

LS-DYNA

KEYWORD USER'S MANUAL

April 2003
Version 970

Copyright © 1992-2003
LIVERMORE SOFTWARE
TECHNOLOGY CORPORATION
All Rights Reserved

Mailing Address:

**Livermore Software Technology Corporation
2876 Waverley Way
Livermore, California 94551**

Support Address:

**Livermore Software Technology Corporation
7374 Las Positas Road
Livermore, California 94551**

TEL: 925-449-2500

FAX: 925-449-2507

EMAIL: sales@lstc.com

**Copyright © 1992-2003 by Livermore Software Technology Corporation
All Rights Reserved**

TABLE OF CONTENTS

TABLE OF CONTENTS

INTRODUCTION	I.1
CHRONOLOGICAL HISTORY.....	I.1
DESCRIPTION OF KEYWORD INPUT.....	I.16
MATERIAL MODELS.....	I.25
SPATIAL DISCRETIZATION.....	I.26
CONTACT-IMPACT INTERFACES.....	I.29
INTERFACE DEFINITIONS FOR COMPONENT ANALYSIS.....	I.30
CAPACITY.....	I.30
SENSE SWITCH CONTROLS.....	I.30
PRECISION.....	I.31
EXECUTION SYNTAX.....	I.31
RESTART ANALYSIS.....	I.35
VDA/IGES DATABASES.....	I.36
MESH GENERATION.....	I.36
LS-PREPOST	I.37
EXECUTION SPEEDS	I.39
UNITS	I.40
GENERAL CARD FORMAT.....	I.40
*AIRBAG.....	1.1
*AIRBAG_OPTION1_{OPTION2}_{OPTION3}<OPTIONAL NUMERIC ID>	1.1
*AIRBAG_INTERACTION.....	1.42
*AIRBAG_REFERENCE_GEOMETRY_OPTION_OPTION.....	1.44
*ALE.....	2.1
*ALE_MULTI-MATERIAL_GROUP	2.2
*ALE_REFERENCE_SYSTEM_CURVE	2.6
*ALE_REFERENCE_SYSTEM_GROUP	2.8
*ALE_REFERENCE_SYSTEM_NODE.....	2.12
*ALE_REFERENCE_SYSTEM_SWITCH.....	2.14
*ALE_SMOOTHING.....	2.16
*ALE_TANK_TEST.....	2.17
*BOUNDARY.....	3.1
*BOUNDARY_ACOUSTIC_COUPLING.....	3.2

TABLE OF CONTENTS

*BOUNDARY_AMBIENT_EOS	3.3
*BOUNDARY_CONVECTION_OPTION	3.5
*BOUNDARY_CYCLIC.....	3.7
*BOUNDARY_ELEMENT_METHOD_OPTION	3.9
*BOUNDARY_FLUX_OPTION.....	3.20
*BOUNDARY_MCOL	3.23
*BOUNDARY_NON_REFLECTING.....	3.25
*BOUNDARY_NON_REFLECTING_2D	3.26
*BOUNDARY_OUTFLOW_CFD_OPTION.....	3.28
*BOUNDARY_PRESCRIBED_CFD_OPTION.....	3.30
*BOUNDARY_PRESCRIBED_MOTION_{OPTION1}_{OPTION2}	3.32
*BOUNDARY_PRESSURE_CFD_SET.....	3.37
*BOUNDARY_PRESSURE_OUTFLOW_OPTION.....	3.39
*BOUNDARY_RADIATION_OPTION	3.40
*BOUNDARY_SLIDING_PLANE.....	3.44
*BOUNDARY_SPC_{OPTION1}_{OPTION2}.....	3.45
*BOUNDARY_SPH_FLOW.....	3.47
*BOUNDARY_SPH_SYMMETRY_PLANE.....	3.49
*BOUNDARY_SYMMETRY_FAILURE	3.50
*BOUNDARY_TEMPERATURE_OPTION.....	3.51
*BOUNDARY_THERMAL_WELD.....	3.52
*BOUNDARY_USA_SURFACE	3.55
*COMPONENT.....	4.1
*COMPONENT_GEBOD_OPTION	4.2
*COMPONENT_GEBOD_JOINT_OPTION	4.4
*COMPONENT_HYBRIDIII	4.8
*COMPONENT_HYBRIDIII_JOINT_OPTION.....	4.10
*CONSTRAINED	5.1
*CONSTRAINED_ADAPTIVITY	5.2
*CONSTRAINED_EULER_IN_EULER.....	5.3
*CONSTRAINED_EXTRA_NODES_OPTION.....	5.5
*CONSTRAINED_GENERALIZED_WELD_OPTION_{OPTION}	5.7
*CONSTRAINED_GLOBAL.....	5.20
*CONSTRAINED_INTERPOLATION_{OPTION}.....	5.22
*CONSTRAINED_JOINT_OPTION_{OPTION}_{OPTION}_{OPTION}	5.27

TABLE OF CONTENTS

*CONSTRAINED_JOINT_STIFFNESS_OPTION	5.38
*CONSTRAINED_LAGRANGE_IN_SOLID	5.50
*CONSTRAINED_LINEAR_GLOBAL	5.56
*CONSTRAINED_LINEAR_LOCAL	5.59
*CONSTRAINED_NODAL_RIGID_BODY_{OPTION}_{OPTION}	5.61
*CONSTRAINED_NODE_SET_{OPTION}	5.68
*CONSTRAINED_POINTS	5.71
*CONSTRAINED_RIGID_BODIES	5.73
*CONSTRAINED_RIGID_BODY_STOPPERS	5.75
*CONSTRAINED_RIVET_{OPTION}	5.78
*CONSTRAINED_SHELL_TO_SOLID	5.80
*CONSTRAINED_SPOTWELD_{OPTION}_{OPTION}	5.82
*CONSTRAINED_TIE-BREAK	5.86
*CONSTRAINED_TIED_NODES_FAILURE	5.87
*CONTACT	6.1
*CONTACT_OPTION1_{OPTION2}_{OPTION3}_{OPTION4}	6.2
*CONTACT_AUTO_MOVE	6.41
*CONTACT_COUPLING	6.42
*CONTACT_ENTITY	6.44
*CONTACT_GEBOD_OPTION	6.53
*CONTACT_INTERIOR	6.56
*CONTACT_RIGID_SURFACE	6.58
*CONTACT_1D	6.61
*CONTACT_2D_OPTION1_{OPTION2}_{OPTION3}	6.62
*CONTROL	7.1
*CONTROL_ACCURACY	7.4
*CONTROL_ADAPSTEP	7.6
*CONTROL_ADAPTIVE	7.7
*CONTROL_ALE	7.12
*CONTROL_BULK_VISCOSITY	7.14
*CONTROL_CFD_AUTO	7.15
*CONTROL_CFD_GENERAL	7.17
*CONTROL_CFD_MOMENTUM	7.19
*CONTROL_CFD_PRESSURE	7.22
*CONTROL_CFD_TRANSPORT	7.24

TABLE OF CONTENTS

*CONTROL_CFD_TURBULENCE.....	7.28
*CONTROL_CHECK_{ <i>OPTION</i> }.....	7.29
*CONTROL_COARSEN.....	7.31
*CONTROL_CONTACT.....	7.33
*CONTROL_COUPLING.....	7.39
*CONTROL_CPU.....	7.41
*CONTROL_DYNAMIC_RELAXATION.....	7.42
*CONTROL_EFG.....	7.44
*CONTROL_ENERGY.....	7.45
*CONTROL_EXPLOSIVE_SHADOW.....	7.46
*CONTROL_HOURLASS_{ <i>OPTION</i> }.....	7.47
*CONTROL_IMPLICIT_AUTO.....	7.49
*CONTROL_IMPLICIT_BUCKLE.....	7.51
*CONTROL_IMPLICIT_DYNAMICS.....	7.52
*CONTROL_IMPLICIT_EIGENVALUE.....	7.54
*CONTROL_IMPLICIT_GENERAL.....	7.56
*CONTROL_IMPLICIT_MODES.....	7.58
*CONTROL_IMPLICIT_SOLUTION.....	7.60
*CONTROL_IMPLICIT_SOLVER.....	7.64
*CONTROL_IMPLICIT_STABILIZATION.....	7.67
*CONTROL_MPP_DECOMPOSITION_AUTOMATIC.....	7.68
*CONTROL_MPP_DECOMPOSITION_CHECK_SPEED.....	7.69
*CONTROL_MPP_DECOMPOSITION_CONTACT_DISTRIBUTE.....	7.70
*CONTROL_MPP_DECOMPOSITION_CONTACT_ISOLATE.....	7.71
*CONTROL_MPP_DECOMPOSITION_FILE.....	7.72
*CONTROL_MPP_DECOMPOSITION_METHOD.....	7.73
*CONTROL_MPP_DECOMPOSITION_NUMPROC.....	7.74
*CONTROL_MPP_DECOMPOSITION_RCBLOG.....	7.75
*CONTROL_MPP_DECOMPOSITION_SHOW.....	7.76
*CONTROL_MPP_DECOMPOSITION_TRANSFORMATION.....	7.77
*CONTROL_MPP_IO_NOD3DUMP.....	7.79
*CONTROL_MPP_IO_NODUMP.....	7.80
*CONTROL_MPP_IO_NOFILL.....	7.81
*CONTROL_MPP_IO_SWAPBYTES.....	7.82
*CONTROL_NONLOCAL.....	7.83

TABLE OF CONTENTS

*CONTROL_OUTPUT	7.84
*CONTROL_PARALLEL	7.86
*CONTROL_REMESHING	7.88
*CONTROL_RIGID	7.89
*CONTROL_SHELL	7.91
*CONTROL_SOLID	7.96
*CONTROL_SOLUTION	7.98
*CONTROL_SPH	7.99
*CONTROL_STRUCTURED_{ <i>OPTION</i> }	7.101
*CONTROL_SUBCYCLE	7.102
*CONTROL_TERMINATION	7.103
*CONTROL_THERMAL_NONLINEAR	7.104
*CONTROL_THERMAL_SOLVER	7.105
*CONTROL_THERMAL_TIMESTEP	7.107
*CONTROL_TIMESTEP	7.108
*DAMPING	8.1
*DAMPING_FREQUENCY_RANGE	8.1
*DAMPING_GLOBAL	8.3
*DAMPING_PART_MASS	8.5
*DAMPING_PART_STIFFNESS	8.7
*DAMPING_RELATIVE	8.9
*DATABASE	9.1
*DATABASE_ <i>OPTION</i>	9.2
*DATABASE_ADAMS	9.7
*DATABASE_BINARY_ <i>OPTION</i>	9.8
*DATABASE_CROSS_SECTION_ <i>OPTION1</i> _{ <i>OPTION2</i> }	9.12
*DATABASE_EXTENT_ <i>OPTION</i>	9.17
*DATABASE_FORMAT	9.25
*DATABASE_FSI	9.26
*DATABASE_HISTORY_ <i>OPTION</i>	9.28
*DATABASE_NODAL_FORCE_GROUP	9.31
*DATABASE_SPRING_FORWARD	9.32
*DATABASE_SUPERPLASTIC_FORMING	9.33
*DATABASE_TRACER	9.34

TABLE OF CONTENTS

*DEFINE	10.1
*DEFINE_BOX.....	10.2
*DEFINE_BOX_ADAPTIVE.....	10.3
*DEFINE_BOX_COARSEN.....	10.5
*DEFINE_BOX_DRAWBEAD.....	10.6
*DEFINE_BOX_SPH.....	10.7
*DEFINE_COORDINATE_NODES.....	10.9
*DEFINE_COORDINATE_SYSTEM.....	10.10
*DEFINE_COORDINATE_VECTOR.....	10.12
*DEFINE_CURVE.....	10.13
*DEFINE_CURVE_FEEDBACK.....	10.15
*DEFINE_CURVE_SMOOTH.....	10.18
*DEFINE_CURVE_TRIM_{OPTION}.....	10.20
*DEFINE_SD_ORIENTATION.....	10.25
*DEFINE_TABLE.....	10.27
*DEFINE_TRANSFORMATION.....	10.29
*DEFINE_VECTOR.....	10.33
*DEFORMABLE_TO_RIGID	11.1
*DEFORMABLE_TO_RIGID.....	11.2
*DEFORMABLE_TO_RIGID_AUTOMATIC.....	11.3
*DEFORMABLE_TO_RIGID_INERTIA.....	11.7
*ELEMENT	12.1
*ELEMENT_BEAM_{OPTION}_{OPTION}.....	12.2
*ELEMENT_DIRECT_MATRIX_INPUT.....	12.10
*ELEMENT_DISCRETE.....	12.12
*ELEMENT_INERTIA_{OPTION}.....	12.13
*ELEMENT_MASS.....	12.16
*ELEMENT_PLOTEL.....	12.17
*ELEMENT_SEATBELT.....	12.18
*ELEMENT_SEATBELT_ACCELEROMETER.....	12.19
*ELEMENT_SEATBELT_PRETENSIONER.....	12.21
*ELEMENT_SEATBELT_RETRACTOR.....	12.25
*ELEMENT_SEATBELT_SENSOR.....	12.31
*ELEMENT_SEATBELT_SLIPRING.....	12.35
*ELEMENT_SHELL_{OPTION}.....	12.37

TABLE OF CONTENTS

*ELEMENT_SOLID_{ <i>OPTION</i> }	12.41
*ELEMENT_SPH	12.47
*ELEMENT_TRIM	12.48
*ELEMENT_TSHELL	12.49
*EOS	13.1
*EOS_LINEAR_POLYNOMIAL	13.4
*EOS_JWL	13.6
*EOS_SACK_TUESDAY	13.7
*EOS_GRUNEISEN	13.8
*EOS_RATIO_OF_POLYNOMIALS	13.10
*EOS_LINEAR_POLYNOMIAL_WITH_ENERGY_LEAK	13.14
*EOS_IGNITION_AND_GROWTH_OF_REACTION_IN_HE	13.16
*EOS_TABULATED_COMPACTION	13.20
*EOS_TABULATED	13.22
*EOS_PROPELLANT_DEFLAGRATION	13.24
*EOS_TENSOR_PORE_COLLAPSE	13.29
*EOS_IDEAL_GAS	13.32
*EOS_JWL_B	13.34
*HOURGLASS	14.1
*HOURGLASS	14.1
*INCLUDE	15.1
*INCLUDE_{ <i>OPTION</i> }	15.1
*INITIAL	16.1
*INITIAL_CFD	16.2
*INITIAL_DETONATION	16.4
*INITIAL_FOAM_REFERENCE_GEOMETRY	16.7
*INITIAL_GAS_MIXTURE	16.8
*INITIAL_MOMENTUM	16.9
*INITIAL_STRAIN_SHELL	16.10
*INITIAL_STRESS_BEAM	16.11
*INITIAL_STRESS_SHELL	16.13
*INITIAL_STRESS_SOLID	16.15
*INITIAL_TEMPERATURE_ <i>OPTION</i>	16.17
*INITIAL_VEHICLE_KINEMATICS	16.18

TABLE OF CONTENTS

*INITIAL_VELOCITY	16.21
*INITIAL_VELOCITY_NODE.....	16.24
*INITIAL_VELOCITY_RIGID_BODY	16.25
*INITIAL_VELOCITY_GENERATION.....	16.26
*INITIAL_VOID_OPTION	16.28
*INITIAL_VOLUME_FRACTION	16.29
*INITIAL_VOLUME_FRACTION_GEOMETRY	16.30
*INTEGRATION	17.1
*INTEGRATION_BEAM.....	17.1
*INTEGRATION_SHELL.....	17.6
*INTERFACE.....	18.1
*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
*LOAD.....	19.1
*LOAD_BEAM_OPTION	19.2
*LOAD_BLAST.....	19.4
*LOAD_BODY_OPTION.....	19.6
*LOAD_BODY_GENERALIZED.....	19.10
*LOAD_BRODE	19.12
*LOAD_DENSITY_DEPTH.....	19.14
*LOAD_HEAT_GENERATION_OPTION	19.16
*LOAD_MASK.....	19.17
*LOAD_NODE_OPTION.....	19.19
*LOAD_RIGID_BODY.....	19.22
*LOAD_SEGMENT.....	19.24
*LOAD_SEGMENT_SET.....	19.26
*LOAD_SHELL_OPTION	19.28
*LOAD_SSA.....	19.30
*LOAD_SUPERPLASTIC_FORMING.....	19.33
*LOAD_THERMAL_OPTION.....	19.35

TABLE OF CONTENTS

*MAT	20.1
*MAT_ADD_EROSION.....	20.15
*MAT_NONLOCAL.....	20.17
*MAT_ELASTIC_{ <i>OPTION</i> }.....	20.21
*MAT_OPTIONTROPIC_ELASTIC	20.24
*MAT_PLASTIC_KINEMATIC.....	20.30
*MAT_ELASTIC_PLASTIC_THERMAL.....	20.33
*MAT_SOIL_AND_FOAM	20.36
*MAT_VISCOELASTIC.....	20.40
*MAT_BLATZ-KO_RUBBER.....	20.41
*MAT_HIGH_EXPLOSIVE_BURN	20.42
*MAT_NULL.....	20.45
*MAT_ELASTIC_PLASTIC_HYDRO_{ <i>OPTION</i> }.....	20.47
*MAT_STEINBERG.....	20.51
*MAT_STEINBERG_LUND.....	20.55
*MAT_ISOTROPIC_ELASTIC_PLASTIC.....	20.58
*MAT_ISOTROPIC_ELASTIC_FAILURE	20.59
*MAT_SOIL_AND_FOAM_FAILURE.....	20.61
*MAT_JOHNSON_COOK.....	20.62
*MAT_PSEUDO_TENSOR.....	20.66
*MAT_ORIENTED_CRACK.....	20.74
*MAT_POWER_LAW_PLASTICITY	20.76
*MAT_STRAIN_RATE_DEPENDENT_PLASTICITY.....	20.78
*MAT_RIGID.....	20.81
*MAT_ORTHOTROPIC_THERMAL.....	20.85
*MAT_COMPOSITE_DAMAGE.....	20.89
*MAT_TEMPERATURE_DEPENDENT_ORTHOTROPIC	20.93
*MAT_PIECEWISE_LINEAR_PLASTICITY	20.98
*MAT_GEOLOGIC_CAP_MODEL	20.102
*MAT_HONEYCOMB.....	20.108
*MAT_MOONEY-RIVLIN_RUBBER.....	20.115
*MAT_RESULTANT_PLASTICITY.....	20.118
*MAT_FORCE_LIMITED.....	20.119
*MAT_SHAPE_MEMORY	20.125
*MAT_FRAZER_NASH_RUBBER_MODEL.....	20.129
*MAT_LAMINATED_GLASS.....	20.132

TABLE OF CONTENTS

*MAT_BARLAT_ANISOTROPIC_PLASTICITY.....	20.134
*MAT_BARLAT_YLD96.....	20.137
*MAT_FABRIC.....	20.141
*MAT_PLASTIC_GREEN-NAGHDI_RATE.....	20.147
*MAT_3-PARAMETER_BARLAT.....	20.148
*MAT_TRANSVERSELY_ANISOTROPIC_ELASTIC_PLASTIC.....	20.152
*MAT_BLATZ-KO_FOAM.....	20.155
*MAT_FLD_TRANSVERSELY_ANISOTROPIC.....	20.156
*MAT_NONLINEAR_ORTHOTROPIC.....	20.158
*MAT_USER_DEFINED_MATERIAL_MODELS.....	20.162
*MAT_BAMMAN.....	20.165
*MAT_BAMMAN_DAMAGE.....	20.170
*MAT_CLOSED_CELL_FOAM.....	20.173
*MAT_ENHANCED_COMPOSITE_DAMAGE.....	20.175
*MAT_LOW_DENSITY_FOAM.....	20.181
*MAT_LAMINATED_COMPOSITE_FABRIC.....	20.185
*MAT_COMPOSITE_FAILURE_OPTION_MODEL.....	20.191
*MAT_ELASTIC_WITH_VISCOSITY.....	20.195
*MAT_KELVIN-MAXWELL_VISCOELASTIC.....	20.198
*MAT_VISCOUS_FOAM.....	20.200
*MAT_CRUSHABLE_FOAM.....	20.202
*MAT_RATE_SENSITIVE_POWERLAW_PLASTICITY.....	20.204
*MAT_MODIFIED_ZERILLI_ARMSTRONG.....	20.206
*MAT_LINEAR_ELASTIC_DISCRETE_BEAM.....	20.209
*MAT_NONLINEAR_ELASTIC_DISCRETE_BEAM.....	20.211
*MAT_NONLINEAR_PLASTIC_DISCRETE_BEAM.....	20.214
*MAT_SID_DAMPER_DISCRETE_BEAM.....	20.219
*MAT_HYDRAULIC_GAS_DAMPER_DISCRETE_BEAM.....	20.223
*MAT_CABLE_DISCRETE_BEAM.....	20.225
*MAT_CONCRETE_DAMAGE.....	20.227
*MAT_LOW_DENSITY_VISCOUS_FOAM.....	20.231
*MAT_ELASTIC_SPRING_DISCRETE_BEAM.....	20.235
*MAT_BILKHU/DUBOIS_FOAM.....	20.237
*MAT_GENERAL_VISCOELASTIC.....	20.239
*MAT_HYPERELASTIC_RUBBER.....	20.242
*MAT_OGDEN_RUBBER.....	20.246

TABLE OF CONTENTS

*MAT_SOIL_CONCRETE.....	20.250
*MAT_HYSTERETIC_SOIL	20.254
*MAT_RAMBERG-OSGOOD.....	20.257
*MAT_PLASTICITY_WITH_DAMAGE_{ <i>OPTION</i> }	20.259
*MAT_FU_CHANG_FOAM.....	20.264
*MAT_WINFRITH_CONCRETE	20.270
*MAT_WINFRITH_CONCRETE_REINFORCEMENT.....	20.274
*MAT_ORTHOTROPIC_VISCOELASTIC.....	20.276
*MAT_CELLULAR_RUBBER.....	20.279
*MAT_MTS.....	20.284
*MAT_PLASTICITY_POLYMER.....	20.289
*MAT_ACOUSTIC.....	20.291
*MAT_SOFT_TISSUE_{ <i>OPTION</i> }.....	20.293
*MAT_ELASTIC_6DOF_SPRING_DISCRETE_BEAM.....	20.297
*MAT_INELASTIC_SPRING_DISCRETE_BEAM	20.298
*MAT_INELASTIC_6DOF_SPRING_DISCRETE_BEAM	20.301
*MAT_BRITTLE_DAMAGE	20.302
*MAT_GENERAL_JOINT_DISCRETE_BEAM.....	20.305
*MAT_SIMPLIFIED_JOHNSON_COOK.....	20.307
*MAT_SIMPLIFIED_JOHNSON_COOK_ORTHOTROPIC_DAMAGE.....	20.310
*MAT_SPOTWELD_{ <i>OPTION</i> }.....	20.312
*MAT_GEPLASTIC_SRATE_2000a.....	20.321
*MAT_INV_HYPERBOLIC_SIN.....	20.323
*MAT_ANISOTROPIC_VISCOPLASTIC	20.325
*MAT_ANISOTROPIC_PLASTIC	20.330
*MAT_DAMAGE_1	20.334
*MAT_DAMAGE_2.....	20.340
*MAT_ELASTIC_VISCOPLASTIC_THERMAL.....	20.344
*MAT_JOHNSON_HOLMQUIST_CERAMICS.....	20.347
*MAT_JOHNSON_HOLMQUIST_CONCRETE	20.350
*MAT_FINITE_ELASTIC_STRAIN_PLASTICITY.....	20.353
*MAT_LAYERED_LINEAR_PLASTICITY	20.356
*MAT_UNIFIED_CREEP	20.359
*MAT_COMPOSITE_LAYUP.....	20.360
*MAT_COMPOSITE_MATRIX.....	20.363
*MAT_COMPOSITE_DIRECT.....	20.366

