-Tsay membrane element can be chosen.

C0 Triangular element. The C0 triangle is very good for flat bag inflation and has no tendency to hourglass.

The best choice is a specially developed airbag membrane element with quadrilateral shape. This is an automatic choice when the fabric material is used.

As an airbag inflates, a considerable amount of energy is transferred to the surrounding air. This energy transfer decreases the kinetic energy of the bag as it inflates. In the control volume logic, this is simulated either by using either a mass weighted damping option or a back pressure on the bag based on a stagnation pressure. In both cases, the energy that is absorbed is a function of the fabric velocity relative to a rigid body velocity for the bag. For the mass weighted case, the damping force on a node is proportional to the mass times the damping factor times the velocity vector. This is quite effective in maintaining a stable system, but has little physical justification. The latter approach using the stagnation pressure method estimates the pressure needed to accelerate the surrounding air to the speed of the fabric. The formula for this is:

$$P = Area \times \alpha \times \left( (\vec{V}_i - \vec{V}_{cg}) \cdot \hat{n} \right)^2$$

This formula accomplishes a similar function and has a physical justification. Values of the damping factor,  $\alpha$ , are limited to the range of 0 to 1, but a value of 0.1 or less is more likely to be a good value.

### KNEE BOLSTER

The knee-to-knee bolster interactions are characterized by the stiffness of the knee being comparable to that of the knee bolster. Therefore, modeling the knee as a rigid body may produce large errors in the interaction forces. Calibrated force-deflection curves could be determined, but they would have no predictive value for slight changes to knee bolster designs. For this reason, a more accurate modeling of the compliance of the knee bolster and the knee is required.

The knee can be modeled as a combined rigid/deformable body. The rigid body is coupled to the OSP. Overlaying the rigid body are brick elements which model the “skin” that exists over the knees of the dummy. These brick elements use material type 6 (\*MAT\_VISCOELASTIC) which is a viscoelastic model that does a reasonable job of approximating the hysteretic behavior of rubbers. The inner layer of the brick elements is attached to the rigid body through the \*CONSTRAINED\_EXTRA\_NODES Option. Between the knee bolster is a SURFACE-TO-SURFACE contact definition.

### COMMON ERRORS

#### 1. **Improper airbag inflation or no inflation.**

The most common problem is inconsistency in the units used for the input constants. An inflation load curve must also be specified. The normals for the airbag segments must all be consistent and facing outwards. If a negative volume results, this can sometimes be quickly cured by using the “flip” flag on the control volume definition to force inward facing normals to face outwards.

#### 2. **Excessive airbag distortions.**

Check the material constants. Triangular elements should have less distortion problems than quadrilaterals. Overlapped elements at time zero can cause locking to occur in the contact leading to excessive distortions. The considerable energy input to the bag will create numerical noise and some damping is recommended to avoid problems.

### **3. The dummy passes through the airbag.**

A most likely problem is that the contacts are improperly defined. Another possibility is that the models were developed in an incompatible unit system. The extra check for penetration flag if set to 1 on the contact control cards variable PENCHK in the \*CONTACT\_... definitions may sometimes cause nodes to be prematurely released due to the softness of the penalties. In this case the flag should be turned off.

### **4. The OSP fails to converge.**

This may occur when excessively large forces are passed to the OSP. First, check that unit systems are consistent and then look for improperly defined contacts in the LS-DYNA input.

### **5. Time step approaches zero.**

This is almost always in the airbag. If elastic or orthotropic (\*MAT\_ELASTIC or \*MAT\_COMPOSITE material 1 or 22) is being used, then switch to fabric material \*MAT\_FABRIC which is less time step size sensitive and use the fully integrated membrane element. Increasing the damping in the control volume usually helps considerably. Also, check for “cuts” in the airbag where nodes are not merged. These can allow elements to deform freely and cut the time step to zero.



**APPENDIX G: Interactive Graphics Commands**

Only the first four or less characters of command are significant. These commands are available in the interactive phase of LS-DYNA. The interactive graphics are available by using the "SW5." command after invoking the Ctrl-C interrupt. The MENU command brings up a push button menu.

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.

## Appendix G

---

CX	Rotate slice plane at zmin about x axis.
CY	Rotate slice plane at zmin about y axis.
CZ	Rotate slice plane at zmin about z axis.
DIF	Change diffused light level for material; DIF <mat #, -1 for all> <value>.
DISTANCE	Set distance of model from viewer; DIST <value in normalized model dimensions>.
DMATERIALS	Delete display of material in subsequent views; DMAT <ALL or list of numbers>.
DRAW	Display outside edges of model.
DSCALE	Scale current displacement from initial shape.
DYN	After using TAURUS command will reset display to read current DYNA3D state data.
ELPLT	Set or unset element numbering in subsequent views.
END	Delete display and return to execution.
ESCAPE	Escapes from menu pad mode.
EXECUTE	Return to execution and keep display active.
FCL	Fix or unfix current contour levels.
FOV	Set display field of view angle; FOV <value in degrees>.
FRINGE	View with colored contour fringes; fringe <component #> <list mat #>; see TAURUS manual.
GETFRAME	Display a saved frame; GETF <frame #>.

HARDWARE	Hardware mode; workstation hardware calls are used to draw, move and color model; repeat command to reset to normal mode.
HELP	
HZB	Switch on or off hardware zbuffer for a subsequent view, draw or contour command; rotations and translations will be in hardware.
LIMIT	Set range of node numbers subsequent views; limit <first node #> <last node #>.
MAT	Re-enable display of deleted materials mat <all or list of numbers>.
MENU	Button menu pad mode.
MOTION	Motion of model through mouse movement or use of a dial box. The left button down enables translation in the plane, middle button rotation about axes in the plane; and with right button down in the out of plane axis; left and middle button down quit this mode.
MOV	Drag picked part to new position set with mouse.
NDPLT	Set or unset node numbering in subsequent views.
NOFRAME	Set and unset drawing of a frame around the picture.
PAUSE	Animation display pause in seconds
PHS2 or THISTORY	Time history plotting phase. Similar to LS-TAURUS.
PICK	Get element information with mouse.
POST	Enable or disable postscript mode on the PC and eps file is written as picture is drawn; remove eof and initgraphics for eps use.

## Appendix G

---

QUIT	Same as execute.
RANGE	Set fix range for contour levels; range <minvalue> <maxvalue>.
RAX	Reflect model about xy plane; restore command will switch-off reflections.
RAY	Reflect model about yz plane; restore command will switch-off reflections.
RAZ	Reflect model about zx plane, restore command will switch-off reflections.
RESTORE	Restores model to original position, also switches off element and node numbers, slice capper, reflections and cut model.
RETURN	Exit.
RGB	Change color red green blue element <mat #> <red> <green> <blue>.
RX	Rotate model about x axis.
RY	Rotate model about y axis.
RZ	Rotate model about z axis.
SAVE	Set or unset saving of display for animation.
SEQUENCE	Periodic plot during execution; SEQ <# of cycles> <commands> EXE.
SHR	Shrink element facets towards centroids in subsequent views, shrink <value>.
SIP	Select shell integration point for contour; SIP <#>.



SLICE	Slice model a z-minimum plane; slice <value in normalized model dimension> this feature is removed after using restore. Slice enables internal details for brick elements to be used to generate new polygons on the slice plane.
SNORMAL	Set or unset display of shell direction normals to indicate topology order.
SPOT	Draw node numbers on model spot <first #> <last # for range>.
TAURUS	LS-DYNA database, TAU <state #>, or state <state #>, reads LS-TAURUS file to extract previous state data.
TRIAD	Set or unset display of axis triad.
TSHELL	Set or unset shell element thickness simulation in subsequent views.
TV	Change display type.
TX	Translates model along x axis.
TY	Translates model along y axis.
TZ	Translates model along z axis.
V	Display model using painters algorithm.
VECTOR v or d	View with vector arrows of velocity or displacement; <v> or <d>.
ZB	Switch on or off zbuffer algorithm for subsequent view; or draw commands.
ZIN	Zoom in using mouse to set display size and position.
ZMA	Set position of zmax plane; ZMAX <value in normalized model dimesions>.

## Appendix G

---

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 H: 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-DYNA3D are directly called by this driver, the behavior of the constitutive model is precisely that which can be expected in actual applications. In the current implementation the constitutive subroutines for both shell elements and solid elements can be examined.

**INPUT DEFINITION**

The material model driver is invoked by setting the total number of beam, shell, and solid elements to zero in a standard LS-DYNA input file. The number of material model definitions should be set to one, the number of load curves should be nine, and the termination time to the desired length of the driver run. The complete state dump interval is interpreted as the time step to be used in the material model driver run. Plotting information is saved for every step of a driver run and sufficient memory is allocated to save this information in core for the interactive plotting phase.

The input deck consists only of the TITLE card, the CONTROL cards, one MATERIAL DEFINITION, and NINE LOAD CURVES describing the strain path should be defined. These nine curves define the time history of the displacement gradient components shown in Table H.1.