TABLE OF CONTENTS

*MAT_GENERAL_NONLINEAR_6DOF_DISCRETE_BEAM	20.368
*MAT_GURSON	20.375
*MAT_GURSON_RCDC	20.379
*MAT_GENERAL_NONLINEAR_1DOF_DISCRETE_BEAM	20.385
*MAT_HILL_3R	20.387
*MAT_MODIFIED_PIECEWISE_LINEAR_PLASTICITY	20.390
*MAT_PLASTICITY_COMPRESSION_TENSION	20.393
*MAT_MODIFIED_HONEYCOMB	20.395
*MAT_ARRUDA_BOYCE_RUBBER	20.403
*MAT_HEART_TISSUE	20.406
*MAT_LUNG_TISSUE	20.409
*MAT_SPECIAL_ORTHOTROPIC	20.412
*MAT_MODIFIED_FORCE_LIMITED	20.415
*MAT_VACUUM	20.426
*MAT_RATE_SENSITIVE_POLYMER	20.427
*MAT_TRANSVERSELY_ANISOTROPIC_CRUSHABLE_FOAM	20.429
*MAT_WOOD_{OPTION}	20.432
*MAT_PITZER_CRUSHABLE_FOAM	20.438
*MAT_SHWER_MURRAY_CAP_MODEL	20.440
*MAT_1DOF_GENERALIZED_SPRING	20.446
*MAT_FHWA_SOIL	20.447
*MAT_FHWA_SOIL_NEBRASKA	20.450
*MAT_GAS_MIXTURE	20.451
*MAT_CFD_{OPTION}	20.452
*MAT_DESHPANDE_FLECK_FOAM	20.454
*MAT_COMPOSITE_{OPTION}_MSC	20.456
*MAT_MODIFIED_CRUSHABLE_FOAM	20.467
*MAT_QUASILINEAR_VISCOELASTIC	20.469
*MAT_HILL_FOAM	20.472
*MAT_VISCOELASTIC_HILL_FOAM	20.475
*MAT_LOW_DENSITY_SYNTHETIC_FOAM_{OPTION}	20.479
*MAT_SIMPLIFIED_RUBBER	20.482
*MAT_SEISMIC_BEAM	20.484
*MAT_SOIL_BRICK	20.487
*MAT_DRUCKER_PRAGER	20.489
*MAT_RC_SHEAR_WALL	20.491

TABLE OF CONTENTS

*MAT_CONCRETE_BEAM.....	20.497
*MAT_GENERAL_SPRING_DISCRETE_BEAM.....	20.500
*MAT_SEISMIC_ISOLATOR.....	20.503
*MAT_JOINTED_ROCK.....	20.507
*MAT_SPRING_ELASTIC.....	20.511
*MAT_DAMPER_VISCOUS.....	20.512
*MAT_SPRING_ELASTOPLASTIC.....	20.513
*MAT_SPRING_NONLINEAR_ELASTIC.....	20.514
*MAT_DAMPER_NONLINEAR_VISCOUS.....	20.515
*MAT_SPRING_GENERAL_NONLINEAR.....	20.516
*MAT_SPRING_MAXWELL.....	20.518
*MAT_SPRING_INELASTIC.....	20.519
*MAT_SPRING_TRILINEAR_DEGRADING.....	20.520
*MAT_SPRING_SQUAT_SHEARWALL.....	20.521
*MAT_SPRING_MUSCLE.....	20.522
*MAT_SEATBELT.....	20.527
*MAT_THERMAL_OPTION.....	20.529
*NODE.....	21.1
*NODE.....	21.2
*NODE_RIGID_SURFACE.....	21.4
*NODE_SCALER.....	21.5
*PARAMETER.....	22.1
*PART.....	23.1
*PART_{OPTION1}_{OPTION2}_{OPTION3}_{OPTION4}.....	23.2
*PART_ADAPTIVE_FAILURE.....	23.10
*PART_MODES.....	23.11
*PART_SENSOR.....	23.14
*PART_MOVE.....	23.15
*RAIL.....	24.1
*RAIL_TRACK.....	24.2
*RAIL_TRAIN.....	24.7
*RIGIDWALL.....	25.1
*RIGIDWALL_GEOMETRIC_OPTION_{OPTION}_{OPTION}.....	25.2
*RIGIDWALL_PLANAR_{OPTION}_{OPTION}_{OPTION}.....	25.12

TABLE OF CONTENTS

*SECTION	26.1
*SECTION_BEAM.....	26.2
*SECTION_DISCRETE	26.9
*SECTION_POINT_SOURCE.....	26.12
*SECTION_POINT_SOURCE_MIXTURE.....	26.14
*SECTION_SEATBELT.....	26.19
*SECTION_SHELL_{ <i>OPTION</i> }.....	26.20
*SECTION_SOLID_{ <i>OPTION</i> }	26.29
*SECTION_SPH.....	26.34
*SECTION_TSHELL.....	26.36
*SET	27.1
*SET_BEAM_{ <i>OPTION</i> }.....	27.2
*SET_DISCRETE_{ <i>OPTION</i> }.....	27.5
*SET_MULTI-MATERIAL_GROUP_LIST	27.8
*SET_NODE_{ <i>OPTION</i> }.....	27.9
*SET_PART_{ <i>OPTION</i> }	27.13
*SET_SEGMENT_{ <i>OPTION</i> }	27.16
*SET_SHELL_{ <i>OPTION</i> }.....	27.20
*SET_SOLID_{ <i>OPTION</i> }	27.24
*SET_TSHELL_{ <i>OPTION</i> }.....	27.27
*TERMINATION	28.1
*TERMINATION_BODY.....	28.2
*TERMINATION_CONTACT.....	28.3
*TERMINATION_CURVE	28.4
*TERMINATION_NODE.....	28.5
*TITLE	29.1
*TITLE.....	29.1
*TRANSLATE	30.1
*TRANSLATE_ANSYS_ <i>OPTION</i>	30.1
*TRANSLATE_IDEAS_{ <i>OPTION</i> }.....	30.3
*TRANSLATE_NASTRAN	30.5
*USER	31.1
*USER_INTERFACE_ <i>OPTION</i>	31.1
*USER_LOADING.....	31.3

TABLE OF CONTENTS

RESTART INPUT DATA	32.1
*CHANGE_OPTION	32.3
*CONTROL_DYNAMIC_RELAXATION.....	32.15
*CONTROL_SHELL.....	32.17
*CONTROL_TERMINATION.....	32.19
*CONTROL_TIMESTEP	32.20
*DAMPING_GLOBAL.....	32.21
*DATABASE_OPTION.....	32.22
*DATABASE_BINARY_OPTION.....	32.24
*DELETE_OPTION.....	32.25
*INTERFACE_SPRINGBACK.....	32.27
*RIGID_DEFORMABLE_OPTION.....	32.29
*STRESS_INITIALIZATION_{OPTION}.....	32.32
*TERMINATION_OPTION	32.35
*TITLE.....	32.37
REFERENCES	REF.1
APPENDIX A	
USER DEFINED MATERIALS.....	A.1
APPENDIX B	
USER DEFINED AIRBAG SENSOR	B.1
APPENDIX C	
USER DEFINED SOLUTION CONTROL.....	C.1
APPENDIX D	
USER DEFINED INTERFACE CONTROL.....	D.1
APPENDIX E	
USER DEFINED INTERFACE FRICTION.....	E.1
APPENDIX F	
OCCUPANT SIMULATION INCLUDING COUPLING TO CAL3D AND MADYMO.....	F.1
INTRODUCTION.....	F.1
THE LS-DYNA/OCCUPANT SIMULATION PROGRAM LINK.....	F.1
DUMMY MODELING.....	F.3
AIRBAG MODELING.....	F.3
KNEE BOLSTER.....	F.4

TABLE OF CONTENTS

COMMON ERRORS.....	F.5
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
INTRODUCTION.....	M.1
SETTING UP AN IMPLICIT SOLVER.....	M.1
LINEAR EQUATION SOLVER.....	M.2
NONLINEAR EQUATION SOLVER.....	M.3
ELEMENT FORMULATIONS FOR IMPLICIT ANALYSIS.....	M.4
APPLYING LOADS DURING IMPLICIT ANALYSIS.....	M.4
AUTOMATIC TIME STEP SIZE CONTROL.....	M.5
IMPLICIT STRESS INITIALIZATION.....	M.5
TROUBLESHOOTING CONVERGENCE PROBLEMS.....	M.6
APPENDIX N	
UERS-WRITTEN WELD FAILURE.....	N.1

LS-DYNA USER'S MANUAL

INTRODUCTION

CHRONOLOGICAL HISTORY

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

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

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

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

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

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

INTRODUCTION

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

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

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

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

Capabilities added in 1989-1990:

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

- stonewall energy dissipation,
- airbags (1990),
- nodal rigid bodies,
- automatic sorting of triangular shells into C^0 groups,
- mass scaling for quasi static analyses,
- user defined subroutines,
- warpage checks on shell elements,
- thickness consideration in all contact types,
- automatic orientation of contact segments,
- sliding interface energy dissipation calculations,
- nodal force and energy database for applied boundary conditions,
- defined stonewall velocity with input energy calculations,

Capabilities added in 1991-1992:

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

Capabilities added in 1993-1994:

- Arbitrary Lagrangian Eulerian brick elements,
- Belytschko-Wong-Chiang quadrilateral shell element,
- Warping stiffness in the Belytschko-Tsay shell element,
- Fast Hughes-Liu shell element,
- Fully integrated thick shell element,
- Discrete 3D beam element,
- Generalized dampers,
- Cable modeling,
- Airbag reference geometry,

INTRODUCTION

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

Capabilities added in 1995:

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

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

Capabilities added in 1996-1997 in Version 940:

- Part/Material ID's may be specified with 8 digits.
- Rigid body motion can be prescribed in a local system fixed to the rigid body.
- Nonlinear least squares fit available for the Ogden rubber model.
- 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.
- 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

INTRODUCTION

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

Capabilities added during 1997-1998 in Version 950:

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

- *MAT_STRAIN_RATE_DEPENDENT_PLASTICITY
- *MAT_PIECEWISE_LINEAR_PLASTICITY
- *MAT_RATE_SENSITIVE_POWERLAW_PLASTICITY
- *MAT_ZERILLI-ARMSTRONG
- *MAT_PLASTICITY_WITH_DAMAGE
- *MAT_PLASTICITY_COMPRESSION_TENSION
- Material model, *MAT_PLASTICITY_WITH_DAMAGE, has a piecewise linear damage curve given by a load curve ID.
- The Arruda-Boyce hyper-viscoelastic rubber model is available, see *MAT_ARRUDA_BOYCE.
- Transverse-anisotropic-viscoelastic material for heart tissue, see *MAT_HEART_TISSUE.
- Lung hyper-viscoelastic material, see *MAT_LUNG_TISSUE.
- Compression/tension plasticity model, see *MAT_PLASTICITY_COMPRESSION_TENSION.
- The Lund strain rate model, *MAT_STEINBERG_LUND, is added to Steinberg-Guinan plasticity model.
- Rate sensitive foam model, *MAT_FU_CHANG_FOAM, has been extended to include engineering strain rates, etc.
- Model, *MAT_MODIFIED_PIECEWISE_LINEAR_PLASTICITY, is added for modeling the failure of aluminum.
- Material model, *MAT_SPECIAL_ORTHOTROPIC, added for television shadow mask problems.
- Erosion strain is implemented for material type, *MAT_BAMMAN_DAMAGE.
- The equation of state, *EOS_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.

INTRODUCTION

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

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

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

- Fixed rigid walls PLANAR and PLANAR_FINITE are represented in the binary output file by a single shell element.
- Interference fits can be modeled with the INTERFERENCE option in contact.
- A layered shell theory is implemented for several constitutive models including the composite models to more accurately represent the shear stiffness of laminated shells.
- Damage mechanics is available to smooth the post-failure reduction of the resultant forces in the constitutive model *MAT_SPOTWELD_DAMAGE.
- Finite elastic strain isotropic plasticity model is available for solid elements. *MAT_FINITE_ELASTIC_STRAIN_PLASTICITY.
- A shape memory alloy material is available: *MAT_SHAPE_MEMORY.
- Reference geometry for material, *MAT_MODIFIED_HONEYCOMB, can be set at arbitrary relative volumes or when the time step size reaches a limiting value. This option is now available for all element types including the fully integrated solid element.
- Non orthogonal material axes are available in the airbag fabric model. See *MAT_FABRIC.
- Other new constitutive models include for the beam elements:
 - *MAT_MODIFIED_FORCE_LIMITED
 - *MAT_SEISMIC_BEAM
 - *MAT_CONCRETE_BEAMfor shell and solid elements:
 - *MAT_ELASTIC_VISCOPLASTIC_THERMALfor the shell elements:
 - *MAT_GURSON
 - *MAT_GEPLASTIC_SRATE2000
 - *MAT_ELASTIC_VISCOPLASTIC_THERMAL
 - *MAT_COMPOSITE_LAYUP
 - *MAT_COMPOSITE_LAYUP
 - *MAT_COMPOSITE_DIRECTfor the solid elements:
 - *MAT_JOHNSON_HOLMQUIST_CERAMICS
 - *MAT_JOHNSON_HOLMQUIST_CONCRETE
 - *MAT_INV_HYPERBOLIC_SIN
 - *MAT_UNIFIED_CREEP
 - *MAT_SOIL_BRICK
 - *MAT_DRUCKER_PRAGER
 - *MAT_RC_SHEAR_WALLand for all element options a very fast and efficient version of the Johnson-Cook plasticity model is available:
 - *MAT_SIMPLIFIED_JOHNSON_COOK
- A fully integrated version of the type 16 shell element is available for the resultant constitutive models.
- A nonlocal failure theory is implemented for predicting failure in metallic materials. The keyword *MAT_NONLOCAL activates this option for a subset of elastoplastic constitutive models.
- A discrete Kirchhoff triangular shell element (DKT) for explicit analysis with three in plane integration points is flagged as a type 17 shell element. This element has much better bending behavior than the C0 triangular element.
- A discrete Kirchhoff linear triangular and quadrilateral shell element is available as a type 18 shell. This shell is for extracting normal modes and static analysis.
- A C0 linear 4-node quadrilateral shell element is implemented as element type 20 with drilling stiffness for normal modes and static analysis.
- An assumed strain linear brick element is available for normal modes and statics.
- The fully integrated thick shell element has been extended for use in implicit calculations.

INTRODUCTION

- A fully integrated thick shell element based on an assumed strain formulation is now available. This element uses a full 3D constitutive model which includes the normal stress component and, therefore, does not use the plane stress assumption.
- The 4-node constant strain tetrahedron element has been extended for use in implicit calculations.
- Relative damping between parts is available, see *DAMPING_RELATIVE (SMP only).
- Preload forces can be input for the discrete beam elements.
- Objective stress updates are implemented for the fully integrated brick shell element.
- Acceleration time histories can be prescribed for rigid bodies.
- Prescribed motion for nodal rigid bodies is now possible.
- Generalized set definitions, i.e., SET_SHELL_GENERAL etc. provide much flexibility in the set definitions.
- The command "sw4." will write a state into the dynamic relaxation file, D3DRLF, during the dynamic relaxation phase if the D3DRLF file is requested in the input.
- Added mass by PART ID is written into the MATSUM file when mass scaling is used to maintain the time step size, (SMP version only).
- Upon termination due to a large mass increase during a mass scaled calculation a print summary of 20 nodes with the maximum added mass is printed.
- Eigenvalue analysis of models containing rigid bodies is now available using BCSLIB-EXT solvers from Boeing. (SMP version only).
- Second order stress updates can be activated by part ID instead of globally on the *CONTROL_ACCURACY input.
- Interface frictional energy is optionally computed for heat generation and is output into the interface force file (SMP version only).
- The interface force binary database now includes the distance from the contact surface for the FORMING contact options. This distance is given after the nodes are detected as possible contact candidates. (SMP version only).
- Type 14 acoustic brick element is implemented. This element is a fully integrated version of type 8, the acoustic element (SMP version only).
- A flooded surface option for acoustic applications is available (SMP version only).
- Attachment nodes can be defined for rigid bodies. This option is useful for NVH applications.
- CONSTRAINED_POINTS tie any two points together. These points must lie on a shell elements.
- Soft constraint is available for edge to edge contact in type 26 contact.
- CONSTRAINED_INTERPOLATION option for beam to solid interfaces and for spreading the mass and loads. (SMP version only).
- A database option has been added that allows the output of added mass for shell elements instead of the time step size.
- A new contact option allows the inclusion of all internal shell edges in contact type *CONTACT_GENERAL, type 26. This option is activated by adding _INTERIOR after the GENERAL keyword.
- A new option allows the use deviatoric strain rates rather than total rates in material model 24 for the Cowper-Symonds rate model.
- The CADFEM option for ASCII databases is now the default. Their option includes more significant figures in the output files.
- When using deformable spot welds, the added mass for spot welds is now printed for the case where global mass scaling is activated. This output is in the log file, D3HSP file, and the MESSAG file.
- Initial penetration warnings for edge-to-edge contact are now written into the MESSAG file and the D3HSP file.
- Each compilation of LS-DYNA is given a unique version number.

- Finite length discrete beams with various local axes options are now available for material types 66, 67, 68, 93, and 95. In this implementation the absolute value of SCOOR must be set to 2 or 3 in the *SECTION_BEAM input.
- New discrete element constitutive models are available:
 - *MAT_ELASTIC_SPRING_DISCRETE_BEAM
 - *MAT_INELASTIC_SPRING_DISCRETE_BEAM
 - *MAT_ELASTIC_6DOF_SPRING_DISCRETE_BEAM
 - *MAT_INELASTIC_6DOF_SPRING_DISCRETE_BEAMThe latter two can be used as finite length beams with local coordinate systems.
- Moving SPC's are optional in that the constraints are applied in a local system that rotates with the 3 defining nodes.
- A moving local coordinate system, CID, can be used to determine orientation of discrete beam elements.
- Modal superposition analysis can be performed after an eigenvalue analysis. Stress recovery is based on type 18 shell and brick (SMP only).
- Rayleigh damping input factor is now input as a fraction of critical damping, i.e. 0.10. The old method required the frequency of interest and could be highly unstable for large input values.
- Airbag option "SIMPLE_PRESSURE_VOLUME" allows for the constant CN to be replaced by a load curve for initialization. Also, another load curve can be defined which allows CN to vary as a function of time during dynamic relaxation. After dynamic relaxation CN can be used as a fixed constant or load curve.
- Hybrid inflator model utilizing CHEMKIN and NIST databases is now available. Up to ten gases can be mixed.
- Option to track initial penetrations has been added in the automatic SMP contact types rather than moving the nodes back to the surface. This option has been available in the MPP contact for some time. This input can be defined on the fourth card of the *CONTROL_CONTACT input and on each contact definition on the third optional card in the *CONTACT definitions.
- If the average acceleration flag is active, the average acceleration for rigid body nodes is now written into the D3THDT and NODOUT files. In previous versions of LS-DYNA, the accelerations on rigid nodes were not averaged.
- A capability to initialize the thickness and plastic strain in the crash model is available through the option *INCLUDE_STAMPED_PART, which takes the results from the LS-DYNA stamping simulation and maps the thickness and strain distribution onto the same part with a different mesh pattern.
- A capability to include finite element data from other models is available through the option, *INCLUDE_TRANSFORM. This option will take the model defined in an INCLUDE file: offset all ID's; translate, rotate, and scale the coordinates; and transform the constitutive constants to another set of units.

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

Below is list of new capabilities and features:

- MPP decomposition can be controlled using *CONTROL_MPP_DECOMPOSITION commands in the input deck.
- The MPP arbitrary Lagrangian-Eulerian fluid capability now works for airbag deployment in both SMP and MPP calculations.

INTRODUCTION

- Euler-to-Euler coupling is now available through the keyword *CONSTRAINED_EULER_TO_EULER.
- Up to ten ALE multi-material groups may now be defined. The previous limit was three groups.
- Volume fractions can be automatically assigned during initialization of multi-material cells. See the GEOMETRY option of *INITIAL_VOLUME_FRACTION.
- A new ALE smoothing option is available to accurately predict shock fronts.
- DATABASE_FSI activates output of fluid-structure interaction data to ASCII file DBFSI.
- Point sources for airbag inflators are available. The origin and mass flow vector of these inflators are permitted to vary with time.
- A majority of the material models for solid materials are available for calculations using the SPH (Smooth Particle Hydrodynamics) option.
- The Element Free Galerkin method (EFG or meshfree) is available for two-dimensional and three-dimensional solids. This new capability is not yet implemented for MPP applications.
- A binary option for the ASCII files is now available. This option applies to all ASCII files and results in one binary file that contains all the information normally spread between a large number of separate ASCII files.
- Material models can now be defined by numbers rather than long names in the keyword input. For example the keyword *MAT_PIECEWISE_LINEAR_PLASTICITY can be replaced by the keyword: *MAT_024.
- An embedded NASTRAN reader for direct reading of NASTRAN input files is available. This option allows a typical input file for NASTRAN to be read directly and used without additional input. See the *INCLUDE_NASTRAN keyword.
- Names in the keyword input can represent numbers if the *PARAMETER option is used to relate the names and the corresponding numbers.
- Model documentation for the major ASCII output files is now optional. This option allows descriptors to be included within the ASCII files that document the contents of the file.
- IDs have been added to the following keywords:
 - *BOUNDARY_PRESCRIBED_MOTION
 - *BOUNDARY_PRESCRIBED_SPC
 - *CONSTRAINED_GENERALIZED_WELD
 - *CONSTRAINED_JOINT
 - *CONSTRAINED_NODE_SET
 - *CONSTRAINED_RIVET
 - *CONSTRAINED_SPOTWELD
 - *DATABASE_CROSS_SECTION
 - *ELEMENT_MASS
- The *DATABASE_ADAMS keyword is available to output a modal neutral file d3mnf. This will be available upon customer request since it requires linking to an ADAMS library file.
- Penetration warnings for the contact option, /ignore initial penetration, are added as an option. Previously, no penetration warnings were written when this contact option was activated.
- Penetration warnings for nodes in-plane with shell mid-surface are printed for the AUTOMATIC contact options. Previously, these nodes were ignored since it was assumed that they belonged to a tied interface where an offset was not used; consequently, they should not be treated in contact.
- For the arbitrary spot weld option, the spot welded nodes and their contact segments are optionally written into the D3HSP file. See *CONTROL_CONTACT.
- For the arbitrary spot weld option, if a segment cannot be found for the spot welded node, an option now exists to error terminate. See *CONTROL_CONTACT.

- Spot weld resultant forces are written into the SWFORC file for solid elements used as spot welds.
- Solid materials have now been added to the failed element report and additional information is written for the `!node is deleted!` messages.
- A new option for terminating a calculation is available, `*TERMINATION_CURVE`.
- A 10-noded tetrahedron solid element is available with either a 4 or 5 point integration rule. This element can also be used for implicit solutions.
- A new 4 node linear shell element is available that is based on Wilson's plate element combined with a Pian-Sumihara membrane element. This is shell type 21.
- A shear panel element has been added for linear applications. This is shell type 22. This element can also be used for implicit solutions.
- A null beam element for visualization is available. The keyword to define this null beam is `*ELEMENT_PLOTEL`. This element is necessary for compatibility with NASTRAN.
- A scalar node can be defined for spring-mass systems. The keyword to define this node is `*NODE_SCALAR`. This node can have from 1 to 6 scalar degrees-of-freedom.
- A thermal shell has been added for through-thickness heat conduction. Internally, 8 additional nodes are created, four above and four below the mid-surface of the shell element. A quadratic temperature field is modeled through the shell thickness. Internally, the thermal shell is a 12 node solid element.
- A beam `OFFSET` option is available for the `*ELEMENT_BEAM` definition to permit the beam to be offset from its defining nodal points. This has the advantage that all beam formulations can now be used as shell stiffeners.
- A beam `ORIENTATION` option for orienting the beams by a vector instead of the third node is available in the `*ELEMENT_BEAM` definition for NASTRAN compatibility.
- Non-structural mass has been added to beam elements for modeling trim mass and for NASTRAN compatibility.
- An optional checking of shell elements to avoid abnormal terminations is available. See `*CONTROL_SHELL`. If this option is active, every shell is checked each time step to see if the distortion is so large that the element will invert, which will result in an abnormal termination. If a bad shell is detected, either the shell will be deleted or the calculation will terminate. The latter is controlled by the input.
- An offset option is added to the inertia definition. See `*ELEMENT_INERTIA_OFFSET` keyword. This allows the inertia tensor to be offset from the nodal point.
- Plastic strain and thickness initialization is added to the draw bead contact option. See `*CONTACT_DRAWBEAD_INITIALIZE`.
- Tied contact with offsets based on both constraint equations and beam elements for solid elements and shell elements that have 3 and 6 degrees-of-freedom per node, respectively. See `BEAM_OFFSET` and `CONSTRAINED_OFFSET` contact options. These options will not cause problems for rigid body motions.
- The segment-based (`SOFT=2`) contact is implemented for MPP calculations. This enables airbags to be easily deployed on the MPP version.
- Improvements are made to segment-based contact for edge-to-edge and sliding conditions, and for contact conditions involving warped segments.
- An improved interior contact has been implemented to handle large shear deformations in the solid elements. A special interior contact algorithm is available for tetrahedron elements.
- Coupling with MADYMO 6.0 uses an extended coupling that allows users to link most MADYMO geometric entities with LS-DYNA FEM simulations. In this coupling MADYMO contact algorithms are used to calculate interface forces between the two models.
- Release flags for degrees-of-freedom for nodal points within nodal rigid bodies are available. This makes the nodal rigid body option nearly compatible with the `RBE2` option in NASTRAN.

INTRODUCTION

- Fast updates of rigid bodies for metalforming applications can now be accomplished by ignoring the rotational degrees-of-freedom in the rigid bodies that are typically inactive during sheet metal stamping simulations. See the keyword: *CONTROL_RIGID.
- Center of mass constraints can be imposed on nodal rigid bodies with the SPC option in either a local or a global coordinate system.
- Joint failure based on resultant forces and moments can now be used to simulate the failure of joints.
- CONSTRAINED_JOINT_STIFFNESS now has a TRANSLATIONAL option for the translational and cylindrical joints.
- Joint friction has been added using table look-up so that the frictional moment can now be a function of the resultant translational force.
- The nodal constraint options *CONSTRAINED_INTERPOLATION and *CONSTRAINED_LINEAR now have a local option to allow these constraints to be applied in a local coordinate system.
- Mesh coarsening can now be applied to automotive crash models at the beginning of an analysis to reduce computation times. See the new keyword: *CONTROL_COARSEN.
- Force versus time seatbelt pretensioner option has been added.
- Both static and dynamic coefficients of friction are available for seat belt slip rings. Previously, only one friction constant could be defined.
- *MAT_SPOTWELD now includes a new failure model with rate effects as well as additional failure options.
- Constitutive models added for the discrete beam elements:
 - *MAT_1DOF_GENERALIZED_SPRING
 - *MAT_GENERAL_NONLINEAR_6dof_DISCRETE_BEAM
 - *MAT_GENERAL_NONLINEAR_1dof_DISCRETE_BEAM
 - *MAT_GENERAL_SPRING_DISCRETE_BEAM
 - *MAT_GENERAL_JOINT_DISCRETE_BEAM
 - *MAT_SEISMIC_ISOLATOR
- for shell and solid elements:
 - *MAT_plasticity_with_damage_ortho
 - *mat_simplified_johnson_cook_orthotropic_damage
 - *MAT_HILL_3R
 - *MAT_GURSON_RCDC
- for the solid elements:
 - *MAT_SPOTWELD
 - *MAT_HILL_FOAM
 - *mat_wood
 - *MAT_VISCOELASTIC_HILL_FOAM
 - *MAT_LOW_DENSITY_SYNTHETIC_FOAM
 - *MAT_RATE_SENSITIVE_POLYMER
 - *MAT_QUASILINEAR_VISCOELASTIC
 - *MAT_TRANSVERSELY_ANISOTROPIC_CRUSHABLE_FOAM
 - *MAT_VACUUM
 - *MAT_modified_crushable_foam
 - *MAT_PITZER_CRUSHABLE_FOAM
 - *MAT_JOINTED_ROCK
 - *MAT_SIMPLIFIED_RUBBER
 - *MAT_FHWA_SOIL
 - *MAT_SCHWER_MURRAY_CAP_MODEL
- Failure time added to MAT_EROSION for solid elements.
- Damping in the material models *MAT_LOW_DENSITY_FOAM and *MAT_LOW_DENSITY_VISCOUS_FOAM can now be a tabulated function of the smallest stretch ratio.

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

INTRODUCTION

DESCRIPTION OF KEYWORD INPUT

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

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

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

Alternatively, acceptable input could also be of the form:

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

A data block begins with a keyword followed by the data pertaining to the keyword. The next keyword encountered during the reading of the block data defines the end of the block and the beginning of a new block. A keyword must be left justified with the "*" contained in column one. A

INTRODUCTION

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

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

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

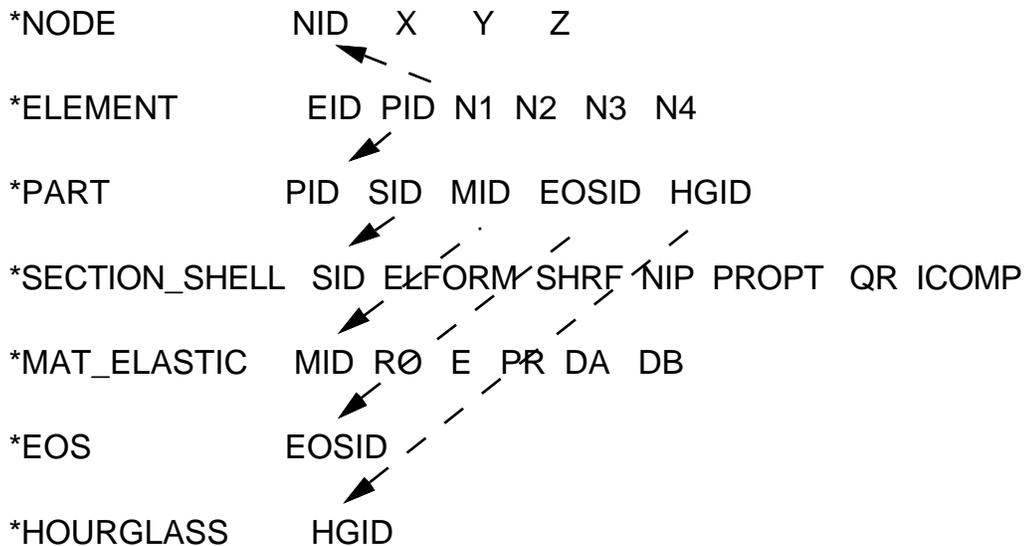


Figure I.1 Organization of the keyword input.

INTRODUCTION

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

*NODE

10101,x ,y ,z

10201,x ,y ,z

*ELEMENT_SHELL

10201,pid,n1,n2,n3,n4

10301,pid,n1,n2,n3,n4

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

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

*AIRBAG

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

*BOUNDARY

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

*COMPONENT

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

*CONSTRAINED

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

*CONTACT

This section is divided in to three main sections. The *CONTACT section allows the user to define many different contact types. These contact options are primarily for treating contact of deformable to deformable bodies, single surface contact in deformable bodies, deformable body to rigid body contact, and tying deformable structures with an option to release the tie based on plastic strain. The

INTRODUCTION

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

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.

INTRODUCTION

***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.

***INTEGRATION**

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

***INTERFACE**

Interface definitions are used to define surfaces, nodal lines, and nodal points for which the displacement and velocity time histories are saved at some user specified frequency. This data may then 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-DYNA execution line. When starting the second analysis, the name of the interface segment file created in the first run should be specified using the **L=**parameter on the LS-DYNA command line. Following the above procedure, multiple levels of sub-modeling are easily accommodated. The interface file may contain a multitude of interface definitions so that a single run of a full model can provide enough interface data for many component analyses. The interface feature represents a powerful extension of LS-DYNA's analysis capabilities.

***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-DYNA including springs, dampers, and seat belts. The material identifier, **MID**, points to the **MID** on the ***PART** card.

***NODE**

Define nodal point identifiers and their coordinates.

***PART**

This keyword serves two purposes.

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

***RIGIDWALL**

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

INTRODUCTION

***SET**

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

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

***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.

Topic	Component	Keyword
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,
- 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],

INTRODUCTION

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

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

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

SPATIAL DISCRETIZATION

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

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

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

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

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

frequently used since collapsed quadrilateral shell elements tend to lock and give very bad results. LS-DYNA automatically treats collapsed quadrilateral shell elements as C^0 triangular elements

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

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

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

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

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

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

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

The thick shell element is a shell element with only nodal translations for the eight nodes. The assumptions of shell theory are included in a non-standard fashion. It also uses hourglass control or selective-reduced integration. This element can be used in place of any four node shell element. It is favorably used for shell-brick transitions, as no additional constraint conditions are

INTRODUCTION

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 slippers are possible. The actions of the various seatbelt definitions can also be arbitrarily combined.

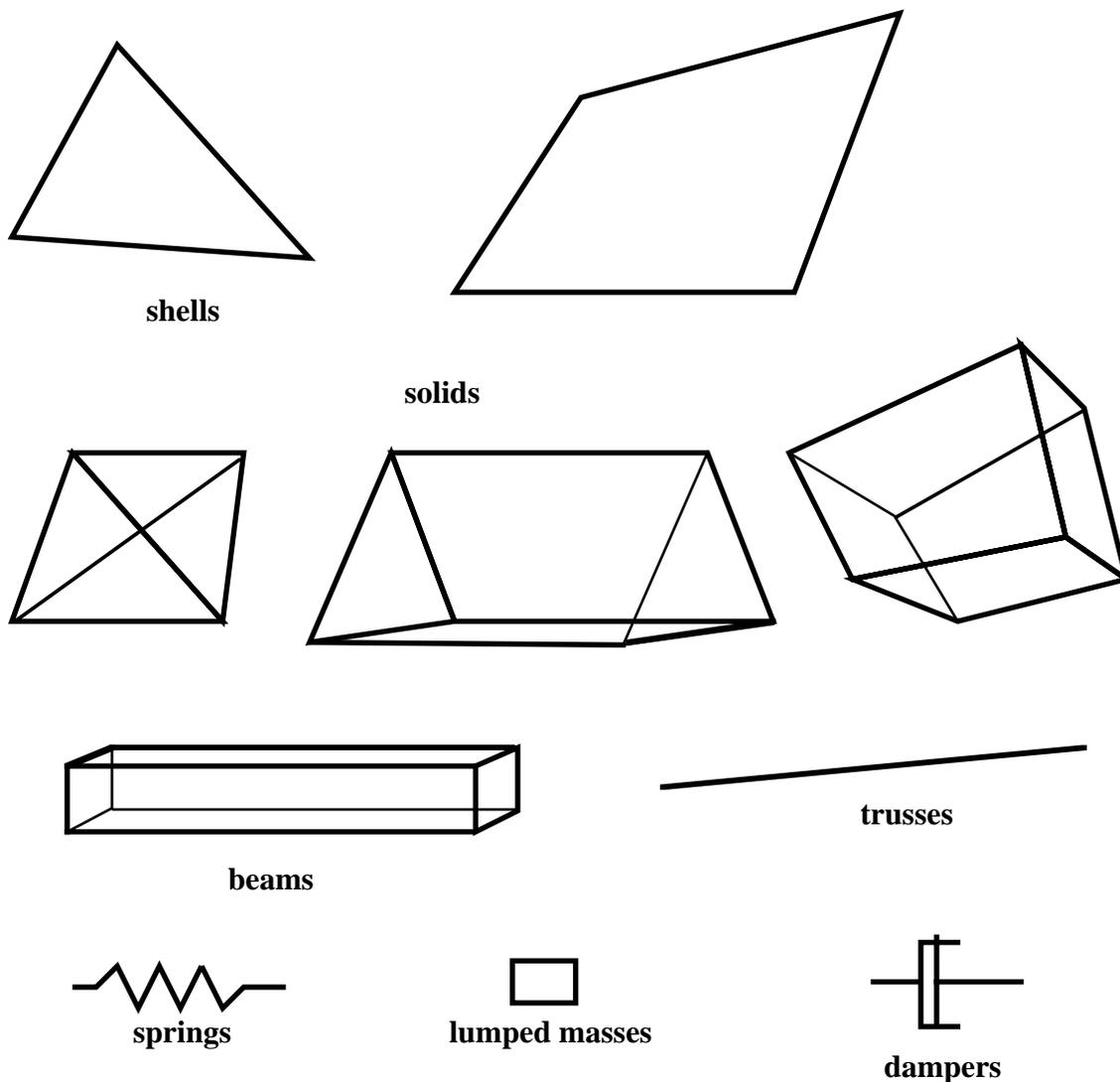


Figure I.2. Elements in LS-DYNA.

CONTACT-IMPACT INTERFACES

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

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

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

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

INTRODUCTION

INTERFACE DEFINITIONS FOR COMPONENT ANALYSIS

Interface definitions for component analyses are used to define surfaces, nodal lines, or nodal points (*INTERFACE_COMPONENTS) for which the displacement and velocity time histories are saved at some user specified frequency (*CONTROL_OUTPUT). This data may then 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.

CAPACITY

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

SENSE SWITCH CONTROLS

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

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

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

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

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

PRECISION

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

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

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

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

EXECUTION SYNTAX

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

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

where

- inf** = input file (user specified)
- otf** = high speed printer file (default=D3HSP)
- ptf** = binary plot file for graphics (default=D3PLOT)

INTRODUCTION

- dpf** = dump file for restarting (default=D3DUMP). This file is written at the end of every run and during the run as requested in the input. To stop the generation of this file set the file name to NODUMP.
- thf** = binary plot file for time histories of selected data (default=D3THDT)
- xtf** = binary plot file for time extra data (default=XTFIL)
- 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)
- epl** = echo file containing optional input echo with or without node/element data
- root** = root file name for general print option
- scl** = scale factor for binary file sizes (default=7)
- cpu** = cpu limit in seconds, applies to total calculation not just cpu from a restart
- kill** = if LS-DYNA encounters this file name it will terminate with a restart file (default=D3KIL)
- vda** = VDA/IGES database for geometrical surfaces
- c3d** = CAL3D input file
- nwds** = Number of words to be allocated. On engineering workstations a word is usually 32bits. **This number is ignored if memory is specified on the *KEYWORD card at the beginning of the input deck.**
- ncpu** = Overrides NCPU and CONST defined in *CONTROL_PARALLEL. A positive value sets CONST=2 and a negative values sets CONST=1. See *CONTROL_PARALLEL for an explanation of these parameters.
- npara** = Overrides PARA defined in *CONTROL_PARALLEL.
- time** = Overrides ENDTIM defined in *CONTROL_TERMINATION.
- ncycle** = Overrides ENDCYC defined in *CONTROL_TERMINATION.

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

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

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

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

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

File Organization

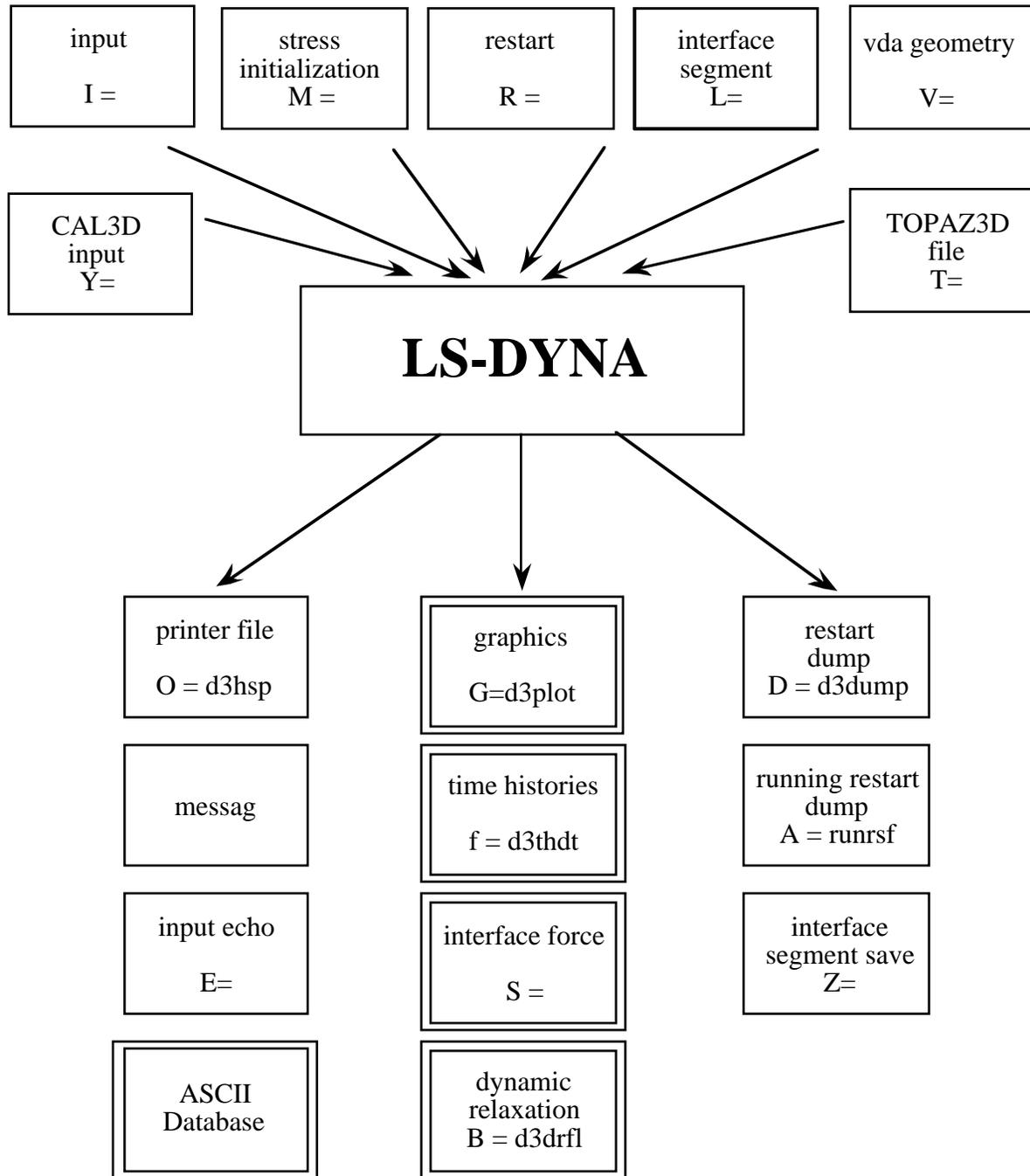


Figure I.3

INTRODUCTION

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

When restarting from a dump file, the execution line becomes

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

where

rtf = restart filename.

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

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

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

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

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

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

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

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

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

and in the second by:

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

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

```
I=inf
```

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

```
I=inf R=rtf
```

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

RESTART ANALYSIS

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

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

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

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

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

LS-DYNA R=*D3DUMPnn*

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

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

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

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

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

INTRODUCTION

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.

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 TRUEGRID¹, PATRAN, FEMB, HYPERMESH, and MEDINA. Several third-party translation programs are available for PATRAN and IDEAS.

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

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

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

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

Capabilities specialized to automotive applications:

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

LS-PREPOST

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

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

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

File Organization

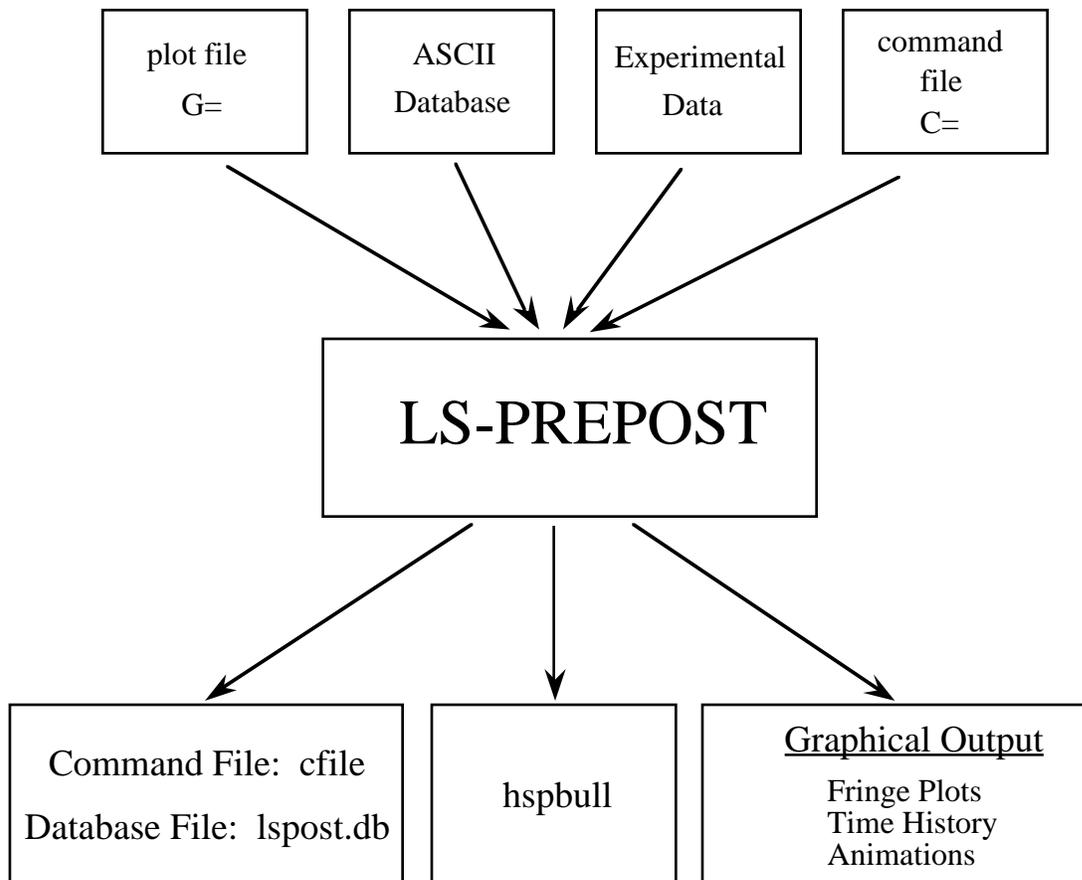


Figure I.4

EXECUTION SPEEDS

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

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

These relative timings are very approximate. Each interface node of the sliding interfaces is roughly equivalent to one-half zone cycle in cost. Figure I.5 illustrates the relative cost of the various shell formulations in LS-DYNA.

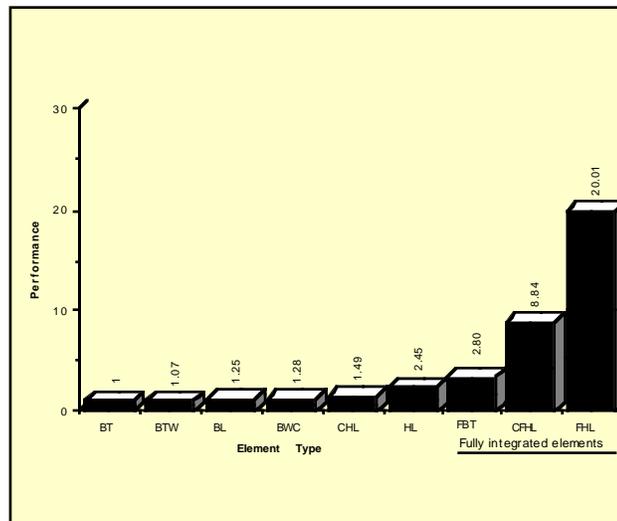


Figure I.5. Relative cost of the four noded shells available in LS-DYNA where BT is the Belytschko-Tsay shell, BTW is the Belytschko-Tsay shell with the warping stiffness taken from the Belytschko-Wong-Chiang, BWC, shell. The BL shell is the Belytschko-Leviathan shell. CHL denotes the Hughes-Liu shell, 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	VEZ		
Type	I	I	F	F	F	I		
Default	none	none	1.0	1.0	0	1		
Remarks	1			2		3		

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

***AIRBAG**

Purpose: Define an airbag or control volume.

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

***AIRBAG_OPTION1_{OPTION2}_{OPTION3}_{OPTION4}**
***AIRBAG_INTERACTION**
***AIRBAG_REFERENCE_GEOMETRY_OPTION_OPTION**

***AIRBAG_OPTION1_{OPTION2}_{OPTION3}_{OPTION4}**

OPTION1 specifies one of the following thermodynamic relationships:

SIMPLE_PRESSURE_VOLUME
SIMPLE_AIRBAG_MODEL
ADIABATIC_GAS_MODEL
WANG_NEFSKE
WANG_NEFSKE_JETTING
WANG_NEFSKE_MULTIPLE_JETTING
LOAD_CURVE
LINEAR_FLUID
HYBRID
HYBRID_JETTING
HYBRID_CHEMKIN

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

POP

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

CM

OPTION4 given by:

ID

VARIABLE	DESCRIPTION
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 ($V_{femodel}$) and applies a pressure ($P_{femodel}$). The thermodynamics of a control volume may be computed in a different unit system; thus, there is a separate volume ($V_{cvolume}$) and pressure ($P_{cvolume}$) which are used for integrating the differential equations for the control volume. The conversion is as follows:

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

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

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

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

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.

*AIRBAG

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

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

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 for the user subroutine.

*AIRBAG

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

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

Acceleration/Velocity/Displacement Activation

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

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

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

VARIABLE	DESCRIPTION
AX	Acceleration level in local x-direction to activate inflator. The absolute value of the x-acceleration is used. EQ.0: inactive.
AY	Acceleration level in local y-direction to activate inflator. The absolute value of the y-acceleration is used. EQ.0: inactive.
AZ	Acceleration level in local z-direction to activate inflator. The absolute value of the z-acceleration is used. EQ.0: inactive.
AMAG	Acceleration magnitude required to activate inflator. EQ.0: inactive.
TDUR	Time duration acceleration must be exceeded before the inflator activates. This is the 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.