The velocity gradient matrix,  $L_{ij}$ , is approximated by taking the time derivative of the components in Table H.1. If these components are considered to form a tensor  $S_{ij}$ , then

$$L_{ij}(t) = \frac{S_{ij}(t) - S_{ij}(t_{k-1})}{(t - t_k)}$$

and the strain rate tensor is defined as

$$d_{ij} = \frac{L_{ij} + L_{ij}^t}{2}$$

and the spin tensor as

$$\omega_{ij} = \frac{L_{ij} - L_{ij}^t}{2}$$

**Table H.1.** Load Curve Definitions versus Time

Load Curve Number	Component Definition
1	$\frac{\partial u}{\partial x}$
2	$\frac{\partial v}{\partial y}$
3	$\frac{\partial w}{\partial z}$
4	$\frac{\partial u}{\partial y}$
5	$\frac{\partial v}{\partial x}$
6	$\frac{\partial u}{\partial z}$
7	$\frac{\partial w}{\partial x}$
8	$\frac{\partial v}{\partial z}$
9	$\frac{\partial w}{\partial y}$

---

## INTERACTIVE DRIVER COMMANDS

After reading the input file and completing the calculations, LS-DYNA gives a command prompt to the terminal. A summary of the available interactive commands is given below. An on-line help package is available by typing HELP.

ACCL	Scale all abscissa data by f. Default is f=1.
ASET amin omax	Set min and max values on abscissa to amin and amax, respectively. If amin=amax=0, scaling is automatic.
CHGL n	Change label for component n. LS-DYNA prompts for new label.
CONTINUE	Re-analyze material model.
CROSS c1 c2	Plot component c1 versus c2.
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 $\ln(v/v_0)$ ,

## Appendix H

---

- 18 relative volume,
- 19  $v_0/v - 1.0$ ,
- 20 1st history variable,
- 21 2nd history variable.

Adding 100 or 400 to component numbers 1-16 yields strains and strain rates, respectively.

FILE name	Change pampers filename to name for printing.
GRID	Graphics displays will be overlaid by a grid of orthogonal lines.
NOGRID	Graphics displays will not be overlaid by a grid of orthogonal lines.
OSCL	Scale all ordinate data by f. Default is f=1.
OSET omin omax	Set min and max values on ordinate to omin and omax, respectively. If omin=omax=0, scaling is automatic.
PRINT	Print plotted time history data into file "pampers." Only data plotted after this command is printed. File name can be changed with the "file" command.
QUIT, END, T	Exit the material model driver program.
RDLC m n r <sub>1</sub> z <sub>1</sub> ... r <sub>n</sub> z <sub>n</sub>	Redefine load curve m using n coordinate pairs (r <sub>1</sub> ,z <sub>1</sub> ), (r <sub>2</sub> ,z <sub>2</sub> ),...,(r <sub>n</sub> ,z <sub>n</sub> ).
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 driver is implemented for solid and shell element material models. The driver does not yet support material models for beam elements.

---

**APPENDIX I: VDA Database**

---

VDA surfaces describe the surface of geometric entities and are useful for the simulation of sheet forming problems. The German automobile and automotive supplier industry (VDA) has defined the VDA guidelines [VDA, 1987] for a proper surface definition used for the exchange of surface data information. In LS-DYNA, this format can be read and used directly. Some files have to be provided for proper linkage to the motion of the correlation parts/materials in LS-DYNA3D.

Linking is performed via names. To these names surfaces are attached, which in turn can be linked together from many files externally to LS-DYNA. Thus, arbitrary surfaces can be provided by a preprocessor and then can be written to various files. The so called VDA file given on the LS-DYNA3D execution line via `V=vda` contains references to all other files. It also contains several other parameters affecting the treatment in the contact subroutines; see below.

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

```
file afile bfile {  
    alias definitions  
    }  
    alias definitions  
    followed by optional runtime parameters and a final end statement.
```

The file, **afile**, is optional, and if given must be the name of an ASCII input file formatted in accordance with the VDA Surface Interface Definitions as defined by the German automobile and automotive supply industry. **bfile** is required, and is the name of a binary VDA file. In a first run **afile** is given and **bfile** is created. In any further run, if the definitions have not changed, **afile** can be dropped and only **bfile** is needed. The purpose of **bfile** is that it allows for much faster initialization if the same VDA surfaces are to be used in a future LS-DYNA run.

If **afile** is given, **bfile** will always be created or overwritten. The alias definitions are used for linking to LS-DYNA and between the various surface definitions in the files defined by **afile** and **bfile**.

The alias definitions are of the form

# Appendix I

---

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

where **name** is any string of up to 12 characters, and el1,...,eln are the names of VDA elements as specified in **afile**. The list of elements can be empty, in which case all the SURF and FACE VDA elements in **afile** will be used. Care should be taken to ensure that the alias **name** is unique, not only among the other aliases, but among the VDA element names in **afile**. This collection of VDA elements can later be indicated by the alias **name**. In particular, **name** may appear in later alias definitions.

Often it is required that a punch or die be created by a simple offset. This can be achieved in the **vda** files in two ways, either on VDA elements directly, or on parts defined by aliases. This feature offers great capability in generating and using surface data information.

## Offset version 1:

As an option, the keyword **offset** may appear in the alias list which allows a new surface to be created as a normal offset (plus translation) of a VDA element in the file. The keyword **offset** may be applied to VDA elements only, not aliases. The usage of **offset** follows the form

offset elem **normal x y z**

where **normal** is the amount to offset the surface along the normal direction, and **x,y,z** are the translations to be applied. The default normal direction is given by the cross product of the local u and v directions on the VDA surface, taken in that order. **normal** can be negative.

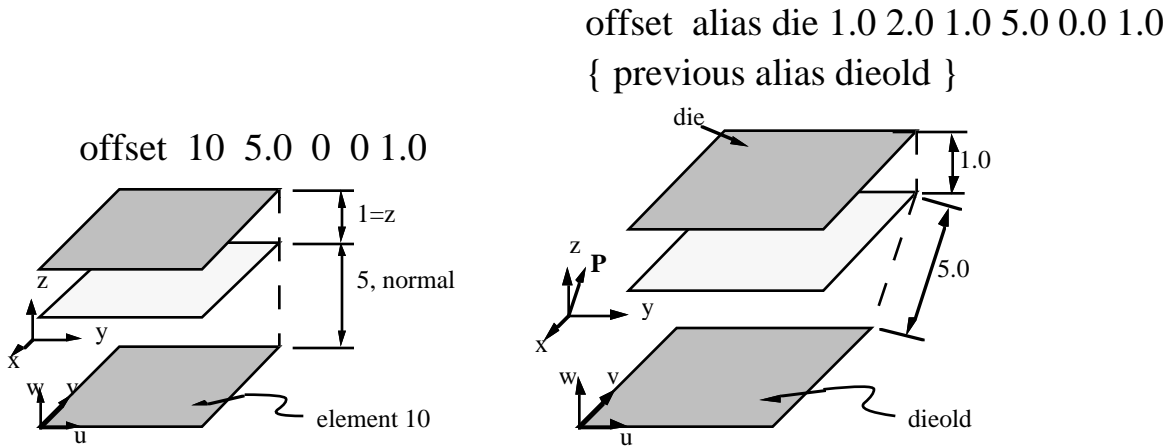
## Offset version 2:

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

goffset **alias\_name x<sub>c</sub> y<sub>c</sub> z<sub>c</sub> normal x y z { previous alias\_name }**

where **normal**, **x**, **y**, and **z** are defined as in the offset keyword. A reference point **x<sub>c</sub>**, **y<sub>c</sub>**, and **z<sub>c</sub>** defines a point in space which determines the normal direction to the VDA surface, which is a vector from the origin to P(x<sub>c</sub>,y<sub>c</sub>,z<sub>c</sub>). See example below.





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

- gap** [5.0]      The maximum allowable surface gap to be filled in during the iterations. Points following the surface will effectively extend the edges of surfaces if necessary to keep them from falling through cracks in the surface smaller than this. This number should be set as small as possible while still allowing correct results. In particular, if your VDA surfaces are well formed (having no gaps), this parameter can be set to 0.0. The default value is 5.0.
  
- 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. The default value is 2.0.
  
- 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 than if the actual distance to the surface were known, but also allows many global searches to be skipped. The default value is 5.0.
  
- ntrack** [4]      The number of VDA surfaces for which each point maintains actual distance information. A global lower bound on distance is maintained for all remaining surfaces. Whenever the point moves far enough to violate this global lower

# Appendix I

---

bound, all VDA surfaces must have the global search performed for them. Hence, this parameter should be set to the maximum number of surfaces that any point can be expected to be near at one time (the largest number of surfaces that come together at one point). Setting **ntrack** higher will require more memory but result in faster execution. If **ntrack** is too low, performance may be unacceptably slow. The default value is 4.0.

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

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

**iterate** [8] Maximum number of surface iterations allowed. Since points being tracked are checked every cycle, if convergence fails it will be tried again next cycle, so setting this parameter high does not necessarily help much. On the other hand, a point converging to a crease in the VDA surface (a crease between patches with discontinuous derivative, for example) may bounce back and forth between patches up to this many times, without actually moving. Hence, this value should not be too large. The default value is 8.

**el\_size** [t mx mn]

Controls the generation of elements where:

t =surface tolerance for mesh generation,

mx=maximum element size to generate,

mn=minimum element size to generate.

The default values are [0.25 100. 1.0]

**aspect** [s1 s2]

Controls the generation of elements where:

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

s2=maximum aspect ratio for any generated element.

The default values are [1.5 4.0]

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

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

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

**somepatch** [n] Like onepatch, but generates an element for 1 out of every [n] patches.

Example for file V=**vda**. It contains the following data:

```

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

```

**Explanation:**

**vda1** This file contains the surfaces/face elements sur0001, sur0003, fce0006, and sur0008.

# Appendix I

---

**alias die**            Combines the surface/face elements sur0001, sur0003, and the offsetted surface  
face                    fce0006 to a global surface.