*AIRBAG

Additional card required for SIMPLE_PRESSURE_VOLUME option

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

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

Remarks:

The relationship is the following:

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

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

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

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

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							

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>
CV	Heat capacity at constant volume
CP	Heat capacity at constant pressure
T	Temperature of input gas
LCID	Load curve ID specifying input mass flow rate. See *DEFINE_CURVE.
MU	Shape factor for exit hole, μ : LT.0.0: $ \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, ρ

*AIRBAG

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

Remarks:

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

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

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

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

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

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

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

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

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

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

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

Additional card required for ADIABATIC_GAS_MODEL option

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

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

Remarks:

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

$$p = PSF p_0$$

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

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

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

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

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

*AIRBAG

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:

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

Additional 4 cards are required for all WANG_NEFSKE models

Card 1 1 2 3 4 5 6 7 8

Variable	CV	CP	T	LCT	LCMT	TVOL	LCDT	IABT
Type	F	F	F	I	I	F	I	F
Default	none	none	0.	0	none	0.	0.	not used

Card 2 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

*AIRBAG

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
T	Temperature of input gas. For temperature variations a load curve, LCT, may be defined.
LCT	Optional load curve number defining temperature of input gas versus time. This overrides columns T.
LCMT	Load curve specifying input mass flow rate or tank pressure versus time. If the tank volume, TVOL, is nonzero the curve ID is assumed to be tank pressure versus time. If LCMT=0, then the inflator has to be modeled, see Card 4. During the dynamic relaxation phase the airbag is ignored unless the curve is flagged to act during dynamic relaxation.
TVOL	Tank volume which is required only for the tank pressure versus time curve, LCMT.
LCDT	Load curve for time rate of change of temperature (dT/dt) versus time.
IABT	Initial airbag temperature. (Optional, generally not defined.)
C23	Vent orifice coefficient which applies to exit hole. Set to zero if LCC23 is defined below.
LCC23	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

*AIRBAG

<u>VARIABLE</u>	<u>DESCRIPTION</u>
RO	Ambient density
GC	Gravitational conversion constant (mandatory - no default). If consistent units are being used for all parameters in the airbag definition then unity should be input.
LCEFR	Optional curve for exit flow rate versus (gauge) pressure
POVER	Initial relative overpressure (gauge), P_{over} in control volume
PPOP	Pop Pressure: relative pressure (gauge) for initiating exit flow, P_{pop}
OPT	Fabric venting option, if nonzero CP23, LCCP23, AP23, and LCAP23 are set to zero. EQ. 1: Wang-Nefske formulas for venting through an orifice are used. Blockage is not considered. EQ. 2: Wang-Nefske formulas for venting through an orifice are used. Blockage of venting area due to contact is considered. EQ. 3: Leakage formulas of Graefe, Krummheuer, and Siejak [1990] are used. Blockage is not considered. EQ. 4: Leakage formulas of Graefe, Krummheuer, and Siejak [1990] are used. Blockage of venting area due to contact is considered. EQ. 5: Leakage formulas based on flow through a porous media are used. Blockage is not considered. EQ. 6: Leakage formulas based on flow through a porous media are used. Blockage of venting area due to contact is considered.
KNKDN	<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/°K)
B	Second heat capacity coefficient of inflator gas, (e.g., Joules/mole/°K ²)

VARIABLE	DESCRIPTION
MW	Molecular weight of inflator gas (e.g., Kg/mole).
GASC	Universal gas constant of inflator gas (e.g., 8.314 Joules/mole/°K)
TDP	Time delay before initiating exit flow after pop pressure is reached.
AXP	Pop acceleration magnitude in local x-direction. EQ. 0.0: Inactive.
AYP	Pop acceleration magnitude in local y-direction. EQ. 0.0: Inactive.
AZP	Pop acceleration magnitude in local z-direction. EQ. 0.0: Inactive.
AMAGP	Pop acceleration magnitude. EQ. 0.0: Inactive.
TDURP	Time duration pop acceleration must be exceeded to initiate exit flow. This is a cumulative time from the beginning of the calculation, i.e., it is not continuous.
TDA	Time delay before initiating exit flow after pop acceleration is exceeded for the prescribed time duration.
RBIDP	Part ID of the rigid body for checking accelerations against pop accelerations.

Remarks:

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

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

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

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

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

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

*AIRBAG

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

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

where γ 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

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

and

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

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:

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

where C_o , A_o , and ρ_I are the inflator orifice coefficient, area, and gas density, respectively.

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

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

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

If OPT set to 3 or 4 then:

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

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:

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

*AIRBAG

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	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	1.0						
Remark	1	1	1	1	1	1		

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

Card 2 1 2 3 4 5 6 7 8

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

VARIABLE

DESCRIPTION

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

*AIRBAG

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

Remarks:

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

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

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

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

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

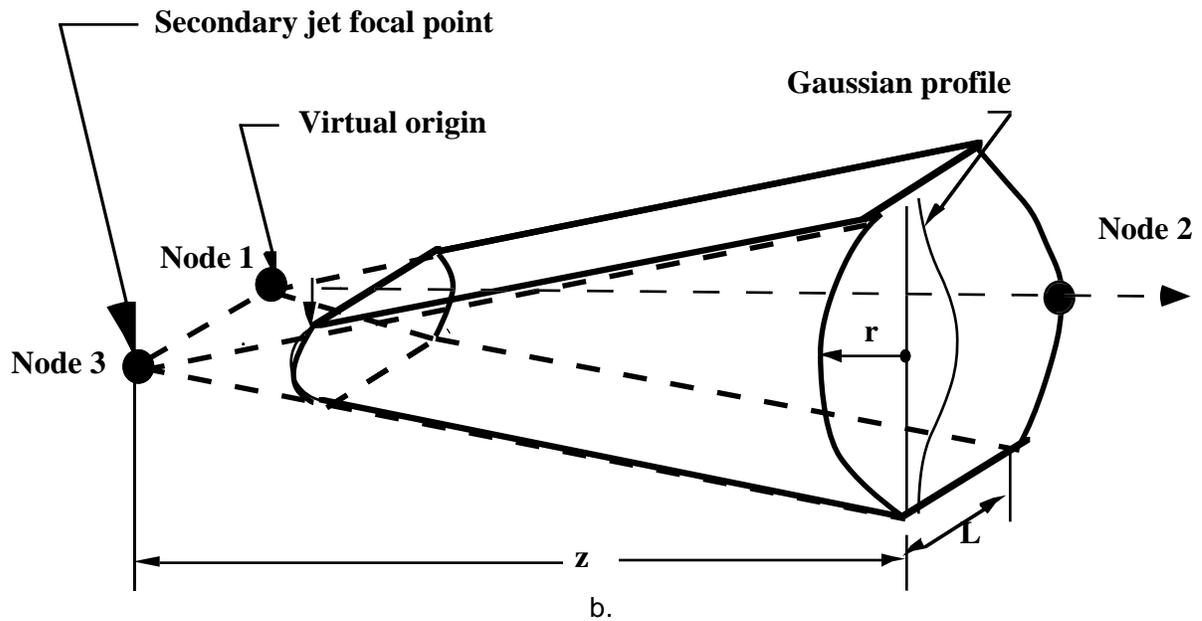
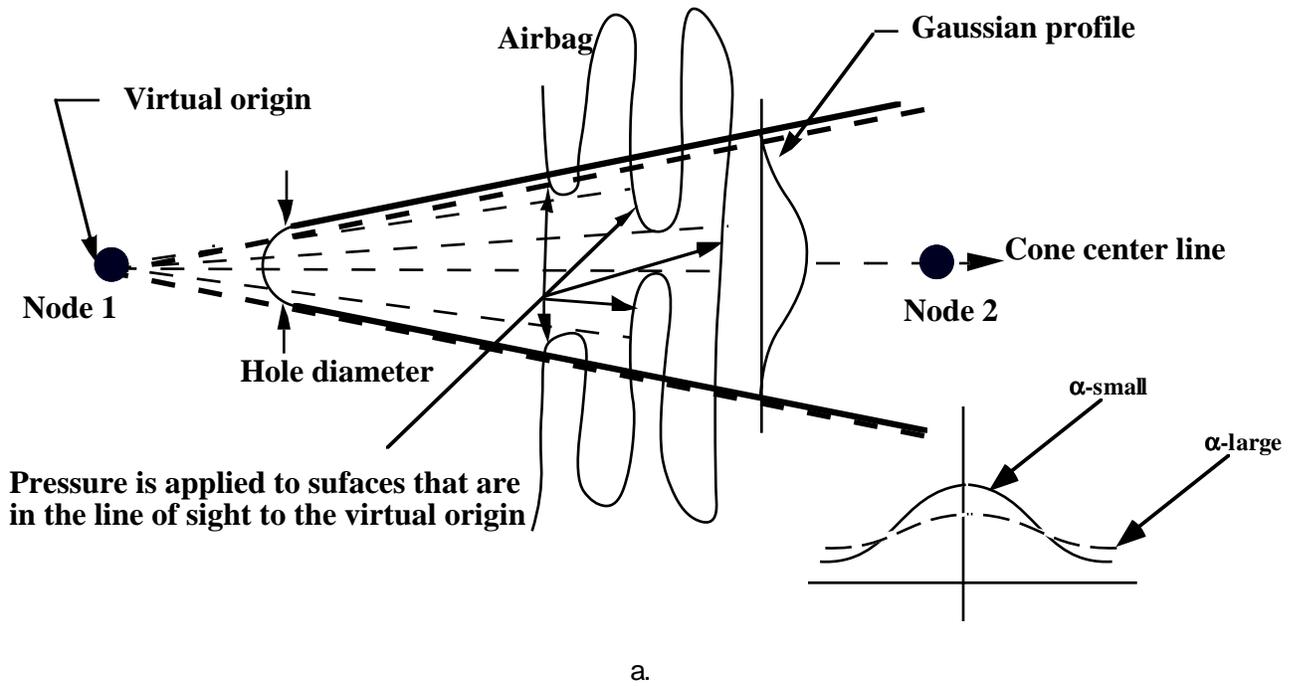


Figure 1.1 Jetting configuration for (a.) driver's side airbag (pressure applied only if centroid of surface is in line-of-sight) and (b.) the passenger's side bag.

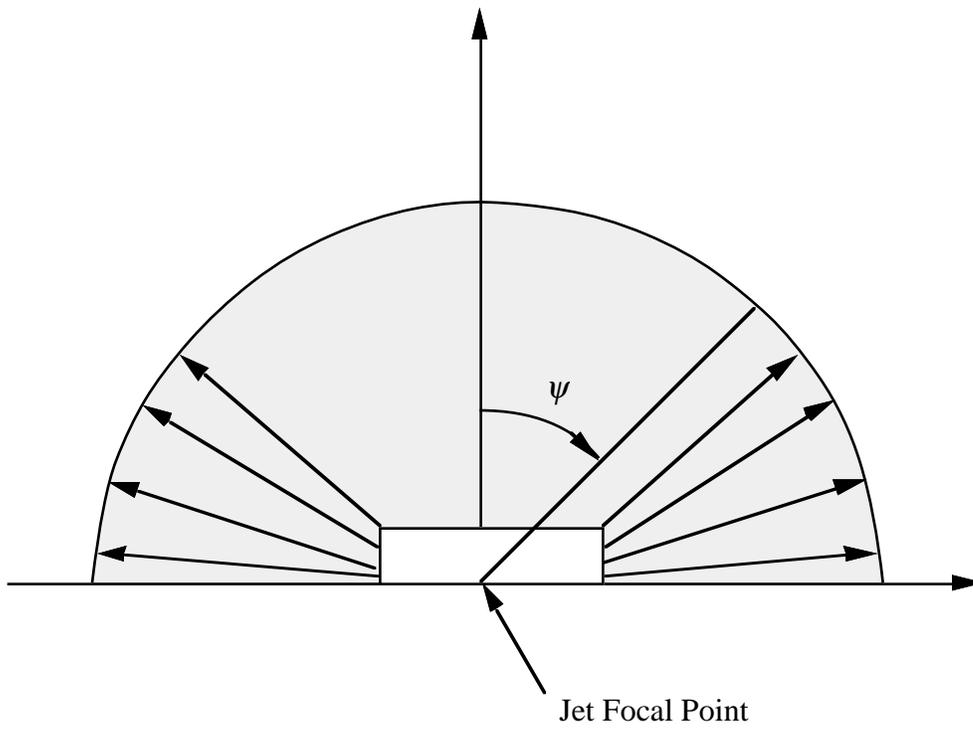


Figure 1.2 Multiple jet model for driver's side airbag.

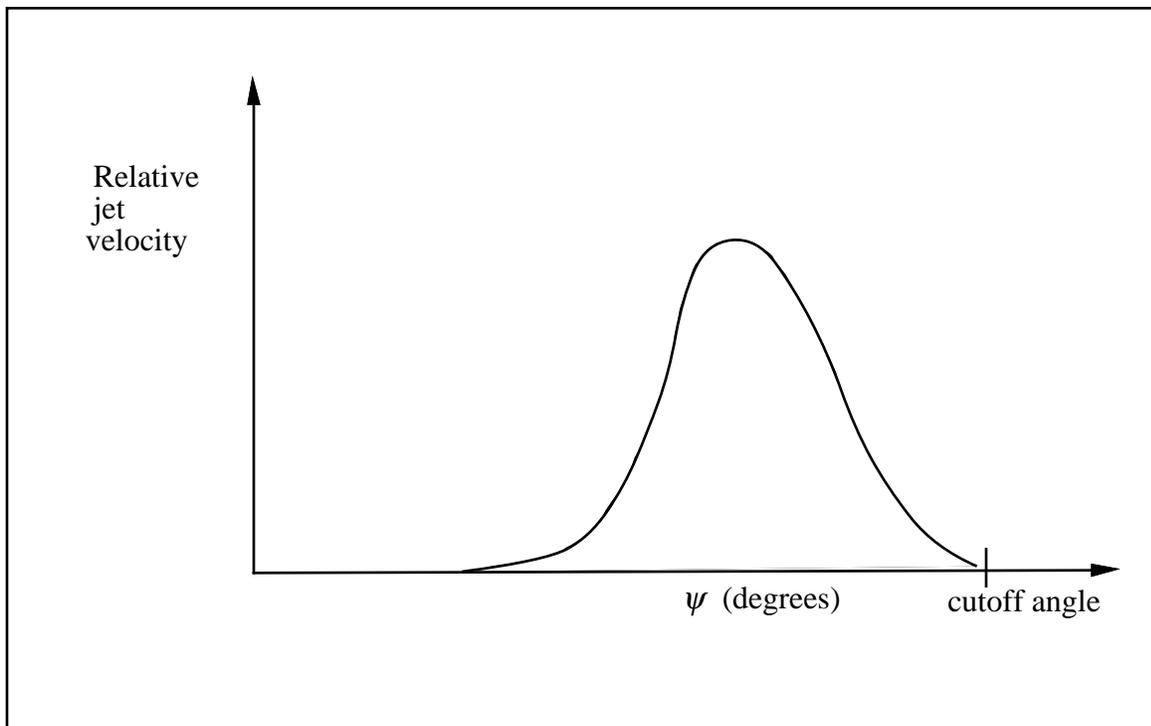


Figure 1.3 Normalized jet velocity versus angle for multiple jet driver's side airbag.

Further additional required for CM option.

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

Additional card required for _CM option

Card 1 1 2 3 4 5 6 7 8

Variable	NREACT								
Type	I								
Default	none								
Remark									

VARIABLE

DESCRIPTION

NREACT

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

Remarks:

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

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

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

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

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

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

*AIRBAG

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

$$v_{r,z} = v_{r=0,z} e^{-\left(\frac{r}{\alpha z}\right)^2}$$

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

Momentum at nozzle = Momentum at z

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

where

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

$F = \text{distance between jet foci for a passenger jet}$

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

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

By combining the equations above

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

The total force exerted by the jet is given by:

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

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

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

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.

*AIRBAG

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	ρ , density of the fluid
LCINT	$F(t)$ input flow curve defining mass per unit time as a function of time, see *DEFINE_CURVE.
LCOUTT	$G(t)$, output flow curve defining mass per unit time as a function of time. This load curve is optional.
LCOUTP	$H(p)$, output flow curve defining mass per unit time as a function of pressure. This load curve is optional.
LCFIT	$L(t)$, added pressure as a function of time. This load curve is optional.
LCBULK	Curve defining the bulk modulus as a function of time. This load curve is optional, but if defined, the constant, BULK, is not used.
LCID	Load curve ID defining pressure versus time, see *DEFINE_CURVE.

Remarks:

If LCID = 0 then the pressure is determined from:

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

where

$P(t)$ Pressure,

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

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

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

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

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.

*AIRBAG

Additional cards required for HYBRID and HYBRID_JETTING options

1 2 3 4 5 6 7 8

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

1 2 3 4 5 6 7 8

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

1 2 3 4 5 6 7 8

Variable	OPT	PVENT	NGAS					
Type	I	F	I					
Default	none	none	none					

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

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

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

<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.
A23	Vent orifice area which applies to exit hole. Set to zero if LCA23 is defined below.

*AIRBAG

<u>VARIABLE</u>	<u>DESCRIPTION</u>
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)
LCIDM	Load curve ID for inflator mass flow rate (eq. 0 for gas in the bag at time 0) GT.0: piece wise linear interpolation LT.0: cubic spline interpolation
LCIDT	Load curve ID for inflator gas temperature (eq.0 for gas in the bag at time 0) GT.0: piece wise linear interpolation LT.0: cubic spline interpolation
BLANK	(not used)
MW	Molecular weight
INITM	Initial mass fraction of gas component

<u>VARIABLE</u>	<u>DESCRIPTION</u>
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 ²)
C	Coefficient for molar heat capacity of inflator gas at constant pressure, (e.g., Joules/mole/°K ³)
FMASS	Fraction of additional aspirated mass.

*AIRBAG

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							
Remark	1	1	1	1	1	1		

Card 2 1 2 3 4 5 6 7 8

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

VARIABLE

DESCRIPTION

XJFP x-coordinate of jet focal point, i.e., the virtual origin in Figure 1.1. See Remark 1 below.

YJFP y-coordinate of jet focal point, i.e., the virtual origin in Figure 1.1.

ZJFP z-coordinate of jet focal point, i.e., the virtual origin in Figure 1.1.

XJVH x-coordinate of jet vector head to defined code centerline

VARIABLE	DESCRIPTION
YJVH	y-coordinate of jet vector head to defined code centerline
ZJVH	z-coordinate of jet vector head to defined code centerline
CA	Cone angle, α , defined in radians. LT.0.0: $ \alpha $ is the load curve ID defining cone angle as a function of <i>time</i>
BETA	Efficiency factor, β , 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 *SET_PART. If zero all elements are included in the airbag.
IDUM	Dummy field (Variable not used)
NODE1	Node ID located at the jet focal point, i.e., the virtual origin in Figure 1.1. See Remark 1 below.
NODE2	Node ID for node along the axis of the jet .
NODE3	Optional node ID located at secondary jet focal point.

Remarks:

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

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

*AIRBAG

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

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

2. This variable is not used and has been included to maintain the same format as the WANG_NEFSKE_JETTING options.
3. Care must be used to place the jet focal point within the bag. If the focal point is outside the bag, inside surfaces will not be visible so jetting pressure will not be applied correctly.

Additional cards required for HYBRID_CHEMKIN model

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

Card 1 1 2 3 4 5 6 7 8

Variable	LCIDM	LCIDT	NGAS	DATA	ATMT	ATMP	RG	
Type	I	I	I	I	F	F	F	
Default	none	none	none	none	none	none	none	

Card 2 1 2 3 4 5 6 7 8

Variable	HCONV							
Type	F							
Default	0.							

Card 3 1 2 3 4 5 6 7 8

Variable	C23	A23						
Type	F	F						
Default	0.	0.						

VARIABLE

DESCRIPTION

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

*AIRBAG

<u>VARIABLE</u>	<u>DESCRIPTION</u>
NGAS	Number of gas inputs to be defined below. (Including initial air)
DATA	Thermodynamic database EQ. 1. NIST database (3 additional property cards are required below) EQ. 2. CHEMKIN database (no additional property cards are required) EQ. 3. Polynomial data (1 additional property card is required below)
ATMT	Atmospheric temperature.
ATMP	Atmospheric pressure
RG	Universal gas constant
HCONV	Convection heat transfer coefficient
C23	Vent orifice coefficient
A23	Vent orifice area

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

Control Card

card 1 1 2 3 4 5 6 7 8

Variable	CHNAME	MW	LCIDN	FMOLE	FMOLET			
Type	A	F	I	F	F			
Default	none	none	0	none	0.			

<u>VARIABLE</u>	<u>DESCRIPTION</u>
CHNAME	Chemical symbol for this gas species (e.g., N2 for nitrogen, AR for argon). Required for DATA=2 (CHEMKIN), optional for DATA=1 or DATA=3.
MW	Molecular weight of this gas species.
LCIDN	Load curve specifying the input mole fraction versus time for this gas species. If >0, FMOLE is not used.
FMOLE	Mole fraction of this gas species in the inlet stream.
FMOLET	Initial mole fraction of this gas species in the tank.

Additional thermodynamic data cards for each gas species. No additional cards are needed if using the CHEMKIN database (DATA=2). However, the CHEMKIN database file with file name chemkin, must reside in the same directory that you are running LS-DYNA.

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

card 1 1 2 3 4 5 6 7 8

Variable	TLOW	TMID	THIGH					
Type	F	F	F					
Default	none	none	none					

card 2

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

Card 3

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

*AIRBAG

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

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

card 1 1 2 3 4 5 6 7 8

Variable	a	b	c	d	e			
Type	F	F	F	F	F			
Default	none	0.	0.	0.	0.			

<u>VARIABLE</u>	<u>DESCRIPTION</u>
a	Coefficient, see below.
b	Coefficient, see below.
c	Coefficient, see below.
d	Coefficient, see below.
e	Coefficient, see below.

Heat capacity curve fits:

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

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

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

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

*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: One way flow from AB1 to AB2 only. EQ.0: Two way flow between AB1 and AB2 GT.0: One way flow from AB2 to AB1 only.

Remarks:

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

All input options are valid for the following airbag types:

- *AIRBAG_SIMPLE_AIRBAG_MODEL
- *AIRBAG_WANG_NEFSKE
- *AIRBAG_WANG_NEFSKE_JETTING
- *AIRBAG_WANG_NEFSKE_MULTIPLE_JETTING
- *AIRBAG_HYBRID
- *AIRBAG_HYBRID_JETTING

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

- *AIRBAG_LOAD_CURVE
- *AIRBAG_LINEAR_FLUID

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

*AIRBAG

*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.

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

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

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 cards in this section are defined in alphabetical order:

- *ALE_MULTI-MATERIAL_GROUP**
- *ALE_REFERENCE_SYSTEM_CURVE**
- *ALE_REFERENCE_SYSTEM_GROUP**
- *ALE_REFERENCE_SYSTEM_NODE**
- *ALE_REFERENCE_SYSTEM_SWITCH**
- *ALE_SMOOTHING**
- *ALE_TANK_TEST**

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

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

For single gaseous material:

- *MAT_NULL**
- *EOS_LINEAR_POLYNOMIAL**
- *EOS_IDEAL_GAS**

For multiple gaseous material:

- *MAT_GAS_MIXTURE**
- *INITIAL_GAS_MIXTURE**

*ALE

*ALE_MULTI-MATERIAL_GROUP

Purpose: This command defines the appropriate ALE material groupings for interface reconstruction when many ALE Multi-Material Groups (AMMG) are present in a model. This card is required when ELFORM=11 in the *SECTION_SOLID card. This is the ALE Multi-Material element formulation requiring at least 2 ALE materials to be present in a model.

Card Format

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

<u>VARIABLE</u>	<u>DESCRIPTION</u>
SID	Set ID.
IDTYPE	Set type: EQ.0: Part set, EQ.1: Part.

Remarks:

- When ELFORM=12 in the *SECTION_SOLID card (single material and void), this card should not be used. In one model, ELFORM=12 cannot be used together with ELFORM=11.
- Each AMMG is given an ID (AMMGID), and consists of one or more PART ID's. The interface of each AMMGID is reconstructed as it evolves dynamically. Each AMMGID is represented by one material contour color in LS-PREPOST.
- The maximum number of AMMGIDs allowed has been increased to 20.
- To plot these AMMGIDs in LS-PREPOST:

[FCOMP] ⇒ [MISC] ⇒ [HISTORY VARIABLES 2] ⇒ [APPLY]

HISTORY VARIABLE 1 = density = Rho

HISTORY VARIABLE 2 = 1st AMMG ⇒ AMMGID = 1

HISTORY VARIABLE 3 = 2nd AMMG ⇒ AMMGID = 2

HISTORY VARIABLE n = nth AMMG ⇒ AMMGID = n

HISTORY VARIABLE NALEGP = NALEGPth AMMG ⇒ AMMGID = NALEGP

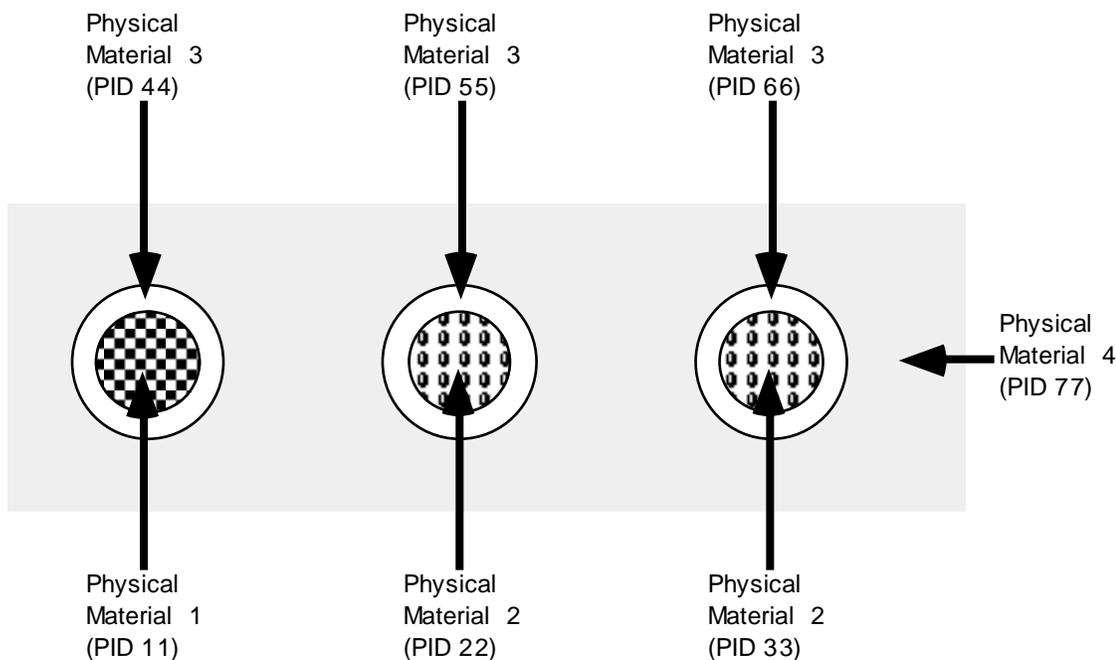
etc. (Note: Contour definitions maybe different for gas mixture application)

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

A PART may be any mesh component (simply a geometric entity). An AMMGID represents a material group which is treated as one material entity (represented by 1 material color contour in LS-PREPOST plotting). AMMGID is used in dealing with multiple ALE or Eulerian materials. For example, it can be used to specify a master ALE group in a coupling card.

Example 1

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



*ALE

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

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

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

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

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

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

Example2:

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

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

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

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

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

*ALE

*ALE_REFERENCE_SYSTEM_CURVE

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

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	ID							
Type	I							
Default	none							

Card 2 1 2 3 4 5 6 7 8

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

Card 3 1 2 3 4 5 6 7 8

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

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

Remark:

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

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

$f_1(t)$ is the value of load curve LCID1 at time t etc. Note that $f_1(t)$, $f_5(t)$, $f_9(t)$ correspond to the translation component and the remaining functions correspond to the rotation and deformation.

Example 1

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

```

$... | .....1..... | .....2..... | .....3..... | .....4..... | .....5..... | .....6..... | .....7..... | .....8
*ALE_REFERENCE_SYSTEM_GROUP
$      SID      STYPE      PRTYP      PRID      BCTRAN      BCEXP      BCROT
ICoord
      1          0          3          11          0          7          0
$      XC          YC          ZC      EXPLIM
      0          0          0          0
*ALE_REFERENCE_SYSTEM_CURVE
$ CURVESID
      11
$      LCID1      LCID2      LCID3      LCID4      LCID5      LCID6      LCID7      LCID8
      111          0          0          0          222          0          0          0
$      LCID9      LCID10      LCID11      LCID12
      0          0          0          0
*DEFINE_CURVE
111
0.0, 5.0
0.15, 4.0
*DEFINE_CURVE
222
0.0, -1.0
0.15, -5.0
$... | .....1..... | .....2..... | .....3..... | .....4..... | .....5..... | .....6..... | .....7..... | .....8

```

*ALE

*ALE_REFERENCE_SYSTEM_GROUP

Purpose: This command defines the type of reference system that may be prescribed for a geometric entity (where a geometric entity may be any part, part set, node set, or segment set). A geometric entity (or mesh) may be smoothed, moved, expanded, etc. according to the reference system type defined.

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	SID	STYPE	PRTYPE	PRID	BCTRAN	BCEXP	BCROT	ICOORD
Type	I	I	I	I	I	I	I	I
Default	none	0	0	0	0	0	0	0

Card 2 1 2 3 4 5 6 7 8

Variable	XC	YC	ZC	EXPLIM	EFAC			
Type	F	F	F	F	F			
Default	0.0	0.0	0.0	inf.	0.0			

VARIABLE	DESCRIPTION
-----------------	--------------------

SID Set ID.

STYPE Set type:
EQ.0: part set,
EQ.1: part,
EQ.2: node set,
EQ.3: segment set.

VARIABLE	DESCRIPTION
PRTYPE	Reference system type (See Remark 1 below) EQ.0: Eulerian, EQ.1: Lagrangian, EQ.2: Normal ALE mesh smoothing, EQ.3: Prescribed motion following load curves, see *ALE_REFERENCE_SYSTEM_CURVE, EQ.4: Automatic mesh motion following mass weighted average velocity in ALE mesh, EQ.5: Automatic mesh motion following coordinate system defined by three user defined nodes, see *ALE_REFERENCE_SYSTEM_NODE, EQ.6: Switching in time between different reference system types, see *ALE_REFERENCE_SYSTEM_SWITCH, EQ.7: Automatic mesh expansion in order to enclose up to twelve user defined nodes, see *ALE_REFERENCE_SYSTEM_NODE. EQ.8: Mesh smoothing option for shock waves, where the element grid contracts in the vicinity of the shock front. This may be referred to as the Delayed-ALE option. It controls how much the mesh is to be moved during the remap step. This option requires the definition of the 5th parameter in the 2nd card, EFAC; see below for definition.
PRID	ID of switch list (PRTYPE 6), node group (PRTYPE 5 or 7), or curve group (PRTYPE 3).
BCTRAN	Translational constraints (PRTYPE 3, 4, 5 and 7): EQ.0: no constraints, EQ.1: constrained x translation, EQ.2: constrained y translation, EQ.3: constrained z translation, EQ.4: constrained x and y translation, EQ.5: constrained y and z translation, EQ.6: constrained z and x translation, EQ.7: constrained x, y, and z translation.
BCEXP	Mesh expansion constraints (PRTYPE 3, 4, 5 and 7): EQ.0: no constraints, EQ.1: constrained x expansion, EQ.2: constrained y expansion, EQ.3: constrained z expansion, EQ.4: constrained x and y expansion, EQ.5: constrained y and z expansion, EQ.6: constrained z and x expansion, EQ.7: constrained x, y, and z expansion.
BCROT	Mesh rotation constraints (PRTYPE 3,4 5 and 7): EQ.0: no constraints, EQ.1: constrained x rotation, EQ.2: constrained y rotation, EQ.3: constrained z rotation, EQ.4: constrained x and y rotation, EQ.5: constrained y and z rotation, EQ.6: constrained z and x rotation,

*ALE

VARIABLE	DESCRIPTION
	EQ.7: constrained x, y, and z rotation.
ICOORD	Center of mesh expansion and rotation (PRTYPE 3, 4, 5 and 7): EQ.0: at center of gravity, EQ.1: at (XC,YC,ZC).
XC,YC,ZC	Center of mesh expansion.
EXPLIM	Limit ratio for mesh expansion and shrinkage. Each cartesian direction is treated separately. The distance between the nodes is not allowed to increase by more than a factor EXPLIM, or decrease to less than a factor 1/EXPLIM.
EFAC	Mesh contraction factor for PRTYPE=8 only, ranging between 0.0 and 1.0. Note: The smaller the value of EFAC, the better the mesh will follow the material flow in the vicinity of a shock front. However, a very small value might lead to severe mesh distortions.

Remarks:

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

Example 1

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

```

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

```

Example 2

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

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

*ALE

*ALE_REFERENCE_SYSTEM_NODE

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

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	ID							
Type	I							
Default	none							

Card Format

Card 1 1 2 3 4 5 6 7 8

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

Card Format

Card 1 1 2 3 4 5 6 7 8

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

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

Remark:

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

$$\begin{aligned}
 x' &= (x_2 - x_1) / |x_2 - x_1| \\
 z' &= x' \times (x_3 - x_1) / |x' \times (x_3 - x_1)| \\
 y' &= z' \times x'
 \end{aligned}$$

Note that $x_1 \rightarrow x_2$ is the local x' axis, $x_1 \rightarrow x_3$ is the local y' axis and x' crosses y' gives the local z' axis.

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

Example 1

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

```

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

```

*ALE

*ALE_REFERENCE_SYSTEM_SWITCH

Purpose: The PRTYP parameter in the *ALE_REFERENCE_SYSTEM_GROUP card allows (8) choices of the reference system types for any ALE geometric entity. This command allows for the time-dependent switches between these different types of reference systems, i.e., Lagrangian, Eulerian and ALE formulations, etc. during the simulation. This command is required only when PRTYPE=6 in *ALE_REFERENCE_SYSTEM_GROUP. Please see example 2 in the *ALE_REFERENCE_SYSTEM_GROUP section.

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	ID							
Type	I							
Default	none							

Card Format

Card 2 1 2 3 4 5 6 7 8

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

Card Format

Card 3 1 2 3 4 5 6 7 8

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

Card Format

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

<u>VARIABLE</u>	<u>DESCRIPTION</u>
ID	Switch list ID, see *ALE_REFERENCE_SYSTEM_GROUP,
T1...T7	Times for switching reference system type. By default, the reference system TYPE1 occurs between time=0 and time=T1, and TYPE2 occurs between time=T1 and time=T2, etc.
TYPE1...TYPE8	Reference system types: EQ.0: Eulerian, EQ.1: Lagrangian, EQ.2: Normal ALE mesh smoothing, EQ.3: Prescribed motion following load curves, see *ALE_REFERENCE_SYSTEM_CURVE, EQ.4: Automatic mesh motion following mass weighted average velocity in ALE mesh, EQ.5: Automatic mesh motion following coordinate system defined by three user defined nodes, see *ALE_REFERENCE_SYSEM_NODE, EQ.7: Automatic mesh expansion in order to enclose up to twelve user defined nodes, see *ALE_REFERENCE_SYSEM_NODE. EQ.8: Mesh smoothing option for shock waves, where the element grid contracts in the vicinity of the shock front. This may be referred to as the Delayed-ALE option. It controls how much the mesh is to be moved during the remap step.
ID1...ID8	IDs of the reference node group(s) or curve group(s) (PRTYPE 3, 5 or 7).

Remark:

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

*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:

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

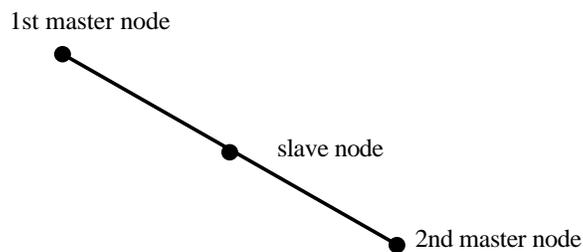


Figure 2.1 This simple constraint, which ensures that a slave node remains on a straight line between two master nodes, is sometimes necessary during ALE smoothing.

***ALE_TANK_TEST**

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

Card Format

	1	2	3	4	5	6	7	8
Variable	MDOTLC	TANKVOL	PAMB	PFINAL	MACHLIM	VELMAX	AORIF	
Type	I	F	F	F	F	F	F	
Default	0	0.0	0.0	0.0	0.0	0.0	0.0	
Variable	AMMGIDG	AMMGIDA	NUMPNT					
Type	I	I	I					
Default	0	0	50					

VARIABLE

DESCRIPTION

MDOTLC	LCID for mass flow rate as a function of time. This may be obtained directly from the control-volume type input data.
TANKVOL	Volume of the tank used in a tank test from which the tank pressure is measured, and $\dot{m}(t)$ and $\bar{T}_{gas}(t)$ are computed from this tank pressure data.
PAMB	The pressure inside the tank before jetting (usually 1bar).
PFINAL	The final equilibrated pressure inside the tank from the tank test.
MACHLIM	A limiting MACH number for the gas at the throat (MACH=1 preferred).
VELMAX	Maximum allowable gas velocity across the inflator orifice (not preferred).
AORIF	Total inflator orifice area (optional, only needed if the *SECTION_POINT_SOURCE card is not used).
AMMGIDG	The ALE multi-material group ID (AMMGID) of the gas.

*ALE

<u>VARIABLE</u>	<u>DESCRIPTION</u>
AMMGIDA	The ALE multi-material group ID (AMMGID) of the air.
NUMPNT	The number of points in $\dot{m}(t)$ and $\bar{T}_{gas}(t)$ curves. If NUMPNT=0, defaults to 50 points.

Remark:

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

If the *ALE TANK TEST card is present:

2. The definitions of the relative volume, $v_r(t)$ and $vel(t)$ curves in the *SECTION_POINT_SOURCE card will be ignored. They are computed internally inside LS-DYNA.
3. The $\dot{m}(t)$ curve will be read in on *ALE_TANK_TEST card.
4. The $\bar{T}_{gas}(t)$ curve (stagnation temperature) will be read in on *SECTION_POINT_SOURCE card (not $\bar{T}_{gas_corrected}(t)$). A fine distinction between the two temperatures may be made. $\bar{T}_{gas}(t)$ is derived directly from the tank pressure data based on a lump-parameter approach. $\bar{T}_{gas_corrected}(t)$ is computed from $\dot{m}(t)$ and $\bar{T}_{gas}(t)$ with additional isentropic and sonic flow assumption for the maximum velocity at an orifice ($\bar{T}_{gas_corrected}(t)$ is the static temperature). These assumptions are necessary since in $\dot{m}(t) = \rho(t) * vel(t) * A$, we only know $\dot{m}(t)$ (1 known) but we need $\rho(t)$ and $vel(t)$ (2 unknowns).
5. The inflator area is computed from the *SECTION_POINT_SOURCE card that has the AMMGID of the inflator gas in the *ALE_TANK_TEST card. If the *BOUNDARY_AMBIENT_EOS card is used instead of the *SECTION_POINT_SOURCE card, then the area may be input in this *ALE_TANK_TEST card.

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

Example:

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

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

Also available are

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

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

```

$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|...8
*PART
inflator gas
$      PID      SECID      MID      EOSID      HGID      GRAV      ADPOPT
TMID
      1          1          1          0          0          0          0
0
*PART
air inside the tank
$      PID      SECID      MID      EOSID      HGID      GRAV      ADPOPT
TMID
      2          2          2          0          0          0          0
0 *SECTION_SOLID
$      SECID      ELFORM      AET
      2          11          0
*ALE_MULTI-MATERIAL_GROUP
$      SID      SIDTYPE
      1          1
      2          1
*SECTION_POINT_SOURCE
$      SECID      LCIDT      LCIDVOLR      LCIDVEL      <= 3 curves in tempvolvel.k file
      1          2          4          5
$      NODEID      VECTID      AREA
      24485          3          15.066
...
      24557          3          15.066
*ALE_TANK_TEST
$      MDOTLCID      TANKVOL      PAMBIENT      PFINAL      MACH      VELMAX      ORIFAREA
      1          6.0E7      1.0E-4      5.288E-4      1.0          0.0
$AMMGIDGAS      AMMGIDAIR      NUMPOINT
      1          2          80
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|...8

```


***BOUNDARY**

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

- *BOUNDARY_ACOUSTIC_COUPLING**
- *BOUNDARY_AMBIENT_EOS**
- *BOUNDARY_CONVECTION_OPTION**
- *BOUNDARY_CYCLIC**
- *BOUNDARY_ELEMENT_METHOD_OPTION**
- *BOUNDARY_FLUX_OPTION**
- *BOUNDARY_MCOL**
- *BOUNDARY_NON_REFLECTING**
- *BOUNDARY_NON_REFLECTING_2D**
- *BOUNDARY_OUTFLOW_CFD_OPTION**
- *BOUNDARY_PRESCRIBED_CFD_OPTION**
- *BOUNDARY_PRESCRIBED_MOTION_{OPTION1}_{OPTION2}**
- *BOUNDARY_PRESSURE_CFD_SET**
- *BOUNDARY_PRESSURE_OUTFLOW_OPTION**
- *BOUNDARY_RADIATION_OPTION**
- *BOUNDARY_SLIDING_PLANE**
- *BOUNDARY_SPC_{OPTION1}_{OPTION2}**
- *BOUNDARY_SPH_FLOW**
- *BOUNDARY_SPH_SYMMETRY_PLANE**
- *BOUNDARY_SYMMETRY_FAILURE**
- *BOUNDARY_TEMPERATURE_OPTION**
- *BOUNDARY_THERMAL_WELD**
- *BOUNDARY_USA_SURFACE**

*BOUNDARY

*BOUNDARY_ACOUSTIC_COUPLING

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

Card Format

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

<u>VARIABLE</u>	<u>DESCRIPTION</u>
SSID	Segment set ID, see *SET_SEGMENT

Remark:

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

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

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

***BOUNDARY_AMBIENT_EOS**

Purpose: This command defines the IDs of 2 load curves: (1) internal energy per unit reference specific volume (or temperature if using *EOS_IDEAL_GAS) and (2) relative volume. These 2 curves completely prescribe the thermodynamic state as a function of time for any ALE or Eulerian part with an “ambient” type element formulation (please see remark 3).

Card Format

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

VARIABLE

DESCRIPTION

PID	The ambient Part ID for which the thermodynamic state is being defined.
LCID1	Load curve ID for internal energy per unit reference specific volume (or a temperature load curve ID if *EOS_IDEAL_GAS is being used).
LCID2	Load curve ID for relative volume, $v_r = \left(\frac{v}{v_0} = \frac{\rho_0}{\rho} \right)$.

Remarks:

1. In general, a thermodynamic state of a non-reacting and no phase change material may be defined by 2 thermodynamic variables. By defining [a] an internal energy per unit reference specific volume load curve (or a temperature load curve if using *EOS_IDEAL_GAS) and [b] a relative volume load curve, the pressure as a function of time for this ambient part ID can be computed directly via the equation of state (*EOS_).
2. A reference specific volume, $v_0 = \frac{1}{\rho_0}$, is the inverse of a reference density, ρ_0 . The reference density is defined as the density at which the material is under a reference or nominal state. Please refer to the *EOS section for additional explanation on this.
3. The internal energy per unit reference specific volume may be defined as $e_{ipv0} = \frac{C_v T}{v_0}$. The specific internal energy (or internal energy per unit mass) is defined as $C_v T$.

*BOUNDARY

4. This card is only to be used with “ambient” element type:
- ELFORM = 7, or
 - ELFORM = 11 and AET=4, or
 - ELFORM = 12 and AET=4.

Example:

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

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

***BOUNDARY_CONVECTION_OPTION**

Available options 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				

*BOUNDARY

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	LOC			
Type	I	F	I	F	I			
Default	none	1.0	none	1.0	0			

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

Remarks:

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

h heat transfer coefficient

$(T - T_\infty)$ temperature potential

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

***BOUNDARY_CYCLIC**

Purpose: Define nodes in boundary planes for cyclic symmetry.

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

Card Format

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

VARIABLE**DESCRIPTION**

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

*BOUNDARY

Remarks:

1. Each node set should lie on a plane, and these should generally be boundaries of the model.
2. Previous versions of LS-Dyna, prior to version 970, it was assumed that the nodes are correctly ordered within each set, i.e. the n'th node in NSID1 is equivalent to the n'th node in NSID2. If the ISORT flag is active, the nodes in NSID2 are automatically sorted to achieve equivalence, so the nodes can be picked by the quickest available method. However, for cyclic symmetry (IGLOBAL=0), it is assumed that the axis passes through the origin, i.e., only globally defined axes of rotation are possible.

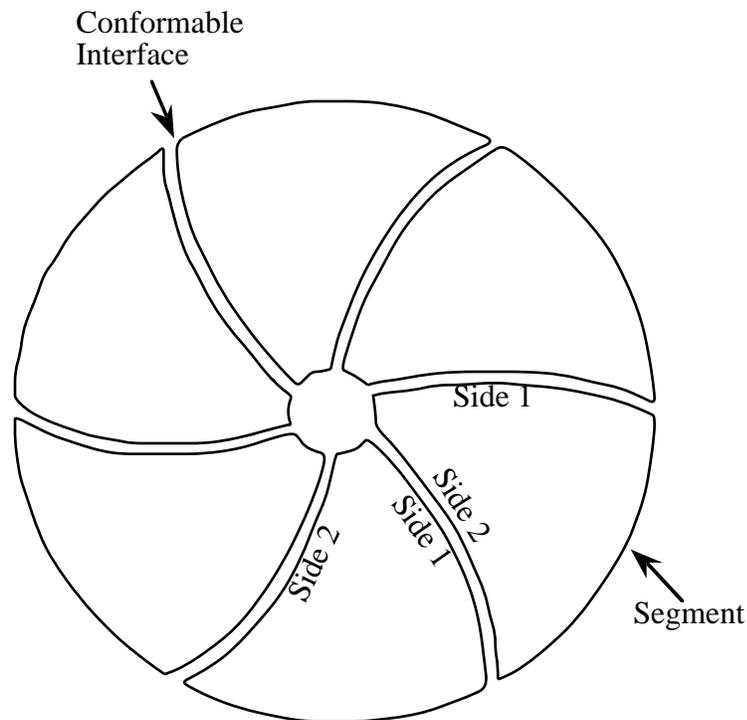


Figure 3.1 With cyclic symmetry only one segment is modeled.

*BOUNDARY_ELEMENT_METHOD_OPTION

Available options are:

CONTROL

FLOW

NEIGHBOR

SYMMETRY

WAKE

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

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

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

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

*BOUNDARY

*BOUNDARY_ELEMENT_METHOD_CONTROL

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

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

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

Define one card.

1 2 3 4 5 6 7 8

Variable	LWAKE	DTBEM	IUPBEM	FARBEM				
Type	I	F	I	F				
Default	50	0.	100	2.0				
Remark	1			2				

VARIABLE

DESCRIPTION

LWAKE

Number of elements in the wake of lifting surfaces. Wakes must be defined for all lifting surfaces.

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

Remarks:

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

*BOUNDARY

*BOUNDARY_ELEMENT_METHOD_FLOW

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

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

Define one card.

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

VARIABLE

DESCRIPTION

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

Remarks:

1. It is recommended that the shell segments in the SSID set use the NULL material (see *MAT_NULL). This will provide for the display of fluid pressures in the post-processor. For triangular shells the 4th node number should be the same as the 3rd node number. For fluid-structure interaction problems it is recommended that the boundary element shells use the same nodes and be coincident with the structural shell elements (or the outer face of solid elements) which define the surface of the body. This approach guarantees that the boundary element segments will move with the surface of the body as it deforms.
2. A pressure of PSTATIC is applied uniformly to all segments in the segment set. If the body of interest is hollow, then PSTATIC should be set to the free-stream static pressure minus the pressure on the inside of the body.
3. The effects of 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

*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			

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

Remarks:

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

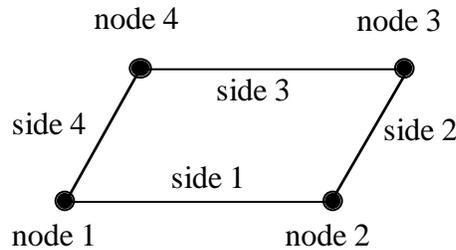


Figure 3.2 Each segment has 4 sides.

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

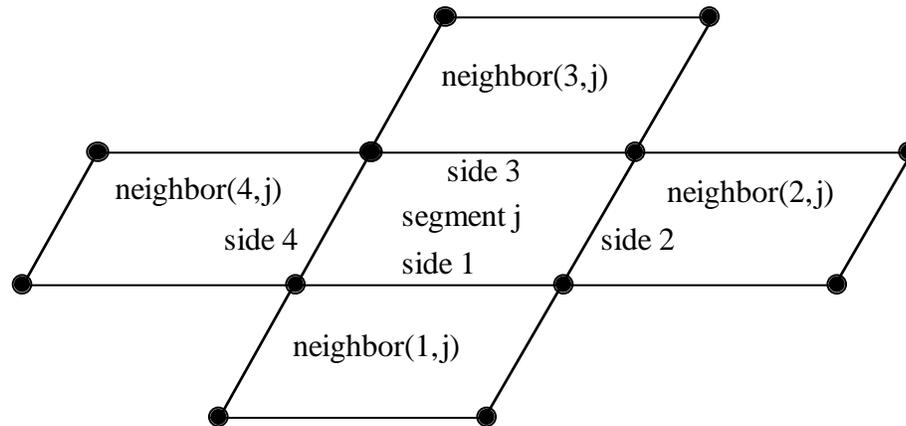


Figure 3.3 Typical neighbor specification.