**alias holder1**        Defines the surface/face element sur0008 as holder1.

**vda2**                    This file contains the surface/face element sur0003.

**alias holder2**        Defines the surface/face element sur0003 as holder2.

**alias holder**         Combines the surfaces holder1 and holder2 into a combined surface holder.

**ntrack 6**              For each point the actual distances to 6 VDA surfaces are maintained.

**gap 0.5**                Surface gaps of 0.5mm or less are filled.

**end**                     Closes reading of this file.

---

**APPENDIX 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 $\Delta t$	Reset the node and element data dump interval $\Delta t$ .
PRTI $\Delta t$	Reset the node and element printout interval $\Delta t$ .
TERM t	Reset the termination to t.

---

**Graphics Window Controls**


---

ESET n	Center picture at element n with a $\Delta r$ by $\Delta z$ 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 $\Delta r$ $\Delta z$	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 $\Delta l$	Center display picture at point (r,z) with square window of width $\Delta l$ . 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 $\Delta r$ $\Delta z$	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 $\Delta l$	Zoom in at point (a,b) with window $\Delta l$ where a, b, and $\Delta l$ 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 $\Delta l$	Zoom out at point (a,b) with window $\Delta l$ where a, b, and $\Delta l$ 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.

## Appendix J

---

Z r z $\Delta$ l	Zoom in at point (r,z) with window $\Delta$ l.
ZOUT r z $\Delta$ l	Zoom out at point (r,z) with window $\Delta$ l. The window is enlarged by the ratio of the current window and $\Delta$ l. 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 omx	Set minimum and maximum values on ordinate to omin and omx, respectively. If omin=omx=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 $0 \leq s \leq 1$ a combined blending is obtained.
CN m r z	Node m has new coordinate (r,z).
DEB n f <sub>1</sub> l <sub>1</sub> ... f <sub>n</sub> l <sub>n</sub>	Delete n element blocks consisting of element numbers f <sub>1</sub> to l <sub>1</sub> , f <sub>2</sub> to l <sub>2</sub> ... , and f <sub>n</sub> to l <sub>n</sub> inclusive. These elements will be inactive when the calculation resume.
DE e <sub>1</sub> e <sub>2</sub>	Delete elements e <sub>1</sub> to e <sub>2</sub> .
DMB n m <sub>1</sub> m <sub>2</sub> ... m <sub>n</sub>	Delete n material blocks consisting of all elements with material numbers m <sub>1</sub> , m <sub>2</sub> ,..., and m <sub>n</sub> . These materials will be inactive when the calculations resume.
DM n m <sub>1</sub> m <sub>2</sub> ... m <sub>n</sub>	Delete n materials including m <sub>1</sub> , m <sub>2</sub> ,..., and m <sub>n</sub> .
DZER k d incr nrow	Delete element row where k is the kept element, d is the deleted element, incr is the increment, and nrow is the number of elements in the row.
DZLN number n <sub>1</sub> n <sub>2</sub> n <sub>3</sub> ...n <sub>last</sub>	Delete nodal row where number is the number of nodes in the row and n <sub>1</sub> , n <sub>2</sub> , ... n <sub>last</sub> are the ordered list of deleted nodes.

## Appendix J

---

DZNR l j incr	Delete nodal row where l is the first node in the row, j is the last node in the row, and incr is the increment.
R	Restore original mesh.
S	Smooth mesh of material n. To smooth a subset of elements, a window can be set via the “GSET”, “FSET”, OR “SETF” commands. Only the elements lying within the window are smoothed.

### Boundary Modifications

---

A	Display all slidelines. Slave sides are plotted as dashed lines.
B	Determine boundary nodes and sides of material n and display boundary with nodes and side numbers.
BD m n	Dekink boundary from boundary node m to boundary node n (counterclockwise).
BDS s	Dekink side s.
DSL n l <sub>1</sub> l <sub>2</sub> ...l <sub>n</sub>	Delete n slidelines including slideline numbers l <sub>1</sub> l <sub>2</sub> ..., and l <sub>n</sub> .
ER m n	Equal space in r-direction boundary nodes m to n (counterclockwise).
ERS s	Equal space in the r-direction boundary nodes on side s.
ES m n	Equal space along boundary, boundary nodes m to n (counterclockwise).
ESS s	Equal space along boundary, boundary nodes on side s.
EZ m n	Equal space in z-direction boundary nodes m to n (counterclockwise).
EZS s	Equal space in the z-direction boundary nodes on side s.
MC n	Check master nodes of slideline n and put any nodes that have penetrated through the slave surface back on the slave surface.
MD n	Dekink master side of slideline n. After using this command, the SC or MC command is sometimes advisable.
MN n	Display slideline n with master node numbers.
SC n	Check slave nodes of slideline n and put any nodes that have penetrated through the master surface back on the master surface.

---

SD n	Dekink slave side of slideline n; after using this command, the SC or MC command is sometimes advisable.
SLN m n	Equal space boundary nodes between nodes m to n on a straight line connecting node m to n.
SLNS n	Equal space boundary nodes along side n on a straight line connecting the corner nodes.
SN n	Display slideline n with slave node numbers.
VS m n r	Vary the spacing of boundary nodes m to n such that r is the ratio of the first segment length to the last segment length.
VSS s r	Vary the spacing of boundary nodes on side s such that r is the ratio of the first segment length to the last segment length.

## **MAZE Line Definitions**

---

B	Determine boundary nodes and sides of material n and display boundary with nodes and side numbers. See command "M".
LD n k l	Line definition n for MAZE includes boundary nodes k to l
LDS n l	Line definition n for MAZE consists of side number l.
M n	Material n is active for the boundary command B.

## **Calculation Graphics Display Control Parameters**

---

MOLP	Overlay the mesh on the contour, fringe, principal stress, and principal strain plots. Retyping "MOLP" turns this option off.
NLOC	Do not plot letters on contour lines.
NUMCON n	Plot n contour levels. The default is 9.
PLOC	Plot letters on contour lines to identify their levels (default).
RANGE r <sub>1</sub> r <sub>2</sub>	Set the range of levels to be between r <sub>1</sub> and r <sub>2</sub> instead of in the range chosen automatically by LS-DYNA. To deactivate this command, type RANGE 0 0.

# Appendix J

---

## Calculation Graphics Display

---

CONTOUR c n m <sub>1</sub> m <sub>2</sub> ...m <sub>n</sub>	Contour component number c on n materials including materials m <sub>1</sub> , m <sub>2</sub> , ..., m <sub>n</sub> . If n is zero, only the outline of material m <sub>1</sub> with contours is plotted. Component numbers are given in Table 1.
FRINGE c n m <sub>1</sub> m <sub>2</sub> ...m <sub>n</sub>	Fringe component number c on n materials including m <sub>1</sub> , m <sub>2</sub> ,...,m <sub>n</sub> . If n is zero, only the outline of material m <sub>1</sub> 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 <sub>c</sub> z <sub>c</sub>	Plot the angular location of the interface based on the center point (r <sub>c</sub> ,z <sub>c</sub> ) 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 <sub>1</sub> m <sub>2</sub> ...m <sub>n</sub>	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 <sub>1</sub> , m <sub>2</sub> ,...,m <sub>n</sub> .
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 <sub>1</sub> n <sub>2</sub> ...n <sub>3</sub>	Define line for “LINE” command using n nodes including node numbers n <sub>1</sub> , n <sub>2</sub> ,...,n <sub>n</sub> . 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 <sub>1</sub> z <sub>1</sub> ...r <sub>n</sub> z <sub>n</sub>	Define line for “LINE” command using n coordinate pairs (r <sub>1</sub> ,z <sub>1</sub> ), (r <sub>2</sub> ,z <sub>2</sub> ), ...(r <sub>n</sub> ,z <sub>n</sub> ). This line is fixed in space.

PRIN c n m <sub>1</sub> m <sub>2</sub> ...m <sub>n</sub>	Plot lines of principal stress and strain in the yz plane on n materials including materials m <sub>1</sub> , m <sub>2</sub> ,...,m <sub>n</sub> . If n is zero, only the outline of material m <sub>1</sub> 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 <sub>1</sub> m <sub>2</sub> ...m <sub>n</sub>	Plot component c versus element number for n materials including materials m <sub>1</sub> , m <sub>2</sub> ,...,m <sub>n</sub> . If n is 0 then component c is plotted for all elements. Component numbers are given in Table 1.
VECTOR c n m <sub>1</sub> m <sub>2</sub> ...m <sub>n</sub>	Make a vector plot of component c on n materials including materials m <sub>1</sub> , m <sub>2</sub> ,...,m <sub>n</sub> . If n is zero, only the outline of material m <sub>1</sub> with vectors is plotted. Component c may be set to “D” and “V” for vector plots of displacement and velocity, respectively.

# Appendix J

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*	VoV-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.

## Appendix J

---

DVS a b r	Use cursor to define points a and b on boundary. Variable space nodes along boundary starting at a, moving counterwise, and ending at b. The ratio of the first segment length to the last segment length is give by r (via terminal).
DZ a b	Use cursor to define points a and b on the diagonal of a window for zooming.
DZOUT a b	Enter two points with the cursor to define the window. The ratio of the current window with the specified window determines the picture size reduction.
DZZ a	Use cursor to define point a and zoom in at this point. The new window is .15 as large as the previous window. The zoom factor can be reset by the crzf command for the .15 default.
DZZO a	Zoom out at point a by enlarging the picture two times.



**APPENDIX K: Rigid Body Dummies**

Rigid body dummies can be generated and simulated within LS-DYNA using the keyword `*COMPONENT_GEBOD`. Physical properties of these dummies draw upon the GEBOD database [Cheng et al, 1994] which represents an extensive measurement program conducted by Wright-Patterson AFB and other agencies. The differential equations governing motion of the dummy are integrated within LS-DYNA separate from the finite element model. Interaction between the dummy and finite element structure is achieved using contact interfaces (see `*CONTACT_GEBOD`).

The dynamical system representing a dummy is comprised of fifteen rigid bodies (segments) and include: lower torso, middle torso, upper torso, neck, head, upper arms, forearms/hands, upper legs, lower legs, and feet. Ellipsoids are used for visualization and contact purposes. Shown in Figure K.1 is a 50th percentile male dummy generated using the keyword command `*COMPONENT_GEBOD_MALE`. Note that the ellipsoids representing the shoulders are considered to be part of the upper torso segment and the hands are rigidly attached to the forearms.

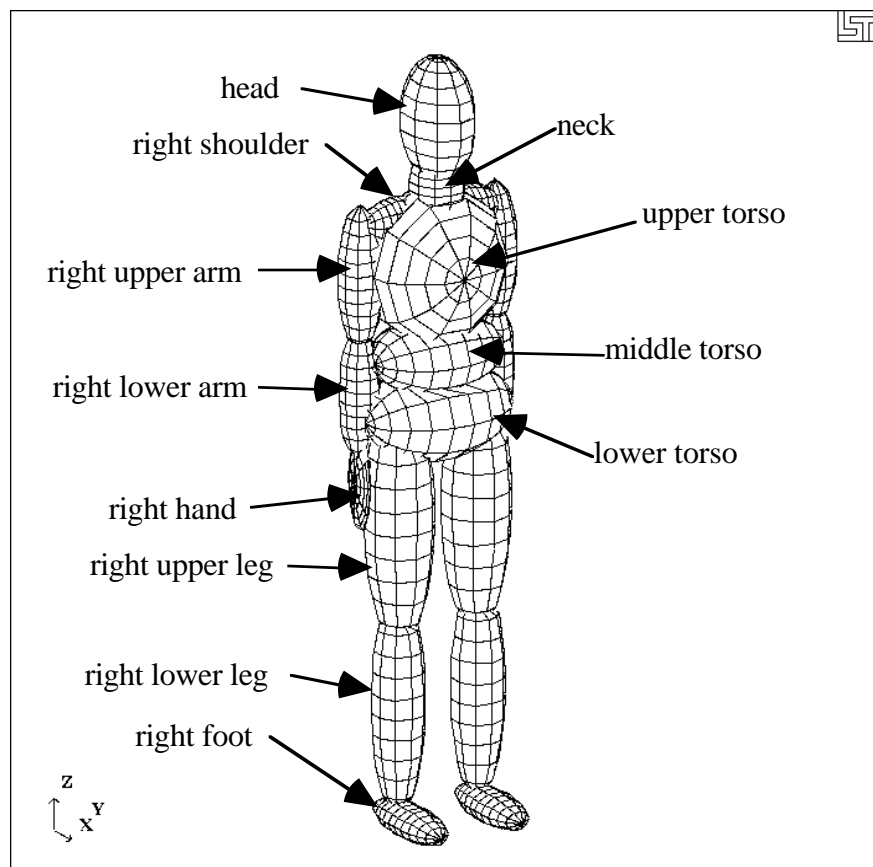


Figure K.1: 50th percentile male dummy in the nominal position.

# Appendix K

---

Each of the rigid segments which make up the dummy is connected to its neighbor with a joint which permits various relative motions of the segments. Listed in the Table K.1 are the joints and their applicable degrees of freedom.

Table K.1: Joints and associated degrees of freedom. Local axes are in parentheses.

Joint Name	Degree(s) of Freedom		
	1st	2nd	3rd
pelvis	lateral flexion (x)	forward flexion (y)	torsion (z)
waist	lateral flexion (x)	forward flexion (y)	torsion (z)
lower neck	lateral flexion (x)	forward flexion (y)	torsion (z)
upper neck	lateral flexion (x)	forward flexion (y)	torsion (z)
shoulders	abduction-adduction (x)	internal-external rotation (z)	flexion-extension (y)
elbows	flexion-extension (y)	n/a	n/a
hips	abduction-adduction (x)	medial-lateral rotation (z)	flexion-extension (y)
knees	flexion-extension (y)	n/a	n/a
ankles	inversion-eversion (x)	dorsi-plantar flexion (y)	medial-lateral rotation (z)

Orientation of a segment is effected by performing successive right-handed rotations of that segment relative to its parent segment - each rotation corresponds to a joint degree of freedom. These rotations are performed about the local segment axes and the sequence is given in Table K.1. For example, the left upper leg is connected to the lower torso by the left hip joint; the limb is first abducted relative to lower torso, it then undergoes lateral rotation, followed by extension. The remainder of the lower extremity (lower leg and foot) moves with the upper leg during this orientation process.

By default all joints are assigned stiffnesses, viscous characteristics, and stop angles which should give reasonable results in a crash simulation. One or all default values of a joint may be altered by applying the *\*COMPONENT\_GEBOD\_JOINT\_OPTION* command to the joint of interest. The default shape of the resistive torque load curve used by all joints is shown in Figure K.2. A scale factor is applied to this curve to obtain the proper stiffness relationship. Listed in Table K.2 are the default values of joint characteristics for dummies of all types and sizes. These values are given in the English system of units; the appropriate units are used if a different system is specified in card 1 of *\*COMPONENT\_GEBOD\_OPTION*.

Table K.2: Default joint characteristics for all dummies.

joint degrees of freedom	load curve scale factor (in-lbf)	damping coef. (in-lbf-s/rad)	low stop angle (degrees)	high stop angle (degrees)	neutral angle (degrees)
pelvis - 1	65000	5.77	-20	20	0
pelvis - 2	65000	5.77	-20	20	0
pelvis - 3	65000	5.77	-5	5	0
waist - 1	65000	5.77	-20	20	0
waist - 2	65000	5.77	-20	20	0
waist - 3	65000	5.77	-35	35	0
lower neck - 1	10000	5.77	-25	25	0
lower neck - 2	10000	5.77	-25	25	0
lower neck - 3	10000	5.77	-35	35	0
upper neck - 1	10000	5.77	-25	25	0
upper neck - 2	10000	5.77	-25	25	0
upper neck - 3	10000	5.77	-35	35	0
l. shoulder - 1	100	5.77	-30	175	0
r. shoulder - 1	100	5.77	-175	30	0
shoulder - 2	100	5.77	-65	65	0
shoulder - 3	100	5.77	-175	60	0
elbow - 1	100	5.77	1	-140	0
l. hip - 1	10000	5.77	-25	70	0
r. hip - 1	10000	5.77	-70	25	0
hip - 2	10000	5.77	-70	70	0
hip - 3	10000	5.77	-140	40	0
knee - 1	100	5.77	-1	120	0
l. ankle - 1	100	5.77	-30	20	0
l. ankle - 1	100	5.77	-20	30	0
ankle - 2	100	5.77	-20	45	0
ankle - 3	100	5.77	-30	30	0

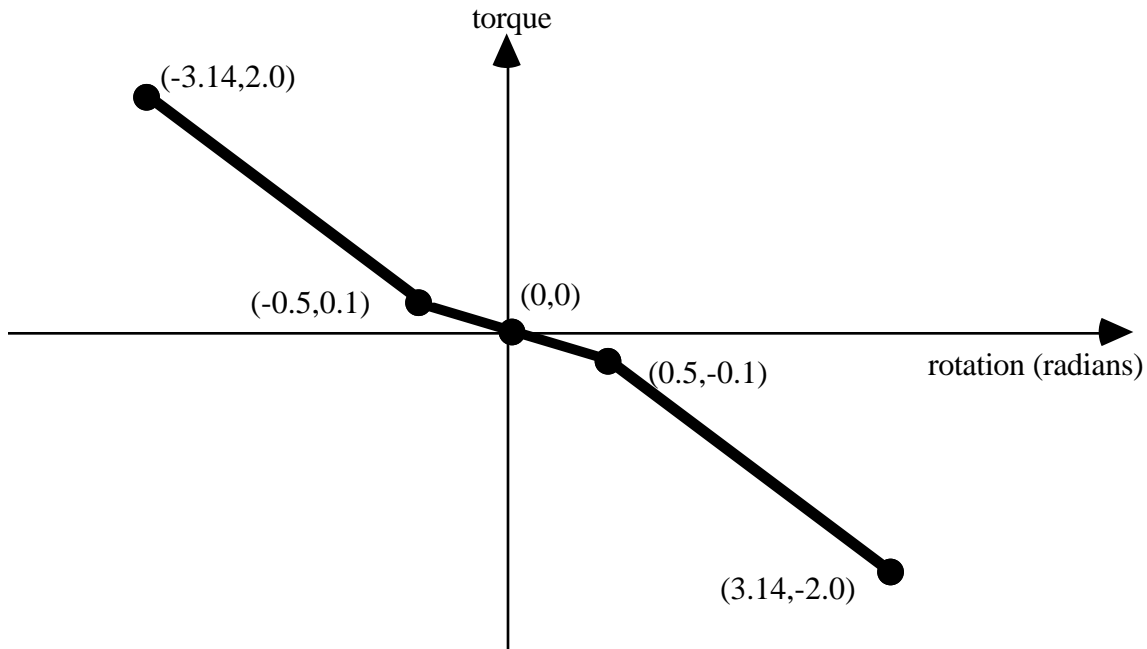


Figure K.2: Characteristic torque curve shape used by all joints.