There are several situations for which the user may desire to directly specify the neighbor array for certain elements. For example, boundary element wakes result in discontinuous doublet distributions, and neighbors which cross a wake should not be used. Figure 3.4 illustrates a situation where a wake is attached to side 2 of segment j . For this situation two options exist. If $\text{neighbor}(2,j)$ 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 .

*BOUNDARY

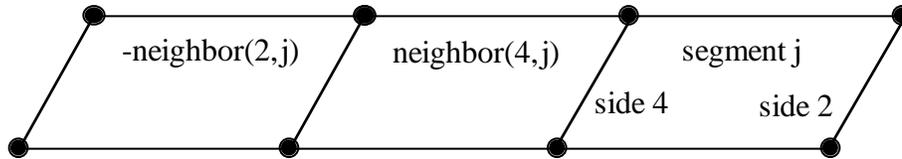


Figure 3.4 If $\text{neighbor}(2,j)$ is a negative number it is assumed to lie on the opposite side of $\text{neighbor}(4,j)$ as segment j .

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).

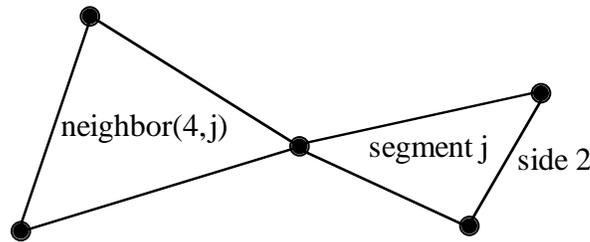


Figure 3.5 Sometimes another triangular boundary element segment can be used as $\text{neighbor}(4,j)$.

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):

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

*BOUNDARY_ELEMENT_METHOD_SYMMETRY

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

For the SYMMETRY option define the the following card:

Define one card.

1 2 3 4 5 6 7 8

Variable	BESYMS							
Type	I							
Default	0							
Remark								

VARIABLE

DESCRIPTION

BESYMS

Defines symmetry plane for boundary element method.

EQ. 0: no symmetry plane is defined

EQ. 1: $x = 0$ is a symmetry plane

EQ. 2: $y = 0$ is a symmetry plane

EQ. 3: $z = 0$ is a symmetry plane

***BOUNDARY_ELEMENT_METHOD_WAKE**

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

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

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

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

VARIABLE**DESCRIPTION**

NELEM

Element number to which a wake is attached.

NSIDE

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

Remarks:

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

*BOUNDARY

*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	LOC		
Type	I	F	F	F	F	I		
Default	none	1.0	1.0	1.0	1.0	0		

VARIABLE

DESCRIPTION

SSID	Segment set ID, see *SET_SEGMENT
N1,N2...	Node ID's defining segment
LCID	Load curve ID for heat flux, see *DEFINE_CURVE: GT.0: function versus time, EQ.0: use constant multiplier values at nodes, LT.0: function versus temperature.
MLC1	Curve multiplier at node N_1 , see Figure 3.2.
MLC2	Curve multiplier at node N_2 , see Figure 3.2.
MLC3	Curve multiplier at node N_3 , see Figure 3.2.
MLC4	Curve multiplier at node N_4 , see Figure 3.2.
LOC	Application of surface for thermal shell elements, see parameter, TSHELL, in the *CONTROL_SHELL input: EQ.-1: lower surface of thermal shell element EQ. 1: upper surface of thermal shell element

Remarks:

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.

*BOUNDARY

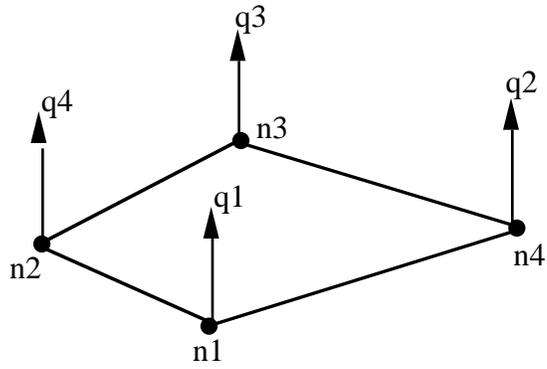


Figure 3.6. Nodal number determines outward normal.

***BOUNDARY_MCOL**

Purpose: Define parameters for MCOL coupling. The MCOL Program is a rigid body mechanics program for modeling the dynamics of ships. See Remark 1 for more information

Card Format

	1	2	3	4	5	6	7	8
Variable	NMCOL	MXSTEP	ENDTMCOL	TSUBC	PRTMCOL			
Type	I	I	F	F	F			
Default	2	none	0.0	0.0	none			
Remarks			2					

Card 2 must be defined for each ship

Card 2	1	2	3	4	5
Variable	RBMCOL	MCOFILE			
Type	I	A60			
Default		none			
Remarks					

VARIABLE

DESCRIPTION

NMCOL

Number of ships in MCOL coupling.

MXSTEP

Maximum of time step in MCOL calculation. If the number of MCOL time steps exceeds MXSTEP, then LS-DYNA will terminate.

ENDTMCOL

Uncoupling termination time, see Remark 2 below.
EQ. 0.0: set to LS-DYNA termination time

*BOUNDARY

<u>VARIABLE</u>	<u>DESCRIPTION</u>
TSUBC	Time interval for MCOL subcycling. EQ. 0.0: no subcycling
PRTMCOL	Time interval for output of MCOL rigid body data.
RBMCOL	LS-DYNA rigid body material assignment for the ship.
MCOLFILE	Filename containing MCOL input parameters for the ship.

Remarks:

1. The basis for MCOL is a convolution integral approach for simulating the equations of motion. A mass and inertia tensor are required as input for each ship. The masses are then augmented to include the effects of the mass of the surrounding water. A separate program determines the various terms of the damping/buoyancy force formulas which are also input to MCOL. The coupling is accomplished in a simple manner: at each time step LS-DYNA computes the resultant forces and moments on the MCOL rigid bodies and passes them to MCOL. MCOL then updates the positions of the ships and returns the new rigid body locations to LS-DYNA. A more detailed theoretical and practical description of MCOL can be found in a separate report (to appear).
2. After the end of the LS-DYNA / MCOL calculation, the analysis can be pursued using MCOL alone. ENDTMCOL is the termination time for this analysis. If ENDTMCOL is lower than the LS-DYNA termination time, the uncoupled analysis will not be activated.
3. The MCOL output is set to the files MCOLOUT (ship position) and MCOLENERGY (energy breakdown). In LS-PREPOST, MCOLOUT can be plotted through the rigid body time history option and MCOLENERGY.

***BOUNDARY_NON_REFLECTING**

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

Card Format

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

VARIABLE**DESCRIPTION**

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

Remarks:

1. Non-reflecting boundaries defined with this keyword are only used with three-dimensional solid elements. Boundaries are defined as a collection of segments, and segments are equivalent to element faces on the boundary. Segments are defined by listing the corner nodes in either a clockwise or counterclockwise order.
2. Non-reflecting boundaries are used on the exterior boundaries of an analysis model of an infinite domain, such as a half-space to prevent artificial stress wave reflections generated at the model boundaries 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.
4. During the dynamic relaxation phase (optional), nodes on non-reflecting segments are constrained in the normal direction. Nodal forces associated with these constraints are then applied as external loads and held constant in the transient phase while the constraints are replaced with the impedance matching functions. In this manner, soil can be quasi-statically prestressed during the dynamic relaxation phase and dynamic loads (with non-reflecting boundaries) subsequently applied in the transient phase.

*BOUNDARY

*BOUNDARY_NON_REFLECTING_2D

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

Card Format

1 2 3 4 5 6 7 8

Variable	NSID							
Type	I							
Default	none							
Remarks	1, 2							

VARIABLE

DESCRIPTION

NSID

Node set ID, see *SET_NODE. See Figure 3.7.

Remarks:

1. Non-reflecting boundaries defined with this keyword are only used with two-dimensional solid elements in either plane strain or axisymmetric geometries. Boundaries are defined as a sequential string of nodes moving counterclockwise around the boundary.
2. Non-reflecting boundaries are used on the exterior boundaries of an analysis model of an infinite domain, such as a half-space to prevent artificial stress wave reflections generated at the model boundaries 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.

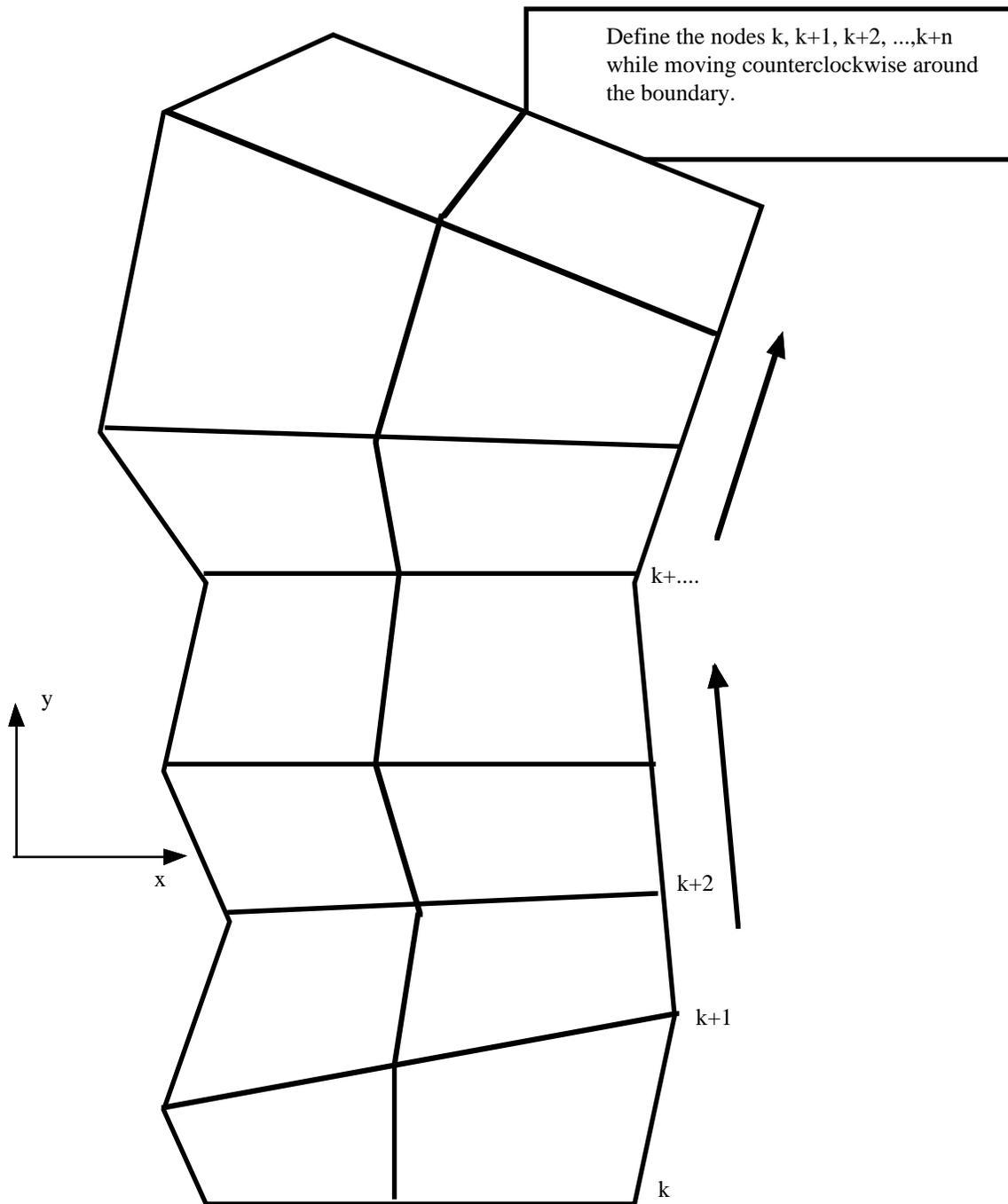


Figure 3.7 When defining a transmitting boundary in 2D define the node numbers in the node set in consecutive order while moving counterclockwise around the boundary.

***BOUNDARY**

***BOUNDARY_OUTFLOW_CFD_OPTION**

Available options include:

SEGMENT

SET

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

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

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	SSID								
Type	I								
Default	none								

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

Card Format

Card 1 1 2 3 4 5 6 7 8

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

VARIABLE

DESCRIPTION

SSID

Segment set ID.

N1, N2, ...

Node ID's defining segment.

Remarks:

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

*BOUNDARY

*BOUNDARY_PRESCRIBED_CFD_OPTION

Available options include:

NODE

SET

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

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	<i>typeID</i>	DOF		LCID	SF			
Type	I	I		I	F			
Default	none	none		none	1.			

VARIABLE

DESCRIPTION

typeID

Node ID (NID) or nodal set ID (NSID), see *SET_NODE.

DOF

Applicable degrees-of-freedom:
 EQ. 101: x-velocity,
 EQ. 102: y-velocity,
 EQ. 103: z-velocity,
 EQ. 104: Temperature,
 EQ. 107: turbulent kinetic energy,
 EQ. 110: turbulent eddy viscosity,
 EQ. 121: Species mass fraction-1,
 EQ. 122: Species mass fraction-2,
 EQ. 123: Species mass fraction-3,
 EQ. 124: Species mass fraction-4,
 EQ. 125: Species mass fraction-5,
 EQ. 126: Species mass fraction-6,
 EQ. 127: Species mass fraction-7,
 EQ. 128: Species mass fraction-8,
 EQ. 129: Species mass fraction-9,
 EQ. 130: Species mass fraction-10,
 EQ. 201: x-, y-, z-velocity,
 EQ. 202: x-, y-velocity,
 EQ. 203: x-, z-velocity,
 EQ. 204: y-, z-velocity,

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			

<u>VARIABLE</u>	<u>DESCRIPTION</u>
ID	Optional PRESCRIBED MOTION set ID to which this node, node set , or rigid body belongs. This ID does not need to be unique.
HEADING	An optional descriptor for the given ID that will be written into the D3HSP file and the BNDOUT file.
NID, NSID, PID	Node ID (NID), nodal set ID (NSID), SEE *SET_NODE, or part ID (PID), see *PART, for a rigid body.
DOF	<p>Applicable degrees-of-freedom:</p> <p>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. Not applicable to rigid bodies.</p>

*BOUNDARY

<u>VARIABLE</u>	<u>DESCRIPTION</u>
	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. Not applicable to rigid bodies. EQ.10: z/x degrees-of-freedom for node rotating about the y-axis at location (OFFSET1,OFFSET2) in the zx-plane, <i>point</i> (z,x). Radial motion is NOT permitted. Not applicable to rigid bodies. EQ.-10: z/x degrees-of-freedom for node rotating about the y-axis at location (OFFSET1,OFFSET2) in the zx-plane, <i>point</i> (z,x). Radial motion is permitted. Not applicable to rigid bodies. EQ.11: x/y degrees-of-freedom for node rotating about the z-axis at location (OFFSET1,OFFSET2) in the xy-plane, <i>point</i> (x,y). Radial motion is NOT permitted. Not applicable to rigid bodies. EQ.-11: x/y degrees-of-freedom for node rotating about the z-axis at location (OFFSET1,OFFSET2) in the xy-plane, <i>point</i> (x,y). Radial motion is permitted. Not applicable to rigid bodies.
VAD	Velocity/Acceleration/Displacement flag: EQ.0: velocity (rigid bodies and nodes), EQ.1: acceleration (rigid bodies and nodes only), EQ.2: displacement (rigid bodies and nodes). EQ.3: velocity versus displacement (rigid bodies and nodes) EQ.4: relative displacement (rigid bodies only)
LCID	Load curve ID to describe motion value versus time, see *DEFINE_CURVE.
SF	Load curve scale factor. (default=1.0)
VID	Vector ID for DOF values of 4 or 8, see *DEFINE_VECTOR.
DEATH	Time imposed motion/constraint is removed: EQ.0.0: default set to 10 ²⁸ .
BIRTH	Time imposed motion/constraint is activated.
OFFSET1	Offset for DOF types 9-11 (y, z, x direction)
OFFSET2	Offset for DOF types 9-11 (z, x, y direction)
MRB	Master rigid body for measuring the relative displacement.
NODE1	Optional orientation node, n1, for relative displacement
NODE2	Optional orientation node, n2, for relative displacement

Remarks:

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

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

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

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

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

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

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

where v_{cm} is the velocity of the center of mass, v_{node} is the specified nodal velocity, ω is the angular velocity of the rigid body, x_{cm} is the current coordinate of the mass center, and x_{node} is the current coordinate of the nodal point. Extreme care must be used when prescribing motion of a rigid body node. Typically, for nodes on a given rigid body, the motion of no more than one node should be prescribed or unexpected results may be obtained.

When the RIGID option is used to prescribe rotation of a rigid body, the axis of rotation will always be shifted such that it passes through the center-of-mass of the rigid body. By using *PART_INERTIA or *CONSTRAINED_NODAL_RIGID_BODY_INERTIA, one can override the internally-calculated location of the center-of-mass.

When the RIGID_LOCAL option is invoked, the orientation of the local coordinate system rotates with time in accordance with rotation of the rigid body.

***BOUNDARY_PRESSURE_CFD_SET**

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

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	SSID	LCID	P					
Type	I	I	F					
Default	none	none	none					
Remarks		1	1					

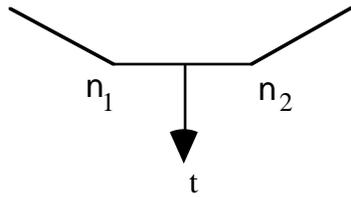
VARIABLE**DESCRIPTION**

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

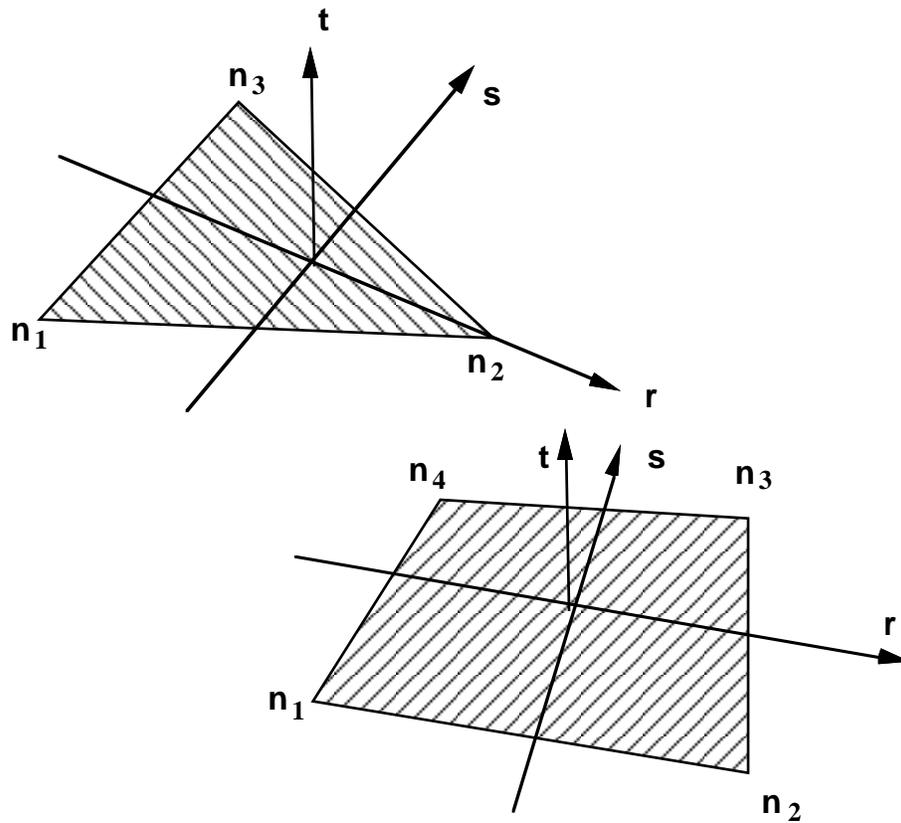
Remarks:

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

*BOUNDARY



a) 2-Dimensional definition for pressure boundary segments.



b) 3-Dimensional definition for pressure boundary segments

Figure 3.8. Nodal numbering for pressure boundary segments. Positive pressure acts in the negative t -direction. For two dimensional problems repeat the second node for the third and fourth nodes in the segment definitions.

***BOUNDARY_PRESSURE_OUTFLOW_OPTION**

Available options are

SEGMENT**SET**

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

For the SET option define the following card

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	SSID							
Type	I							
Default	none							

For the SEGMENT option define the following card

Card Format

Card 1 1 2 3 4 5 6 7 8

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

VARIABLE**DESCRIPTION**

SSID

Segment set ID

N1,N2...

Node ID's defining segment

*BOUNDARY

*BOUNDARY_RADIATION_OPTION

Available options are:

SEGMENT

SET

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

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

1. The first type, models radiation exchange between a finite element surface segment and the environment at temperature T_{∞} . The view factor between the finite element surface segment and the environment is 1.
2. The second type, models the radiation exchange between all the finite element segments that define a completely closed volume. The view factors between all the finite element segments defining the enclosure must be calculated and stored in a file named **viewfl**. With the **_SET** option multiple independent boundary radiation enclosures may be defined.

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

Card Format (Card 1 of 2)

Card 1 1 2 3 4 5 6 7 8

Variable	SSID	TYPE	RAD_GRP	FILE_NO				
Type	I	I	I	I				
Default	none	1	0	0				

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

Card Format (Card 1 of 2)

Card 1 1 2 3 4 5 6 7 8

Variable	N1	N2	N3	N4	TYPE			
Type	I	I	I	I	I			
Default	none	none	none	none	1			

Define the following card boundary radiation type 1 :

Card Format (Card 2 of 2)

	1	2	3	4	5	6	7	8
Variable	RFLCID	RFMULT	TILCID	TIMULT	LOC			
Type	I	F	I	F	I			
Default	none	1.0	none	1.0	0			

Define the following card boundary radiation type 2:

Card Format (Card 2 of 2) Note: The parameter "LOC" does not apply for radiation type 2.

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

<u>VARIABLE</u>	<u>DESCRIPTION</u>
SSID	Segment set ID, see *SET_SEGMENT
N1,N2...	Node ID's defining segment
TYPE	Radiation type: EQ.1: Radiation boundary to environment (default), EQ.2: Radiation in enclosure.
	The following two parameters are used for enclosure radiation definitions defined using the _SET option.
RAD_GRP	Radiation enclosure group ID. The segment sets from all radiation enclosure definitions with the same group ID are augmented to form a single enclosure definition. If RAD_GRP is not specified or set to zero, then the segments are placed in group zero. All segments defined by the _SEGMENT option are placed in set zero.

*BOUNDARY

<u>VARIABLE</u>	<u>DESCRIPTION</u>
FILE_NO	File number for view factor file. FILE_NO is added to viewfl_ to form the name of the file containing the view factors. For example if FILE_NO is specified as 22, then the view factors are read from viewfl_22 . For radiation enclosure group zero FILE_NO is ignored and view factors are read from viewfl . The same file may be used for different radiation enclosure group definitions.
RFLCID	Load curve ID for radiation factor f , see *DEFINE_CURVE: GT.0: function versus time, EQ.0: use constant multiplier value, RFMULT, LT.0: function versus temperature.
RFMULT	Curve multiplier for f , see *DEFINE_CURVE
TILCID	Load curve ID for T_∞ versus time, see *DEFINE_CURVE: EQ.0: use constant multiplier, TIMULT.
TIMULT	Curve multiplier for T_∞ , see *DEFINE_CURVE
LOC	Shell surface for thermal shell elements, see parameter, TSHELL, in the *CONTROL_SHELL input: EQ.-1: lower surface of thermal shell element EQ. 1: upper surface of thermal shell element
SELCID	Load curve ID for surface emissivity, see *DEFINE_CURVE: GT.0: function versus time, EQ.0: use constant multiplier value, SEMULT, LT.0: function versus temperature.
SEMULT	Curve multiplier for surface emissivity, see *DEFINE_CURVE

Remarks:

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

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

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_∞ for the boundary surface.

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

*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-component of vector defining normal or vector
VY	y-component of vector defining normal or vector
VZ	z-component of vector defining normal or vector
COPT	Option: EQ.0: node moves on normal plane, EQ.1: node moves only in vector direction.

Remarks:

Any node may be constrained to move on an arbitrarily oriented plane or line depending on the choice of COPT. Each boundary condition card defines a vector originating at (0,0,0) and terminating at the coordinates defined above. Since an arbitrary magnitude is assumed for this vector, the specified coordinates are non-unique and define only a direction. Use of *BOUNDARY_SPC is preferred over *BOUNDARY_SLIDING_PLANE as the boundary conditions imposed via the latter have been seen to break down somewhat in lengthy simulations owing to numerical roundoff.