The dummy depicted in Figure K.1 appears in what is referred to as its "nominal" position. In this position the dummy is standing upright facing in the positive x direction and the toe-to-head direction points in positive z. Additionally, the dummy's hands are at the sides with palms facing inward and the centroid of the lower torso is positioned at the origin of the global coordinate system. Each of the dummy's segments has a local coordinate system attached to it and in the nominal position all of the local axes are aligned with the global axes.

When performing a simulation involving a \*COMPONENT\_GEBOD dummy, a positioning file named "gebod.did" must reside in the directory with the LS-DYNA input file; here the extension *did* is the dummy ID number, see card 1 of \*COMPONENT\_GEBOD\_OPTION. The contents of a typical positioning file is shown in Table K.3; it consists of 40 lines formatted as (59a1,e30.0). All of the angular measures are input as degrees, while the lower torso global positions depend on the choice of units in card 1 of \*COMPONENT\_GEBOD\_OPTION. Setting all of the values in this file to zero yields the so-called "nominal" position.

Table K.3: Typical contents of a dummy positioning file.

lower torso	centroid global x position		0.0
lower torso	centroid global y position		0.0
lower torso	centroid global z position		0.0
total body	global x rotation		0.0
total body	global y rotation		-20.0
total body	global z rotation		180.0
pelvis	lateral flexion	+ = tilt right	0.0
pelvis	forward flexion	+ = lean fwd	0.0
pelvis	torsion	+ = twist left	0.0
waist	lateral flexion	+ = tilt right	0.0
waist	forward flexion	+ = lean fwd	0.0
waist	torsion	+ = twist left	0.0
lower neck	lateral flexion	+ = tilt right	0.0
lower neck	forward flexion	+ = nod fwd	0.0
lower neck	torsion	+ = twist left	0.0
upper neck	lateral flexion	+ = tilt right	0.0
upper neck	forward flexion	+ = nod fwd	0.0
upper neck	torsion	+ = twist left	0.0
left shoulder	abduction-adduction	+ = abduction	30.0
left shoulder	internal-external rotation	+ = external	-10.0
left shoulder	flexion-extension	- = fwd raise	-40.0
right shoulder	abduction-adduction	- = abduction	-30.0
right shoulder	internal-external rotation	- = external	10.0
right shoulder	flexion-extension	- = fwd raise	-40.0
left elbow	flexion-extension	+ = extension	-60.0
right elbow	flexion-extension	+ = extension	-60.0
left hip	abduction-adduction	+ = abduction	0.0
left hip	medial-lateral rotation	+ = lateral	0.0
left hip	flexion-extension	+ = extension	-80.0
right hip	abduction-adduction	- = abduction	0.0
right hip	medial-lateral rotation	- = lateral	0.0
right hip	flexion-extension	+ = extension	-80.0
left knee	flexion-extension	+ = flexion	50.0
right knee	flexion-extension	+ = flexion	50.0
left ankle	inversion-eversion	+ = eversion	0.0
left ankle	dorsi-plantar flexion	+ = plantar	0.0
left ankle	medial-lateral rotation	+ = lateral	0.0
right ankle	inversion-eversion	- = eversion	0.0
right ankle	dorsi-plantar flexion	+ = plantar	0.0
right ankle	medial-lateral rotation	- = lateral	0.0

## Appendix K

In Figure K.3 the 50th percentile male dummy is shown in a seated position and some of its joints are labeled. The file listed in Table K.3 was used to put the dummy into the position shown. Note that the dummy was first brought into general orientation by setting nonzero values for two of the lower torso local rotations. This is accomplished by performing right-handed rotations successively about local axes fixed in the lower torso, the sequence of which follows: the first about local x, next about local y, and the last about local z. The dummy in Figure K.3 was made to pitch backward by setting "total body global y rotation" equal to -20. Setting the "total body global z rotation" equal to 180 caused the dummy to rotate about the global z axis and face in the -x direction.

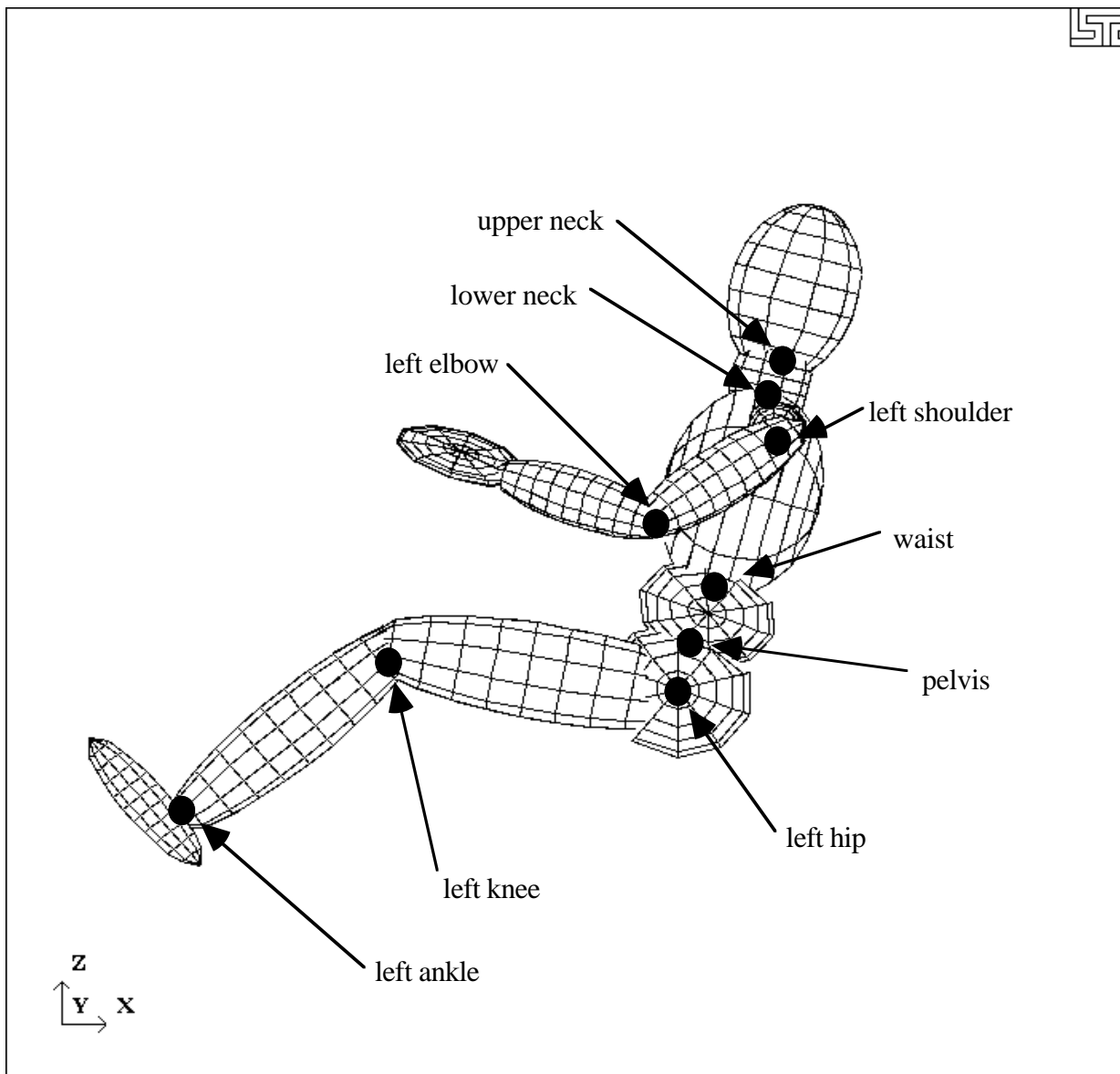


Figure K.3: Dummy seated using the file listed in Table K.3.

## REFERENCES

Cheng, H., Obergefell, L.A., and Rizer, A., March 1994, "Generator of Body (GEBOD) Manual,"  
Report No. AL/CF-TR-1994-0051.





---

**APPENDIX L: LS-DYNA MPP USER GUIDE**

---

**Introduction**

This is a short users guide for the MPP version of LS-DYNA. For a general guide to the capabilities of LS-DYNA and a detailed description of the input, consult the LS-DYNA user manual. If you have questions about this guide, find errors or omissions in it, or have specific features you would like to have supported, please contact **LSTC**.

**Supported Features**

First and foremost, the only input formats currently supported are 920, 930, and keyword. Models in any of the older formats will need to be converted to one of these input formats before they can be run with the current version of LS-DYNA for massively parallel processors, mpp.

The large majority of LS-DYNA options are available on MPP computers. Those that are not supported are being systematically added. Unless otherwise noted here, all the options of LS-DYNA version 93x are supported by LS-DYNA MPP

Here is the list of **unsupported** features:

- \*ALE
- \*BOUNDARY\_CONVECTION
- \*BOUNDARY\_FLUX
- \*BOUNDARY\_RADIATION
- \*BOUNDARY\_USA\_SURFACE
- \*CONSTRAINED\_RIGID\_BODY\_STOPPER
- \*CONSTRAINED\_SHELL\_TO\_SOLID
- \*CONTACT\_CONSTRAINT\_NODES\_TO\_SURFACE
- \*CONTACT\_CONSTRAINT\_SURFACE\_TO\_SURFACE
- \*CONTACT\_ERODING\_NODES\_TO\_SURFACE
- \*CONTACT\_ERODING\_SINGLE\_SURFACE
- \*CONTACT\_ERODING\_SURFACE\_TO\_SURFACE
- \*CONTACT\_TIEBREAK\_SURFACE\_TO\_SURFACE
- \*CONTACT\_1D
- \*DATABASE\_AVIS
- \*DATABASE\_MOVIE
- \*DATABASE\_MPGS
- \*DATABASE\_TRACER
- \*DATABASE\_BINARY\_XTFIELD
- \*DEFOEMABLE\_TO\_RIGID
- \*INTERFACE\_COMPONENT
- \*INTERFACE\_JOY
- \*LOAD\_SUPERELASTIC\_OPTION
- \*USER

## Appendix L

---

- \*PART\_REPOSITION
- \*RIGIDWALL\_PLANAR\_FORCES
- \*TERMINATION

LS-DYNA MPP can restart; however, the restart options are still quite limited. Only the termination time, plot interval, time step control, and restart dump frequency may be changed when restarting. The **supported** keywords are:

- \*CONTROL\_TERMINATION
- \*CONTROL\_TIMESTEP
- \*DATABASE\_BINARY

However, the full deck restart capability is supported, as is explained below in the section on Parallel Specific Options.