***BOUNDARY_SPH_FLOW**

Purpose: Define a flow of particles. This option applies to continuum domains modeled with SPH elements.

Card 1 Format

Card 1 1 2 3 4 5 6 7 8

Variable	ID	STYP	DOF	VAD	LCID	SF	DEATH	BIRTH
Type	I	I	I	I	I	F	F	F
Default	none	none	none	0	none	1.	1.E+20	0.0

Card 2 Format

Card 2 1 2 3 4 5 6 7 8

Variable	NODE	VID						
Type	I	I						
Default	none	0						

VARIABLE**DESCRIPTION**

NSID, PID Nodal set ID (NSID), SEE *SET_NODE, or part ID (PID), see *PART.

STYP Set type:
 EQ. 1: part set ID, see *SET_PART,
 EQ. 2: part ID, see *PART,
 EQ. 3: node set ID, see *NODE_SET,

DOF Applicable degrees-of-freedom:
 EQ. 1: x-translational degree-of-freedom,
 EQ. 2: y-translational degree-of-freedom,
 EQ. 3: z-translational degree-of-freedom,
 EQ. 4: translational motion in direction given by the VID. Movement
 on plane normal to the vector is permitted.

*BOUNDARY

VARIABLE	DESCRIPTION
VAD	Velocity/Acceleration/Displacement flag applied to SPH elements before activation: EQ.0: velocity, EQ.1: acceleration, EQ.2: displacement.
LCID	Load curve ID to describe motion value versus time, see *DEFINE_CURVE.
SF	Load curve scale factor. (default=1.0)
DEATH	Time imposed motion/constraint is removed: EQ.0.0: default set to 10^{20} .
BIRTH	Time imposed motion/constraint is activated.
NODE	Node fixed in space which determines the boundary between activated particles and deactivated particles.
VID	Vector ID for DOF value of 4, see *DEFINE_VECTOR

Remarks:

Initially, the user defines the set of particles that are representing the flow of particles during the simulation. At time $t=0$, all the particles are deactivated which means that no particle approximation is calculated. The boundary of activation is a plane determined by the NODE, and normal to the vector VID. The particles are activated when they reached the boundary. Since they are activated, particle approximation is started.

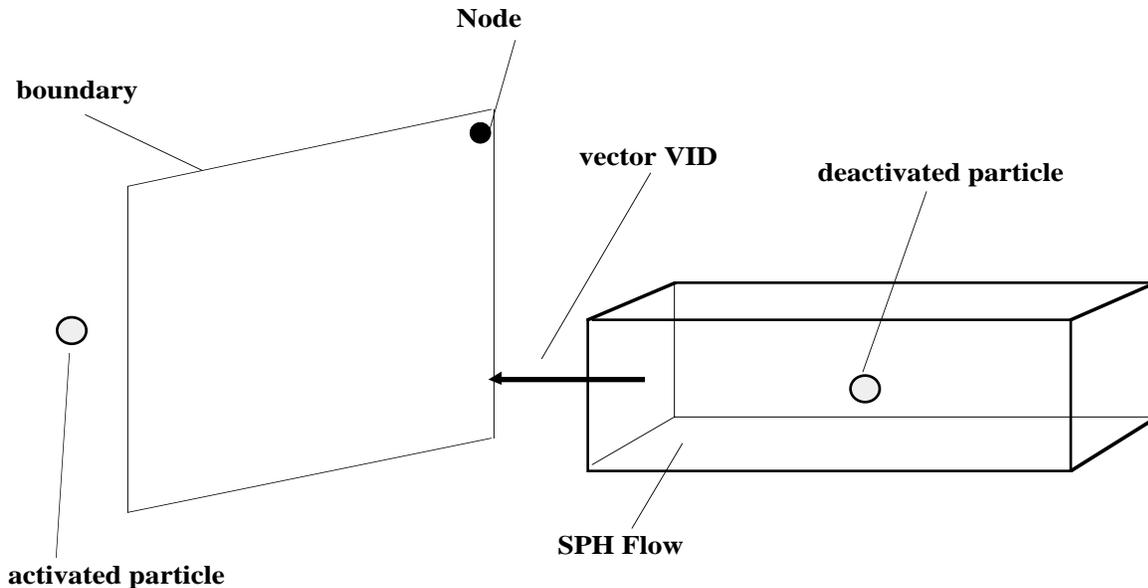


Figure 3.9. Vector **VID** determines the orientation of the SPH flow.

***BOUNDARY_SPH_SYMMETRY_PLANE**

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

Card Format

	1	2	3	4	5	6	7	8
Variable	VTX	VTY	VTZ	VHX	VHY	VHZ		
Type	F	F	F	F	F	F		
Default	0.	0.	0.	0.	0.	0.		

<u>VARIABLE</u>	<u>DESCRIPTION</u>
VTX	x-coordinate of tail of a normal vector originating on the wall (tail) and terminating in the body (head) (i.e., vector points from the symmetry plane into the body).
VTY	y-coordinate of tail
VTZ	z-coordinate of tail
VHX	x-coordinate of head
VHY	y-coordinate of head
VHZ	z-coordinate of head

Remarks:

1. A plane of symmetry is assumed for all SPH elements defined in the model.
2. The plane of symmetry has to be normal to either the x,y or z direction.

*BOUNDARY

*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_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	LOC				
Type	I	I	F	I				
Default	none	0	1.0	0				

VARIABLE**DESCRIPTION**

NID/SID	Node ID/Node Set ID, see *SET_NODE_OPTION
LCID	Load curve ID for temperature versus time: EQ.0: use the constant multiplier value given below by CMULT.
CMULT	Curve multiplier for temperature
LOC	Application of surface for thermal shell elements, see parameter, TSHELL, in the *CONTROL_SHELL input: EQ.-1: lower surface of thermal shell element EQ. 0: middle surface of thermal shell element EQ. 1: upper surface of thermal shell element

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

*BOUNDARY_THERMAL_WELD

Purpose: Define a moving heat source to model welding. Only applicable for a coupled thermal-structural simulations in which the weld source or workpiece is moving.

Card 1 Format

	1	2	3	4	5	6	7	8
Variable	PID	PTYP	NID	NFLAG	X0	Y0	Z0	N2ID
Type	I	I	I	I	F	F	F	I
Default	none	1	none	1	none	none	none	none

Card 2 Format

	1	2	3	4	5	6	7	8
Variable	a	b	c_f	c_r	LCID	Q	F_f	F_r
Type	F	F	F	F	I	F	F	F
Default	none	none	none	none	none	none	none	none

Optional Card 3 Format (define this card only if N2ID = -1 on card 1 above)

	1	2	3	4	5	6	7	8
Variable	tx	ty	tz					
Type	F	F	F					
Default	none	none	none					

<u>VARIABLE</u>	<u>DESCRIPTION</u>
PID	Part ID or Part Set ID to which weld source is applied
PTYP	PID type: EQ.1: PID defines a single part ID EQ.2: PID defines a part set ID
NID	Node ID giving location of weld source EQ.0: location defined by (X0,Y0,Z0) below
NFLAG	Flag controlling motion of weld source EQ.1: source moves with node NID EQ.2: source is fixed in space at original position of node NID
X0,Y0,Z0	Coordinates of weld source, which remains fixed in space (optional, ignored if NID nonzero above)
N2ID	Second node ID for weld beam aiming direction GT. 0: beam is aimed from N2ID to NID, moves with these nodes EQ.-1: beam aiming direction is (tx,ty,tz) input on optional card 3
a	weld pool width
b	weld pool depth (in beam aiming direction)
c _f	weld pool forward direction
c _r	weld pool rearward direction
LCID	load curve ID for weld energy input rate vs. time EQ.0: use constant multiplier value Q.
Q	curve multiplier for weld energy input rate [energy/time , e.g., Watt]
F _f	forward distribution function
F _r	reward distribution function (Note: F _f + F _r = 2.0)
tx,ty,tz	weld beam direction vector in global coordinates (N2ID = -1 only)

Remarks:

This boundary condition allows simulation of a moving weld heat source, following the work of Goldak (J. Goldak, "A New Finite Element Model for Welding Heat Sources", Metallurgical Transactions B, Volume 15B, June 1984, pp 299-305). Heat is generated in a ellipsoidal region centered at the weld source, and decaying exponentially with distance according to:

*BOUNDARY

$$q = \frac{6\sqrt{3}FQ}{\pi\sqrt{\pi abc}} e^{\left(\frac{-3x^2}{a^2}\right)} e^{\left(\frac{-3y^2}{b^2}\right)} e^{\left(\frac{-3z^2}{c^2}\right)}$$

where:

q = weld source power density

(x, y, z) = coordinates of point p in weld material

$$F = \begin{cases} F_f & \text{if point } p \text{ is in front of beam} \\ F_r & \text{if point } p \text{ is behind beam} \end{cases}$$

$$c = \begin{cases} c_f & \text{if point } p \text{ is in front of beam} \\ c_r & \text{if point } p \text{ is behind beam} \end{cases}$$

A local coordinate system is constructed which is centered at the heat source. The relative velocity vector of the heat source defines the "forward" direction, so material points that are approaching the heat source are in "front" of the beam. The beam aiming direction is used to compute the weld pool depth. The weld pool width is measured normal to the relative velocity - aiming direction plane.

***BOUNDARY_USA_SURFACE**

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

Card Format

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

<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_HYBRIDIII**

***COMPONENT_HYBRIDIII_JOINT_OPTION**

*COMPONENT

*COMPONENT_GEBOD_OPTION

Purpose: Generate a rigid body dummy based on dimensions and mass properties from the GEBOD database. The motion of the dummy is governed by equations integrated within LS-DYNA separately from the finite element model. Default joint characteristics (stiffnesses, stop angles, etc.) are set internally and should give reasonable results, however, they may be altered using the *COMPONENT_GEBOD_JOINT command. Contact between the segments of the dummy and the finite element model is defined using the *CONTACT_GEBOD command. The use of a positioning file is essential with this feature, see Appendix K for further details.

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

MALE

FEMALE

CHILD

Card Format (Card 1 of 2)

1 2 3 4 5 6 7 8

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

VARIABLE

DESCRIPTION

DID Dummy ID. A unique number must be specified.

UNITS System of units used in the finite element model.
 EQ.1: lbf*sec²/in - inch - sec
 EQ.2: kg - meter - sec
 EQ.3: kgf*sec²/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

*COMPONENT_GEBOD_JOINT_OPTION

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

The following options are available.

PELVIS

WAIST

LOWER_NECK

UPPER_NECK

LEFT_SHOULDER

RIGHT_SHOULDER

LEFT_ELBOW

RIGHT_ELBOW

LEFT_HIP

RIGHT_HIP

LEFT_KNEE

RIGHT_KNEE

LEFT_ANKLE

RIGHT_ANKLE

Card 1 - Required.

1 2 3 4 5 6 7 8

Variable	DID	LC1	LC2	LC3	SCF1	SCF2	SCF3	
Type	F	I	I	I	F	F	F	

VARIABLE

DESCRIPTION

DID Dummy ID, see *COMPONENT_GEBOD_OPTION.

LCi Load curve ID specifying the loading torque versus rotation (in radians) for the i-th degree of freedom of the joint.

*COMPONENT

VARIABLE

DESCRIPTION

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

Card 2 - Required.

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

VARIABLE

DESCRIPTION

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

NEUTi Neutral angle (degrees) of joint's i-th DOF.

Card 3 - Required.

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

VARIABLE

DESCRIPTION

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

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

*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

*COMPONENT_HYBRIDIII

Purpose: Define a HYBRID III dummy. The motion of the dummy is governed by equations integrated within LS-DYNA separately from the finite element model. The dummy interacts with the finite element structure through contact interfaces. Joint characteristics (stiffnesses, damping, friction, etc.) are set internally and should give reasonable results, however, they may be altered using the *COMPONENT_HYBRIDIII_JOINT command. Joint force and moments can be written to an ASCII file (see *DATABASE_H3OUT).

Card Format (Card 1 of 2)

	1	2	3	4	5	6	7	8
Variable	DID	SIZE	UNITS	DEFRM	VX	VY	VZ	
Type	I	F	I	I	F	F	F	
Default	none	none	none	1	0.	0.	0.	

<u>VARIABLE</u>	<u>DESCRIPTION</u>
DID	Dummy ID. A unique number must be specified.
SIZE	Size of dummy. EQ.1: 5th percentile adult EQ.2: 50th percentile adult EQ.3: 95th percentile adult Note: If negative then the best of currently available joint properties are applied.
UNITS	System of units used in the finite element model. EQ.1: lbf*sec ² /in - inch - sec EQ.2: kg - meter - sec EQ.3: kgf*sec ² /mm - mm - sec EQ.4: metric ton - mm - sec EQ.5: kg - mm - msec
DEFRM	Deformability type. EQ.1: all dummy segments entirely rigid EQ.2: deformable abdomen (low density foam, mat #57) EQ.3: deformable jacket (low density foam, mat #57) EQ.4: deformable headskin (viscoelastic, mat #6) EQ.5: deformable abdomen/jacket EQ.6: deformable jacket/headskin EQ.7: deformable abdomen/headskin EQ.8: deformable abdomen/jacket/headskin
VX,VY,VZ	Initial velocity of the dummy in the global x, y and z directions.

*COMPONENT

Card Format (Card 2 of 2)

Card 1 1 2 3 4 5 6 7 8

Variable	HX	HY	HZ	RX	RY	RZ		
Type	F	F	F	F	F	F		
Default	0.	0.	0.	0.	0.	0.		

VARIABLE

DESCRIPTION

HX,HY,HZ

Initial global x,y, and z coordinate values of the H-point .

RX,RY,RZ

Initial rotation of dummy about the H-point with respect to the global x,y, and z axes (degrees).

```

$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$ *COMPONENT_HYBRIDIII
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$ A 50th percentile adult rigid HYBRID III dummy with an ID number of 7 is defined
$ in the lbf*sec^2-inch-sec system of units. The dummy is assigned an initial
$ velocity of 616 in/sec in the negative x direction. The H-point is initially
$ situated at (x,y,z)=(38,20,0) and the dummy is rotated 18 degrees about the
$ global x-axis.
$
*COMPONENT_HYBRIDIII
$
$.>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$   did   . size   units   defm     vx     vy     vz
$     7     2     1     1    -616.    0.    0.
$   hx    hy     hz     rx     ry     rz
$   38.   20.    0.    18.    0.    0.
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$

```

*COMPONENT

*COMPONENT_HYBRIDIII_JOINT_OPTION

Purpose : Alter the joint characteristics of a HYBRID III dummy. Setting a joint parameter value to zero retains the default value set internally. Joint force and moments can be written to an ASCII file (see *DATABASE_H3OUT). Further details pertaining to the joints are found in the Hybrid III Dummies section of Appendix K.

The following options are available.

LUMBAR	RIGHT_ELBOW	RIGHT_KNEE
LOWER_NECK	LEFT_WRIST	LEFT_ANKLE
UPPER_NECK	RIGHT_WRIST	RIGHT_ANKLE
LEFT_SHOULDER	LEFT_HIP	RIBCAGE
RIGHT_SHOULDER	RIGHT_HIP	LEFT_KNEE_SLIDER
LEFT_ELBOW	LEFT_KNEE	RIGHT_KNEE_SLIDER

Card 1 - Required.

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

Card 2 - Required.

	1	2	3	4	5	6	7	8
Variable	C1	ALO1	BLO1	AHI1	BHI1	QLO1	QHI1	SCLK1
Type	F	F	F	F	F	F	F	F

*COMPONENT

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

	1	2	3	4	5	6	7	8
Variable	C2	ALO2	BLO2	AHI2	BHI2	QLO2	QHI2	SCLK2
Type	F	F	F	F	F	F	F	F

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

	1	2	3	4	5	6	7	8
Variable	C3	ALO3	BLO3	AHI3	BHI3	QLO3	QHI3	SCLK3
Type	F	F	F	F	F	F	F	F

VARIABLE

DESCRIPTION

DID	Dummy ID, see *COMPONENT_HYBRIDIII
Qi	Initial value of the joint's i-th degree of freedom. Units of degrees are defined for rotational DOF. See Appendix K for a listing of the applicable DOF.
FRIC	Friction load on the joint.
Ci	Linear viscous damping coefficient applied to the i-th DOF of the joint.
ALOi	Linear coefficient for the low regime spring of the joint's i-th DOF.
BLOi	Cubic coefficient for the low regime spring of the joint's i-th DOF.
AHi	Linear coefficient for the high regime spring of the joint's i-th DOF.
BHi	Cubic coefficient for the high regime spring of the joint's i-th DOF.
QLOi	Value at which the low regime spring definition becomes active.
QHi	Value at which the high regime spring definition becomes active.
SCLKi	Scale value applied to the stiffness of the joint's i-th DOF (default=1.0).

***CONSTRAINED**

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

- *CONSTRAINED_ADAPTIVITY**
- *CONSTRAINED_EULER_IN_EULER**
- *CONSTRAINED_EXTRA_NODES_OPTION**
- *CONSTRAINED_GENERALIZED_WELD_OPTION_{OPTION}**
- *CONSTRAINED_GLOBAL**
- *CONSTRAINED_INTERPOLATION_{OPTION}**
- *CONSTRAINED_JOINT_OPTION_{OPTION}_{OPTION}_{OPTION}**
- *CONSTRAINED_JOINT_STIFFNESS_OPTION**
- *CONSTRAINED_LAGRANGE_IN_SOLID**
- *CONSTRAINED_LINEAR_GLOBAL**
- *CONSTRAINED_LINEAR_LOCAL**
- *CONSTRAINED_NODAL_RIGID_BODY_{OPTION}_{OPTION}**
- *CONSTRAINED_NODE_SET_{OPTION}**
- *CONSTRAINED_POINTS**
- *CONSTRAINED_RIGID_BODIES**
- *CONSTRAINED_RIGID_BODY_STOPPERS**
- *CONSTRAINED_RIVET_{OPTION}**
- *CONSTRAINED_SHELL_TO_SOLID**
- *CONSTRAINED_SPOTWELD_{OPTION}_{OPTION}**
- *CONSTRAINED_TIE-BREAK**
- *CONSTRAINED_TIED_NODES_FAILURE**

*CONSTRAINED

*CONSTRAINED_ADAPTIVITY

Purpose: Define an adaptive constraint which constrains a node to the midpoint along an edge of a shell element. This keyword is also created by LS-DYNA during an adaptive calculation. This option applies to shell elements.

Card Format

	1	2	3	4	5	6	7	8
Variable	SN!	MN1	MN2					
Type	I	I	I					
Default	none	none	none					

<u>VARIABLE</u>	<u>DESCRIPTION</u>
SN	Slave node. This is the node constrained at the midpoint of an edge of a shell element.
MN1	One node along the edge of the shell element.
MN2	The second node along the edge.

***CONSTRAINED_EULER_IN_EULER**

Purpose: This command defines the coupling interaction between materials in two overlapping, geometrically identical, multi-material Eulerian mesh sets. The command allows a frictionless contact between two or more different Eulerian materials.

Card Format

	1	2	3	4	5	6	7	8
Variable	PSIDSLV	PSIDMST	PFAC					
Type	I	I	F					
Default	0	0	0.1					

VARIABLE

DESCRIPTION

PSIDSLV	Part set ID of the 1 st ALE or Eulerian set of mesh(es) (slave).
PSIDMST	Part set ID of the 2 nd ALE or Eulerian set of mesh(es) (master).
PFAC	A penalty factor for the coupling interaction between the two PSIDs.

Remarks:

1. Contact pressure is built up in two overlapping Eulerian elements if their combined material fill fraction exceeds 1.0 (penalty formulation).
2. This feature needs to be combined with *MAT_VACUUM (element formulation 11), or with initially voided elements (element formulation 12).

Example:

Consider an ALE/Eulerian multi-material model (ELFORM=11) consisting of:

- PID 1 = *MAT_NULL (material 1)
- PID 2 = *MAT_VACUUM \Rightarrow PID 1 is merged at its boundary to PID 2.
- PID 3 = *MAT_NULL (material 3)
- PID 4 = *MAT_VACUUM \Rightarrow PID 3 is merged at its boundary to PID 4.

The mesh set containing PID 1 & 2 intersects or overlaps with the mesh set containing PID 3 & 4. PID 1 is given an initial velocity in the positive x direction. This will cause material 1 to contact material 3 (note that materials 2 & 4 are void). The interaction between materials 1 & 3 is possible by defining this coupling command. In this case material 1 can flow within the mesh region of PID 1 & 2 only, and material 3 can flow within the mesh region of PID 3 & 4 only.

*CONSTRAINED

```
$... | .....1.... | .....2.... | .....3.... | .....4.... | .....5.... | .....6.... | .....7.... | ...8
*ALE_MULTI-MATERIAL_GROUP
$      SID      SIDYTP
      1          1
      2          1
      3          1
      4          1
*CONSTRAINED_EULER_IN_EULER
$      PSID1      PSID2      PENAL
      11          12          0.1
*SET_PART_LIST
      11
      1          2
*SET_PART_LIST
      12
      3          4
$... | .....1.... | .....2.... | .....3.... | .....4.... | .....5.... | .....6.... | .....7.... | ...8
```

***CONSTRAINED_EXTRA_NODES_OPTION**

Available options include:

NODE**SET**

Purpose: Define extra nodes for rigid body.

Card 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_GENERALIZED_WELD_OPTION_{OPTION}**

Then the following options are available:

SPOT

FILLET

BUTT

CROSS_FILLET

COMBINED

To defined an ID for the weld use the option:

ID

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

ID Card - Required if the option `_ID` is active on the keyword card.

Card 1 1 2 3 4 5 6 7 8

Variable	WID								
Type	I								
Default	0								

*CONSTRAINED

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						

<u>VARIABLE</u>	<u>DESCRIPTION</u>
WID	Optional weld ID.
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.
FILTER	Number of force vectors saved for filtering. This option can eliminate spurious failures due to numerical force spikes; however, memory requirements are significant since 6 force components are stored with each vector. LE.1: no filtering EQ.n: simple average of force components divided by n or the maximum number of force vectors that are stored for the time window option below.
WINDOW	Time window for filtering. This option requires the specification of the maximum number of steps which can occur within the filtering time window. If the time step decreases too far, then the filtering time window will be ignored and the simple average is used. EQ.0: time window is not used
NPR	NFW, number of individual nodal pairs in the cross fillet or combined general weld.
NPRT	Print option in file RBDOUT. EQ.0: default from the control card, *CONTROL_OUTPUT, is used, see variable name IPRTF. EQ.1: data is printed EQ.2: data is not printed

Additional Card required for the CONSTRAINED_GENERALIZED_WELD_SPOT option:

Card 2	1	2	3	4	5	6	7	8
Variable	TFAIL	EPSF	SN	SS	N	M		
Type	F	F	F	F	F	F		

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

Remarks:

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

Brittle failure of the spotwelds occurs when:

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

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

*CONSTRAINED

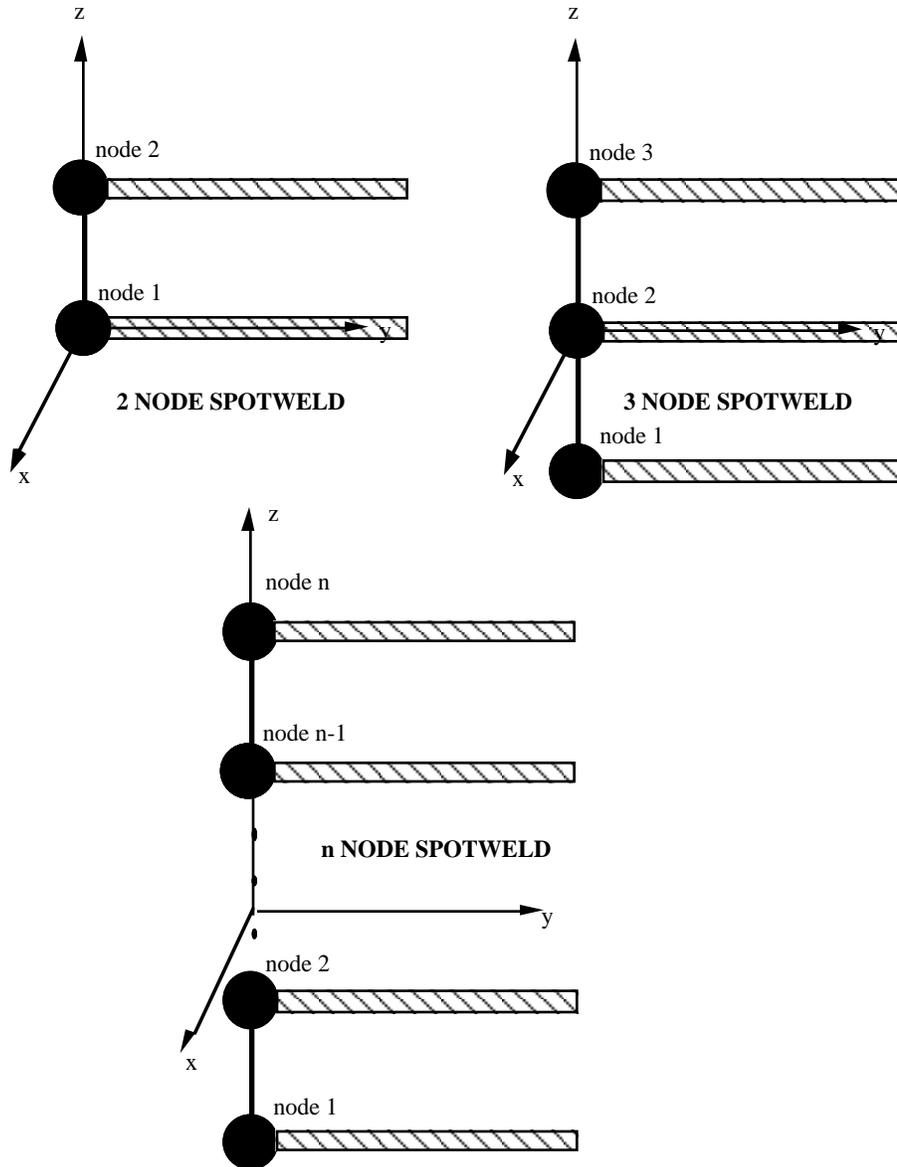


Figure 5.1. Nodal ordering and orientation of the local coordinate system is important for determining spotweld failure.