### Contact Interfaces

LS-DYNA MPP uses a completely redesigned, highly parallel contact algorithm. The contact options currently **supported** include:

- \*CONTACT\_AUTOMATIC\_NODES\_TO\_SURFACE
- \*CONTACT\_AUTOMATIC\_ONE\_WAY\_SURFACE\_TO\_SURFACE
- \*CONTACT\_AUTOMATIC\_SINGLE\_SURFACE
- \*CONTACT\_AUTOMATIC\_SURFACE\_TO\_SURFACE
- \*CONTACT\_NODES\_TO\_SURFACE
- \*CONTACT\_ONE\_WAY\_SURFACE\_TO\_SURFACE
- \*CONTACT\_SINGLE\_SURFACE
- \*CONTACT\_SURFACE\_TO\_SURFACE
- \*CONTACT\_TIED\_NODES\_TO\_SURFACE
- \*CONTACT\_TIED\_SURFACE\_TO\_SURFACE
- \*CONTACT\_AUTOMATIC\_GENERAL
- \*CONTACT\_SINGLE\_EDGE
- \*CONTACT\_SLIDING\_ONLY
- \*CONTACT\_SLIDING\_ONLY\_PENALTY
- \*CONTACT\_TIEBREAK\_NODES\_TO\_SURFACE

Because these options are all supported via the new, parallel contact algorithms, slight differences in results may be observed as compared to the serial and SMP versions of LS-DYNA. Work has been done to minimize these differences, but they may still be evident in some models.

### Pre-decomposition

There is an optional auxiliary serial program, **MPPPRE**, which creates a binary file containing decomposition information for the problem. If **MPPPRE** is not used, LS-DYNA MPP will do the decomposition itself at run time. The advantages of using **MPPPRE** are:

The parallel machine is not tied up while the decomposition is done.

The start up time of the parallel run is greatly reduced.

If the problem is to be run more than once (such as when benchmarking the program), the decomposition need only be done once.

The problem can later be run on any number of processors, which evenly divides the number of processors for which the decomposition was performed.

Here is a sample *pfile* (see the section below on Parallel Specific Options) which shows the options required for using **MPPPRE**:

```
decomposition {  
    numproc 20  
    file decompfile  
}
```

This will create a file *decompfile.pre* that contains the necessary decomposition information for 20 processors. When running LS-DYNA MPP, these options are also required to be in the *pfile* so that the program can find the decomposition information.

When it reads the decomposition file (which contains the information of the number of processors to be used), it will start the calculation if the number of processors currently running evenly divides the number of processors in the file, i.e. 2,4,5,10,etc. Otherwise, the execution will stop.

## Output Files and Post-Processing

For performance reasons, many of the ASCII output files normally created by LS-DYNA have been combined into a new binary format used by LS-DYNA MPP. There is a post-processing program *dumpbdb*, which reads this binary database of files and produces as output the corresponding ASCII files. The new binary files will be created in the directory specified as the global directory in the *pfile* (See section *pfile*). The files (one per processor) are named *dbout.nnnn*, where *nnnn* is replaced by the four-digit processor number. To convert these files to ASCII three steps are required, as follows:

```
cd <global directory>  
cat dbout.* > dbout  
dumpbdb dbout
```

The **supported** ASCII files are:

```
*DATABASE_SECFORC  
*DATABASE_RWFORC
```

## Appendix L

---

\*DATABASE\_NODOUT  
\*DATABASE\_ELOUT  
\*DATABASE\_DEFORC  
\*DATABASE\_MATSUM  
\*DATABASE\_NCFORC  
\*DATABASE\_RCFORC  
\*DATABASE\_NODOFRC  
\*DATABASE\_RBDOUT  
\*DATABASE\_SLEOUT  
\*DATABASE\_SBTOUT  
\*DATABASE\_ABSTAT  
\*DATABASE\_JNTFORC

Some of the normal LS-DYNA files will have corresponding collections of files produced by LS-DYNA MPP, with one per processor. These include the d3dump files (new names = d3dump.nnnn), the messag files (now mesnnnn) and others. Most of these will be found in the local directory specified in the pfile.

The format of the d3plot file has not been changed. It will be created in the global directory, and can be directly handled with your current graphics post-processor.

### Parallel Specific Options

There are a few new command line options that are specific to the MPP version of LS-DYNA.

In the serial and SMP versions of LS-DYNA, the amount of memory required to run the problem can be specified on the command line using the keyword *memory=XXX*, where *XXX* is the number of words of memory to be allocated. For the MPP code, this will result in each processor allocating *XXX* words of memory. If pre-decomposition has not been performed, one processor must perform the decomposition of the problem. This can require substantially more memory than will be required once execution has started. For this reason, there is a second memory command line option, *memory2=YYY*. If used together with the memory keyword, the decomposing processor will allocate *XXX* words of memory, and all other processors will allocate *YYY* words of memory.

For example, in order to run a 250,000 element crash problem on 4 processors, you might need *memory=80000000* and *memory2=20000000*. To run the same problem on 16 processors, you still need *memory=80000000*, but can set *memory2=6000000*. The value for *memory2* drops nearly linearly with the number of processors used to run the program, which works well for shared-memory systems.

The full deck restart capability is supported by the MPP version of LS-DYNA, but in a manner slightly different than the SMP code. Each time a restart dump file is written, a separate restart file is also written with the base name D3FULL. For example, when the third restart file d3dump03 is written (one for each processor, d3dump03.0000, d3dump03.0001, etc), there is also a single file written named d3full03. This file is required in order to do a full deck restart—the d3dump files are

not used in this case by the MPP code. In order to perform a full deck restart with the MPP code, you first must prepare a full deck restart input file as for the serial/SMP version. Then, instead of giving the command line option *r=d3dump03* you would use the special option *n=d3full03*. The presence of this command line option tells the MPP code that this is a restart, not a new problem, and that the file “d3full03” contains the geometry and stress data carried over from the previous run.

There is a new command line option: *p=pfile*. “pfile” contains MPP specific parameters that affect the execution of the program. The file is split into sections, with several options in each section. Currently, these sections: *directory*, *decomposition*, *contact*, and *general* are available. First, here is a sample pfile:

```

directory{
  global rundir
  local /tmp/rundir
}
contact {
  inititer 4
}

```

The file is case insensitive and free format input, with the exception that each word or bracket must be surrounded on both sides with either a space, tab, or newline character. The sections and options currently supported are:

## Directory Specific Options

### **directory**

#### **global path (Default = current working directory)**

Relative path to a directory accessible to all processors. This directory will be created if necessary.

#### **local path (Default = global path)**

Relative path to a processor specific local directory for scratch files. This directory will be created if necessary. This is of primary use on systems where each processor has a local disk attached to it.

## **Decomposition Options**

### **decomposition**

#### **file *filename* (Default = None)**

The name of the file that holds the decomposition information. This file will be created if the pre-decomposition program is being run. Otherwise, it is expected to exist in the current working directory. If the filename does not end with the extension *.pre* then this extension is added. If this option is not specified, LS-DYNA MPP will perform the decomposition.

#### **numproc *n***

For pre-decomposition only. (This option can be specified for the MPP code, but will be ignored. This allows the same pfile to be used by LS-DYNA MPP and MPPPRE.) The problem will be decomposed for *n* processors. The resulting decomposition file can later be used on any number of processors that evenly divides *n*.

#### **costinc *n* (Default = 0)**

The elements involved in contact are considered to be this much more computationally expensive during decomposition. If an average thin shell has a weight of about 30, setting *costinc* to 30 would indicate that each shell element involved in contact is about twice as computationally expensive as a normal shell element. The usage of this may or may not have favorable impact on the total runtime of the problem.

#### **method *name* (Default = *rcb*)**

Currently, there are three decomposition methods supported, namely *rcb*, *rsb* and *greedy*. Method *rcb* is Recursive Coordinate Bisection. Method *rsb* is Recursive Spectral Bisection. (This method is only available when using the pre-decomposition program.) Method *greedy* is a much faster but less sophisticated method. The impact on overall runtime is problem dependent, but generally not very large. Other methods are under consideration.

#### **expdir *n* (Default = 1)**

This only applies when using Recursive Coordinate Bisection where *n* equals to 1 specifies the X coordinate direction, 2 the Y and 3 the Z respectively. For a full explanation see the following item.

#### **expsf *t* (Default = 1)**

This only applies when using Recursive Coordinate Bisection. The model will be compressed by a factor of *t* in the coordinate direction indicated by the keyword **expdir** before RCB is performed. This in no way affects the geometry of the actual model, but it has the effect of expanding the decomposition domains in the indicated direction by a factor of 1/*t*.

Preliminary experience indicates that this can be used to provide greatly improved load balance for contact problems. For example, if **expdir** is set to the punch travel direction for a sheet metal stamping problem, and **expsf** is given as 0, each processor will be responsible

---

for a whole column of the problem. This results in the contact work being very equally distributed among the processors, and in some such problems can result in dramatic speed improvements over the other decomposition methods.

**show**

If this keyword appears in the decomposition section, the d3plot file is doctored so that the decomposition can be viewed with the post processor. Displaying material 1 will show that portion of the problem assigned to processor 0, and so on. The problem will not actually be run, but the code will terminate once the initial d3plot state has been written.

## **Contact Options**

### **contact**

Note: Any conflicting options that might be specified in the problem itself, in accordance with the 920 version manual, are ignored.

**bucket  $n$  (Default = 200)**

Specifies the frequency for bucket sort contact searches.

**ntrack  $n$  (Default = 3)**

Specifies the number of contact segments to keep track of per slave node. Increasing this number requires more storage, and will have some impact on speed. For sheet metal stamping problems, values of 1 or 2 are probably adequate, depending on the problem configuration and definition of contact interfaces.

**inititer  $n$  (Default = 4)**

During contact interface initialization, an attempt is made to move the slave nodes to eliminate initial penetrations. An iterative approach is used, since moving the nodes in one direction may cause problems in a different direction, particularly with the single sided contact options. This parameter specifies the maximum number of iterations to attempt.

After the final iteration, any nodes which still have significant penetrations are deleted from the contact interface. Each processor creates a message file in its local directory, which contains, among other things, a list of all nodes moved and those nodes deleted during this process. The file name is given by appending a four digit processor number to the string "mes" so that, for example, the message file from processor 3 is mes0003.

**brickoff  $t$  (Default = 5% of the characteristic length of the elements in the model)**

This specifies the thickness used for solid elements in the contact surface. Contact nodes that penetrate a solid element by more than half this amount are released. The average thickness of the shell elements in the problem is a good choice. The initial penetration tolerance is set as 5% of this value.

## **General Options**

### **general**

#### **nodump**

If this keyword appears, all restart dump file writing will be suppressed.

Here is a sample pfile with every possible option specified:

```
directory{  
  global rundir  
  local /tmp/rundir  
}
```

```
contact{  
  bucket 250  
  ntrack 4  
  inititer 6  
}
```

```
decomposition {  
  file dcfile64  
  numproc 64  
  costinc 10  
  method greedy  
  show  
}
```

```
general {  
  nodump  
}
```

### **Execution of LS-DYNA MPP**

LS-DYNA MPP is running under parallel environment, which is provided by the hardware vendors. The execution of the program therefore varies from machine to machine. On some platforms, command line parameters can be passed directly on the command line. However, platforms that do not allow command line parameters require the use of the names file. The names file is supported on all systems.

Here is a simple table to show how to run the program on various platforms. Scripts are often written to mask these differences.



<b>Platform</b>	<b>Execution Command</b>
DEC Alpha	<code>dmpirun -np <i>n</i> <i>mpp-dyna</i></code>
Fujitsu	<code>jobexec -vp <i>n</i> -mem <i>m</i> <i>mpp-dyna</i></code>
Hitachi	<code>mpirun -np <i>n</i> <i>mpp-dyna</i></code>
HP	<code><i>mpp-dyna</i> -np <i>n</i></code>
IBM	<code>#!/bin/ksh export MP_PROC=<i>n</i> export MP_LABELIO=no export MP_EUILIB=us export MPI_EUIDEVICE=cross0 poe <i>mpp-dyna</i></code>
NEC	<code>mpirun -np <i>n</i> <i>mpp-dyna</i></code>
SGI	<code>mpirun -np <i>n</i> <i>mpp-dyna</i></code>
Sun	<code>tmrn -np <i>n</i> <i>mpp-dyna</i></code>

Where *n* is number of processors, *mpp-dyna* is the name of the LS-DYNA MPP executable and *m* is the real memory in Mega Bytes (MB).



---

**APPENDIX M: Implicit Solver**

---

**INTRODUCTION**

The terms implicit and explicit refer to time integration algorithms. In the explicit approach, internal and external forces are summed at each node point, and a nodal acceleration is computed by dividing by nodal mass. The solution is advanced by integrating this acceleration in time. The maximum time step size is limited by the Courant condition, producing an algorithm which typically requires many relatively inexpensive time steps.

While explicit analysis is well suited to dynamic simulations such as impact and crash, it can become prohibitively expensive to conduct long duration or static analyses. Static problems such as sheet metal springback after forming are one application area for implicit methods.

In the implicit method, a global stiffness matrix is computed, inverted, and applied to the nodal out-of-balance force to obtain a displacement increment. The advantage of this approach is that time step size may be selected by the user. The disadvantage is the large numerical effort required to form, store, and factorize the stiffness matrix. Implicit simulations therefore typically involve a relatively small number of expensive time steps.

Version 950 includes the first release of implicit analysis capability using LS-DYNA. Initially targeted at metal forming springback simulation, this new capability allows static stress analysis without interfacing to LSTC's implicit software package LS-NIKE3D. Over time, increased implicit functionality will be added to LS-DYNA, and the need for LS-NIKE3D will be eliminated.

**ACTIVATING IMPLICIT MODE**

LS-DYNA's implicit mode may be activated in two ways. Using the new \*CONTROL\_IMPLICIT\_GENERAL keyword, a simulation may be flagged to run entirely in implicit mode. Alternatively, an explicit simulation may be seamlessly switched into implicit mode at a specific time using the \*INTERFACE\_SPRINGBACK\_SEAMLESS keyword. The seamless switching feature is intended to simplify metal forming springback calculations, where the forming phase can be run in explicit mode, followed immediately by an implicit static springback simulation. In case of difficulty, restart capability is supported.

# Appendix M

---

Seven new keywords are available to support implicit analysis. Default values are carefully selected to minimize input necessary for most simulations. These are summarized below:

**\*CONTROL\_IMPLICIT\_GENERAL**

Activates implicit mode, selects time step size.

**\*CONTROL\_IMPLICIT\_LINEAR**

Selects parameters for solving system of linear equations  $[K]\{x\}=\{f\}$ .

**\*CONTROL\_IMPLICIT\_NONLINEAR**

Selects linear or nonlinear solution method, convergence tolerances.

**\*CONTROL\_IMPLICIT\_AUTO**

Activates automatic time step control.

**\*CONTROL\_IMPLICIT\_DYNAMICS**

Activates and controls dynamic implicit solution using Newmark method.

**\*CONTROL\_IMPLICIT\_EIGENVALUE**

Activates and controls eigenvalue analysis.

**\*CONTROL\_IMPLICIT\_STABILIZATION**

Activates and controls artificial stabilization for multi-step springback.

## **FEATURES AVAILABLE FOR IMPLICIT MODE**

Implicit stiffness matrix calculations must be added internally to activate each individual LS-DYNA feature for implicit mode. The summary below shows features currently available for implicit analysis. Contact LSTC or your LS-DYNA distributor if you need a feature which is not shown, since this list is expanding rapidly.

LS-DYNA Features Currently Available for Implicit Analysis (as of April, 1999)

solid element formulations 1, 2

solid material models 1,2,3,5,6,7,12,13,18,20,24

beam element formulation 1

beam material models 1, 20

shell element formulations 2,6,10,16  
 shell element material models 1,2,3,18,20,24,36,37  
 2-D solid element formulations 12,13,15  
 2-D solid material models 1,2,3,5,6,7,12,13,18,20,24  
 discrete springs  
 contact interface types 3, m3, 5, m5, 10, m10, 13:  
     SURFACE\_TO\_SURFACE  
     NODES\_TO\_SURFACE  
     ONE\_WAY\_SURFACE\_TO\_SURFACE  
     FORMING\_SURFACE\_TO\_SURFACE  
     FORMING\_NODES\_TO\_SURFACE  
     FORMING\_ONE\_WAY\_SURFACE\_TO\_SURFACE  
     AUTOMATIC\_SINGLE\_SURFACE  
     2D\_AUTOMATIC\_SURFACE\_TO\_SURFACE  
 prescribed displacement and velocity boundary conditions (global directions)  
 most applied loads (force, pressure, etc.)

#### Implicit Analysis Features

nonlinear equation solvers: Linear, Full Newton, Quasi-Newton, Arc Length  
 sparse linear equations solvers: direct incore/out-of-core, CG and Lanczos iterative  
 automatic time step control  
 Newmark method for implicit dynamics  
 artificial stabilization for multi-step springback analysis  
 eigenvalue analysis  
 seamless switching for springback analysis

### SETTING UP AN IMPLICIT SIMULATION

A nonlinear implicit simulation is typically divided into several steps. In a dynamic simulation, these are *time steps*. In a static simulation, these are *load steps*. Multiple steps may be used to divide the nonlinear behavior into manageable pieces, to obtain results at intermediate stages during the simulation, or perhaps to resolve a particular frequency of motion in dynamic simulations. In each step, an equilibrium geometry is sought which balances internal and external forces in the model. The *nonlinear equation solver* performs an iterative search using one of several Newton based methods. *Convergence* of this iterative process is obtained when norms of displacement and/or energy fall below user-prescribed tolerances.

## Appendix M

---

Control parameters for the nonlinear equation solver are input using the \*CONTROL\_IMPLICIT\_NONLINEAR keyword. By default, the progress of the equilibrium search is not shown to the screen. This output can be activated either using the NLPRINT input parameter, or interactively toggled on and off by entering “<ctrl-c> nlprint”. The box below shows a typical iteration sequence, where the norms of displacement ( $du/u$ ) and energy ( $E_i/E_0$ ) are displayed. When these norms are reduced below user prescribed tolerances (default  $1.0e-3$  and  $1.0e-2$ , respectively), equilibrium is reached within sufficient accuracy, the iteration process is said to have *converged*, and the solution proceeds to the next time step.

```
BEGIN time step      3
=====
                time = 1.50000E-01
        current step size = 5.00000E-02

Iteration:   1      *|du|/|u| = 3.4483847E-01      *Ei/E0 = 1.0000000E+00

Iteration:   2      *|du|/|u| = 1.7706435E-01      *Ei/E0 = 2.9395439E-01

Iteration:   3      *|du|/|u| = 1.6631174E-03      *Ei/E0 = 3.7030904E-02

Iteration:   4      *|du|/|u| = 9.7088516E-05      *Ei/E0 = 9.6749731E-08
```

A typical print-out showing the progress of the nonlinear equation solver. By default, the progress of the equilibrium search is not shown to the screen. This output can be activated either using the NLPRINT input parameter, or interactively toggled on and off by entering:

“<ctrl-c> nlprint”.

### LINEAR EQUATION SOLVER

Within each equilibrium iteration, a linear system of equations of the form  $\mathbf{K}\Delta\mathbf{u} = \mathbf{R}$  must be solved. To do this, the stiffness matrix  $\mathbf{K}$  is inverted and applied to the out-of-balance load or residual  $\mathbf{R}$ , yielding a displacement increment  $\Delta\mathbf{u}$ . Storing and solving this linear system represents a large portion of the memory and CPU costs of an implicit analysis.

Control parameters for solving the linear system  $\mathbf{K}\Delta\mathbf{u} = \mathbf{R}$  are input using the keyword \*CONTROL\_IMPLICIT\_LINEAR. Several different linear equation solvers are available, including direct (Gaussian elimination) and iterative (conjugate gradient, Lanczos) methods. A sparse storage scheme is used to minimize memory requirements, which are still often substantial.

### NONLINEAR EQUATION SOLVER

Several different nonlinear equation solvers are available for finding equilibrium within each step. All are iterative in nature. In the *full Newton method*, a new stiffness matrix is formed and inverted each equilibrium iteration. This is the most costly method, but can require fewer iterations to reach equilibrium. In the *modified Newton method*, several iterations are performed using the same stiffness matrix. After each iteration, the geometry is updated using  $\Delta\mathbf{u}$  and a new  $\mathbf{R}$  is computed. This approach reduces cost by avoiding some forming and factoring of the stiffness matrix  $\mathbf{K}$ , but usually requires more iterations to reach equilibrium.

The default nonlinear equation solver is the BFGS solver, which uses a *quasi-Newton method*. In this method, the inverted stiffness matrix  $\mathbf{K}$  is used for several iterations, but is improved after each iteration using an inexpensive rank two update. If convergence is not reached after 10 iterations, or if *divergence* (increasing  $\mathbf{R}$ ) is detected, then a new stiffness matrix is automatically formed and inverted. This hybrid method combines the efficiency of the modified Newton method with the reliability of the full Newton method. The number of iterations between stiffness matrix reformations is a user input, defaulting to 10. If a value of one is chosen, then the full Newton method is recovered.

$$\mathbf{K}_{n+1}^{-1} = (\mathbf{I} + \mathbf{w}\mathbf{v}^T)\mathbf{K}_n^{-1}(\mathbf{I} + \mathbf{v}\mathbf{w}^T)$$

The BFGS update: A new stiffness matrix inverse is approximated by the old stiffness matrix inverse, and the outer product of two carefully chosen vectors.

# Appendix M

---

```
BEGIN time step      1
=====
              time = 1.00000E+00
      current step size = 1.00000E+00

Iteration:   1      *|du|/|u| = 2.5517753E+00      *Ei/E0 = 1.0000000E+00

DIVERGENCE (increasing residual norm) detected:
  |{Fe}-{Fi}| ( 7.5426269E+03) exceeds |{Fe}| ( 5.0000000E+00)
automatically REFORMING stiffness matrix...

Iteration:   2      *|du|/|u| = 6.0812935E-01      *Ei/E0 = 4.0526413E-01

DIVERGENCE (increasing residual norm) detected:
  |{Fe}-{Fi}| ( 4.3747811E+04) exceeds |{Fe}| ( 7.5426269E+03)
automatically REFORMING stiffness matrix...

Iteration:   4      *|du|/|u| = 1.0974191E-02      *Ei/E0 = 2.3907781E-04
Iteration:   5      *|du|/|u| = 1.0978787E-02      *Ei/E0 = 1.7910795E-04
Iteration:   6      *|du|/|u| = 4.2201181E-03      *Ei/E0 = 4.2557768E-05
Iteration:   7      *|du|/|u| = 4.1142219E-03      *Ei/E0 = 3.0658711E-05
Iteration:   8      *|du|/|u| = 1.9794757E-03      *Ei/E0 = 9.1215551E-06
Iteration:   9      *|du|/|u| = 1.7957653E-03      *Ei/E0 = 6.1669480E-06
Iteration:  10      *|du|/|u| = 1.2022830E-03      *Ei/E0 = 2.9031284E-06

ITERATION LIMIT reached, automatically REFORMING stiffness matrix...

Iteration:  11      *|du|/|u| = 5.4011414E-04      *Ei/E0 = 1.0553019E-06
```

A typical print-out showing the default BFGS nonlinear equation solver. Several automatic stiffness reformations are performed, initially due to divergence, and later when the default limit of 10 iterations is exceeded. By default, the progress of the equilibrium search is not shown to the screen. This output can be activated either using the NLPRINT input parameter, or interactively toggled on and off by entering "<ctrl-c> nlprint".

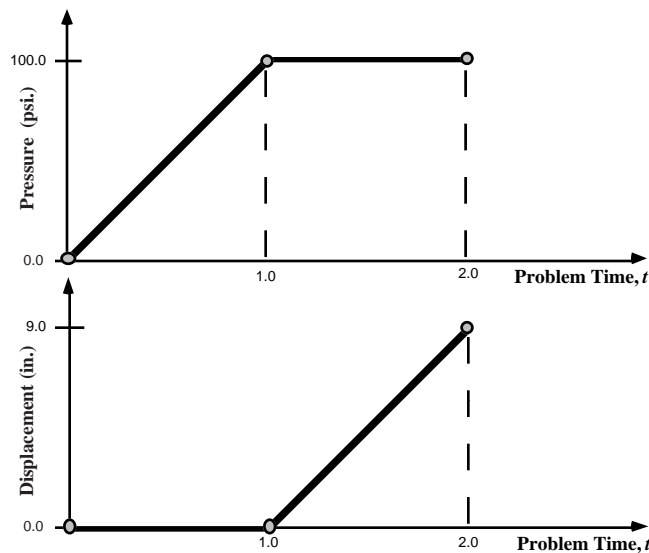


## APPLYING LOADS DURING IMPLICIT ANALYSIS

Loading is applied using the same keywords as in explicit analysis. Load curves are used to control the magnitude of each load as the simulation proceeds. Typically, the magnitude of each load begins at zero, and is increased to its full value at the end of the last step in the simulation. In this case, the load curve may be defined using only two points.

For example, consider a static analysis where a pressure of 100 psi. is to be applied in 4 steps. Since the analysis is static, the step size can be chosen arbitrarily. For convenience, choose a step size of 0.25, giving a termination time of 1.0. For this problem, the load curve has only two points: (0.0, 0.0) and (1.0, 100.0). LS-DYNA will automatically use linear interpolation to determine the load magnitude at each of the intermediate steps.

In a more complex example, consider a static problem with two types of loading. First, a static pressure of 100 psi. is to be applied, followed by a prescribed displacement of 9 inches. Two load curves are used for this problem, one to control the pressure, and one for the displacement, as shown below. Notice that the displacement is prescribed to be zero while the pressure is applied, then the pressure is held constant while the displacement is applied.



Load curves for a static simulation with two loading events. For convenience in this static simulation, the pressure loading is specified to begin at time 0.0 and end at time 1.0, and the displacement begins at time 1.0 and ends at time 2.0.

### AUTOMATIC TIME STEP SIZE CONTROL

In the most simple multi-step nonlinear implicit analysis, the user specifies the *termination time* using the \*CONTROL\_TERMINATION keyword, and the *time step size* using the \*CONTROL\_IMPLICIT\_GENERAL keyword, and each step is the same size. But for many simulations, the degree of nonlinearity varies through the course of the analysis. In this case the step size should ideally be varied such that solving for equilibrium in each step is equally difficult. This is accomplished by invoking automatic time step control, using the \*CONTROL\_IMPLICIT\_AUTO keyword.

There are two advantages to using automatic time step control. First, the time step size is automatically increased and/or decreased in response to the nonlinearity of the analysis. Nonlinearity is measured simply by the number of iterations required to reach equilibrium. An additional advantage is that if the equilibrium search fails during a time step, LS-DYNA does not terminate. Instead, the step is automatically repeated using a different step size. This process of backing up and retrying difficult steps lends much persistence to the analysis, and is often the only procedure for solving highly nonlinear problems short of adjusting the step size manually.

The input parameters for automatic time step control allow specification of the *optimum number of equilibrium iterations per step*. This indicates how hard LS-DYNA should work in each time step. If equilibrium is reached in fewer than optimum iterations, the size of the next step is increased, and likewise if the equilibrium search requires more than the optimum number of iterations, then the next step size is decreased. Minimum and maximum limits for step size are also input.