Additional Card required for the FILLET option:

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

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

Remarks:

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

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

where

- σ_n = normal stress
- τ_n = shear stress in direction of weld (local y)
- τ_t = shear stress normal to weld (local x)
- σ_f = failure stress
- β = failure parameter

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

*CONSTRAINED

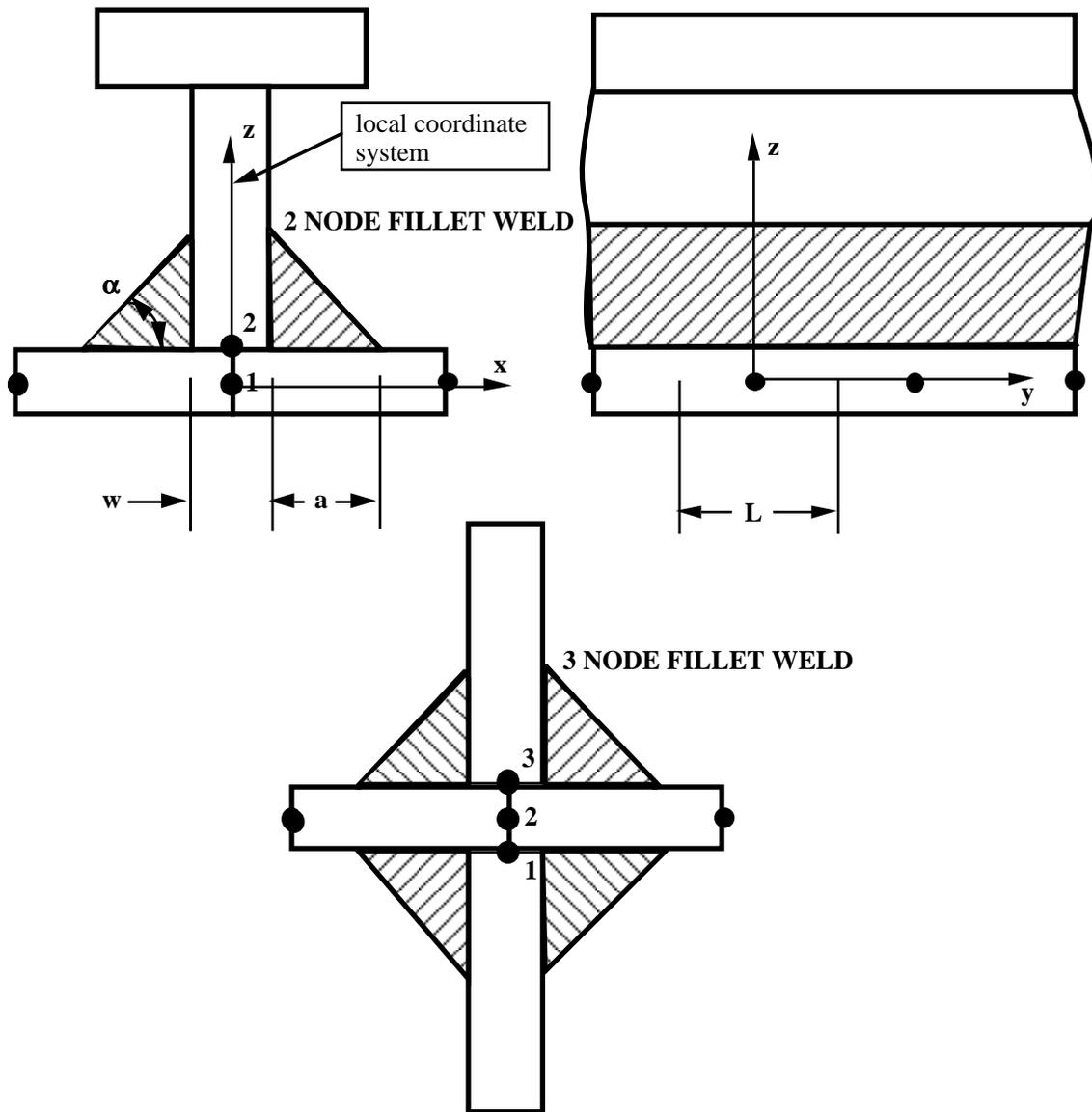


Figure 5.2. Nodal ordering and orientation of the local coordinate system is shown for fillet weld failure. The angle is defined in degrees.

Additional Card required for the BUTT option:

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

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

Remarks:

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

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

where

- σ_n = normal stress
- τ_n = shear stress in direction of weld (local y)
- τ_t = shear stress normal to weld (local z)
- σ_f = failure stress
- β = failure parameter

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

*CONSTRAINED

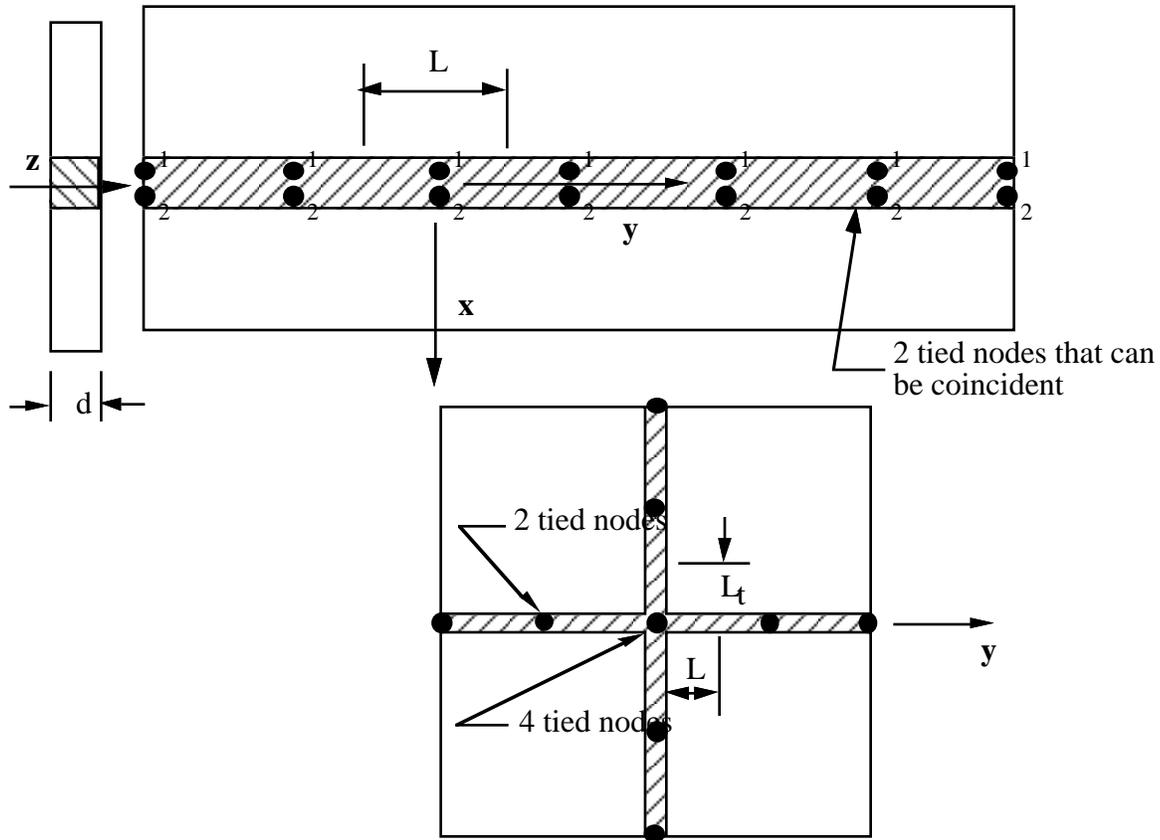


Figure 5.3. Orientation of the local coordinate system and nodal ordering is shown for butt weld failure.

*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					

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

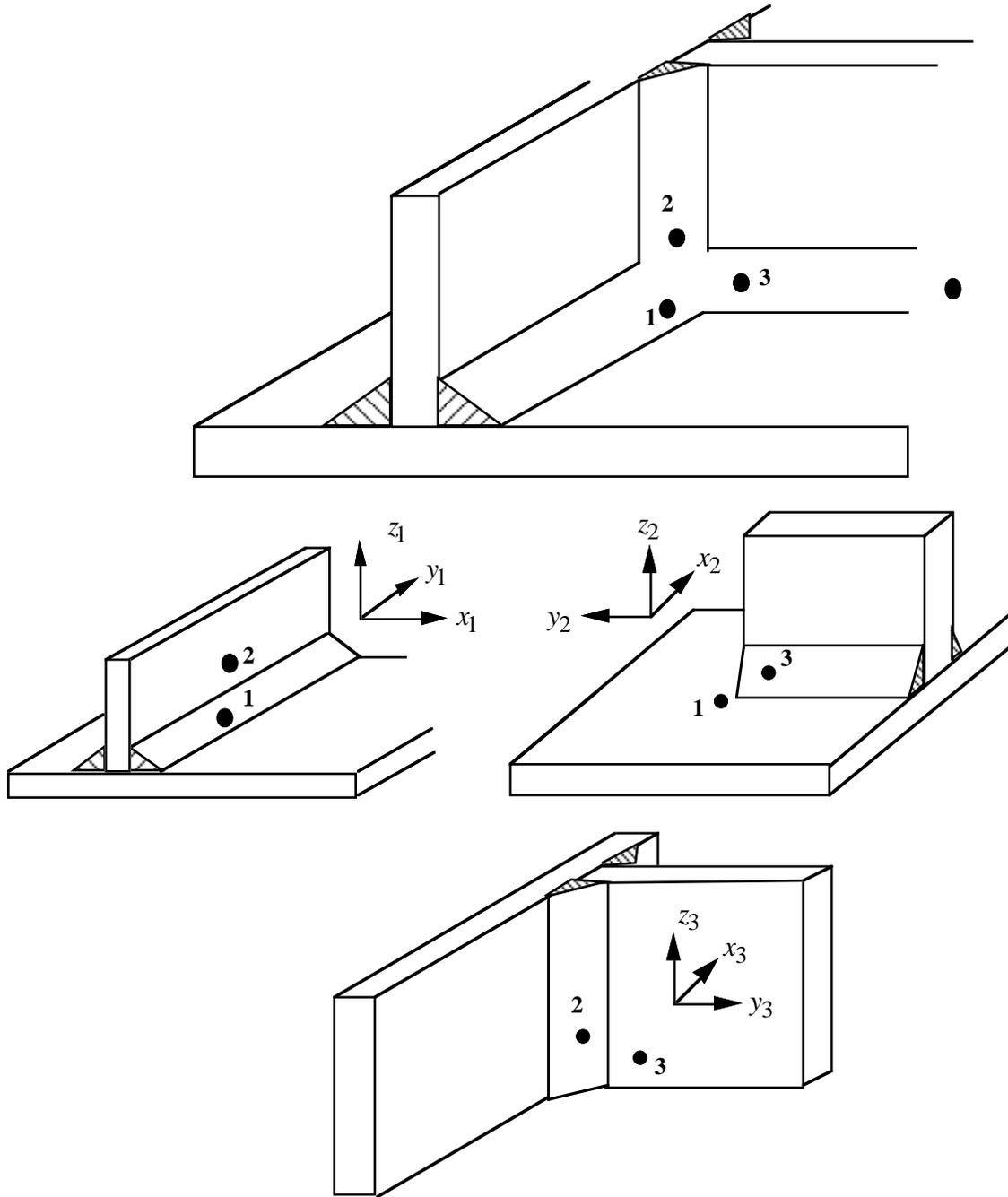


Figure 5.4. A simple cross fillet weld illustrates the required input. Here NFW=3 with nodal pairs (A=2, B=1), (A=3, B=1), and (A=3, B=2). The local coordinate axes are shown. These axes are fixed in the rigid body and are referenced to the local rigid body coordinate system which tracks the rigid body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

*CONSTRAINED

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

Card 2 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, ϵ_{fail}^p defines ductile failure.
SIGY	σ_f , stress at failure for brittle failure.
BETA	β , failure parameter for brittle failure.
L	L, length of fillet/butt weld (see Figure 5.2 and 5.3).
W	w, width of flange (see Figure 5.2).
A	a, width of fillet weld (see Figure 5.2).
ALPHA	α , weld angle (see Figure 5.2) in degrees.
NODEA	Node ID, A, in weld pair (CROSS or COMBINED option only).
NODEB	Node ID, B, in weld pair (CROSS or COMBINED option only).
NCID	Local coordinate system ID (CROSS or COMBINED option only).
WTYP	Weld pair type (GENERAL option only). See Figure 5.5. EQ.0: fillet weld EQ.1: butt weld

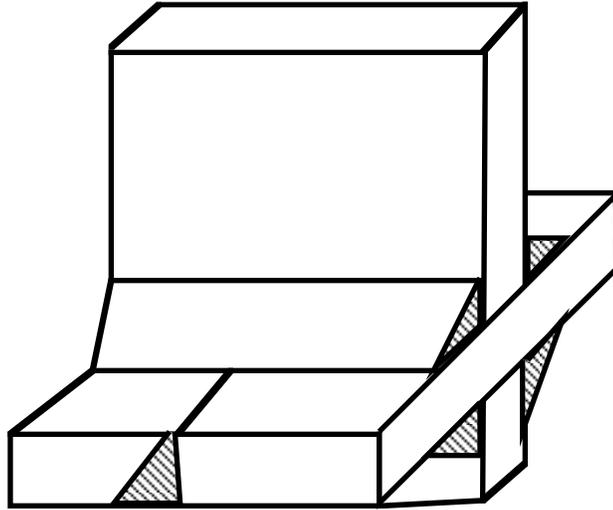


Figure 5.5. A combined weld is a mixture of fillet and butt welds.

*CONSTRAINED

*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		

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

Remarks:

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

*CONSTRAINED

*CONSTRAINED_INTERPOLATION_{OPTION}

The following option is available:

<BLANK>

LOCAL

Purpose: Define an interpolation constraint. With this constraint type, the motion of a single dependent node is interpolated from the motion of a set of independent nodes. This option is useful for the load redistribution of a load, which can be either a translational force or moment, applied to the dependent node to the surrounding independent nodes, and it can also be used to model shell-brick and beam-brick interfaces. The mass and rotary inertia of the dependent nodal point is also redistributed. This constraint is applied in the global coordinate system unless the option LOCAL is active. One *CONSTRAINED_INTERPOLATION card is required for each constraint definition. The input list of independent nodes is terminated when the next "*" card is found.

Card Format

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

Cards 2, 3, 4, etc. Define one card per independent node. If the option LOCAL is active, the define two cards per independent node. Input is terminated when a "*" card is found.

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

Define the second card if and only if the option LOCAL is active

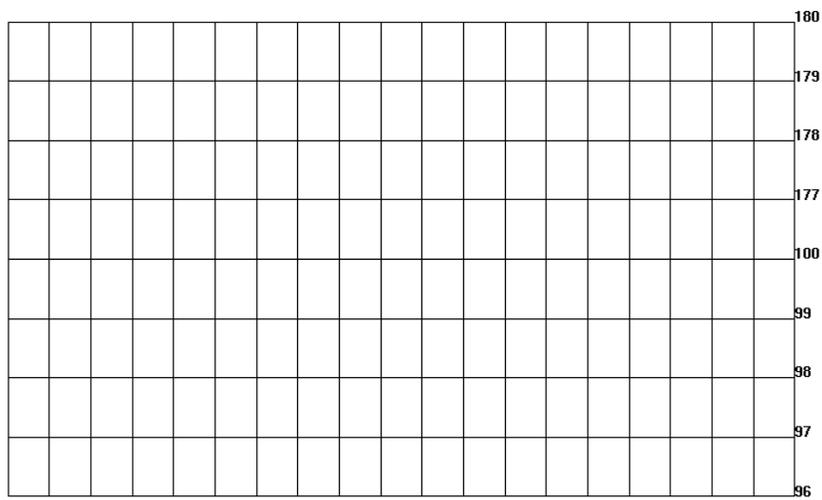
	1	2	3	4	5	6	7	8
Variable	CIDI							
Type	I							
Default	0							

<u>VARIABLE</u>	<u>DESCRIPTION</u>
ICID	Interpolation constraint ID.
DNID	Dependent node ID. This node should not be a member of a rigid body, or elsewhere constrained in the input.
DDOF	Dependent degrees-of-freedom. The list of dependent degrees-of-freedom consists of a number with up to six digits, with each digit representing a degree of freedom. For example, the value 1356 indicates that degrees of freedom 1, 3, 5, and 6 are controlled by the constraint. The default is 123456. Digit: degree of freedom ID's: EQ.1:x EQ.2:y EQ.3:z EQ.4:rotation about x axis EQ.5:rotation about y axis EQ.6:rotation about z axis
CIDD	Local coordinate system ID if LOCAL option is active. If blank the global coordinate system is assumed.
INID	Independent node ID.
IDOF	Independent degrees-of-freedom using the same form as for the dependent degrees-of-freedom, DDOF, above.
TWGHTX	Weighting factor for node INID with active degrees-of-freedom IDOF. This weight scales the x-translational component. It is normally sufficient to define only TWGHTX even if its degree-of-freedom is inactive since the other factors are set equal to this input value as the default. There is no requirement on the values that are chosen as the weighting factors, i.e., that they sum to unity. The default value for the weighting factor is unity.
TWGHTY	Weighting factor for node INID with active degrees-of-freedom IDOF. This weight scales the y-translational component.

*CONSTRAINED

97	1
98	1
99	1
177	1
178	1
179	1
180	1

⊘
*
⊘



***CONSTRAINED_JOINT_OPTION_{OPTION}_{OPTION}_{OPTION}**

Available forms include (one is mandatory):

**CONSTRAINED_JOINT_SPHERICAL
CONSTRAINED_JOINT_REVOLUTE
CONSTRAINED_JOINT_CYLINDRICAL
CONSTRAINED_JOINT_PLANAR
CONSTRAINED_JOINT_UNIVERSAL
CONSTRAINED_JOINT_TRANSLATIONAL
CONSTRAINED_JOINT_LOCKING
CONSTRAINED_JOINT_TRANSLATIONAL_MOTOR
CONSTRAINED_JOINT_ROTATIONAL_MOTOR
CONSTRAINED_JOINT_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 use the option:

LOCAL

to define a joint ID and heading the following option is available:

ID

and to define failure use:

FAILURE

The ordering of the bracketed options is arbitrary.

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

Card Format:

Card 1 is required for all joint types.

Card 2 is required for joint types: MOTOR, GEARS, RACK_AND_PINION, PULLEY, and SCREW

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

In the first seven joint types above excepting the Universal joint, the nodal points within the nodal pairs (1,2), (3,4), and (5,6) (see Figure 5.6) should coincide in the initial configuration, and the nodal pairs should be as far apart as possible to obtain the best behavior. For the Universal Joint the nodes within the nodal pair (3,4) do not coincide, but the lines drawn between nodes (1,3) and (2,4) must be perpendicular.

For the Gear joint the nodes within the nodal pair (1,2) must not coincide.

The geometry of joints is defined in Figure 5.6. When the penalty method is used (see *CONTROL_RIGID), at each time step, the relative penalty stiffness is multiplied by a function dependent on the step size to give the maximum stiffness that will not destroy the stability of the solution. Instabilities can result in the explicit time integration scheme if the penalty stiffness is too large. If instabilities occur, the recommended way to eliminate these problems is to decrease the time step or reduce the scale factor on the penalties..

*CONSTRAINED

<u>VARIABLE</u>	<u>DESCRIPTION</u>
N6	Node 6, in rigid body B. Define for joint types TRANSLATIONAL, LOCKING, ROTATIONAL_MOTOR, CONSTANT_VELOCITY, GEARS, RACK_AND_PINION, PULLEY, and SCREW
RPS	Relative penalty stiffness (default = 1.0).
DAMP	Damping scale factor on default damping value. (Revolute and Spherical Joints): EQ.0.0: default is set to 1.0, LE.0.01 and GT.0.0: no damping is used.

Card 2. Required for joint types MOTOR, GEARS, RACK_AND_PINION, PULLEY, and SCREW only.

Card 1 1

Variable	PARAM	LCID	TYPE	R1				
Type	F	I	I	F				
Default	none							

<u>VARIABLE</u>	<u>DESCRIPTION</u>
PARAM	Parameter which a function of joint type. Leave blank for MOTORS. Gears: define R_2 / R_1 Rack and Pinion: define h Pulley: define R_2 / R_1 Screw: define \dot{x} / ω
LCID	Define load curve ID for MOTOR joints.

*CONSTRAINED

<u>VARIABLE</u>	<u>DESCRIPTION</u>
TYPE	Define integer flag for MOTOR joints as follows: EQ.0: translational/rotational velocity EQ.1: translational/rotational acceleration EQ.2: translational/rotational displacement
R1	Radius, R_1 , for the gear and pulley joint type. If left undefined, nodal points 5 and 6 are assumed to be on the outer radius.

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

Card 1 1 2 3 4 5 6 7 8

Variable	RAID	LST						
Type	I	I						
Default	0	0						

<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

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

Card 1 1 2 3 4 5 6 7 8

Variable	CID	TFAIL	COUPL					
Type	I	F	F					
Default	0	0	0.					

Card 2

Variable	NRR	NRS	NRT	MRR	MSS	MTT		
Type	F	F	F	F	F	F		
Default	0	0	0	0	0	0		

<u>VARIABLE</u>	<u>DESCRIPTION</u>
CID	Coordinate ID for resultants in the failure criteria. If zero, the global coordinate system is used.
TFAIL	Time for joint failure. If zero, joint never fails.
COUPL	Coupling between the force and moment failure criteria. If COUPL is less than or equal to zero, the failure criteria is identical to the spotwelds. When COUPL is greater than zero, the force and moment results are considered independently. See the remark below.
NXX	Axial force resultant N_{xx_F} at failure. If zero, failure due to this component is not considered.
NYF	Force resultant N_{yy_F} at failure. If zero, failure due to this component is not considered.
NZF	Force resultant N_{zz_F} at failure. If zero, failure due to this component is not considered.
MXF	Torsional moment resultant M_{xx_F} at failure. If zero, failure due to this component is not considered.
MYF	Moment resultant M_{yy_F} at failure. If zero, failure due to this component is not considered.
MZF	Moment resultant M_{zz_F} at failure. If zero, failure due to this component is not considered.

Remarks:

When COUPL is less than or equal to zero, the failure criteria is

$$\left(\frac{\max(N_{xx}, 0)}{N_{xx_F}}\right)^2 + \left(\frac{N_{yy}}{N_{yy_F}}\right)^2 + \left(\frac{N_{zz}}{N_{zz_F}}\right)^2 + \left(\frac{M_{xx}}{M_{xx_F}}\right)^2 + \left(\frac{M_{yy}}{M_{yy_F}}\right)^2 + \left(\frac{M_{zz}}{M_{zz_F}}\right)^2 - 1 = 0$$

*CONSTRAINED

otherwise it is

$$\left(\frac{\max(N_{xx}, 0)}{N_{xx_F}}\right)^2 + \left(\frac{N_{yy}}{N_{yy_F}}\right)^2 + \left(\frac{N_{zz}}{N_{zz_F}}\right)^2 - 1 = 0 \text{ and } \left(\frac{M_{xx}}{M_{xx_F}}\right)^2 + \left(\frac{M_{yy}}{M_{yy_F}}\right)^2 + \left(\frac{M_{zz}}{M_{zz_F}}\right)^2 - 1 = 0.$$

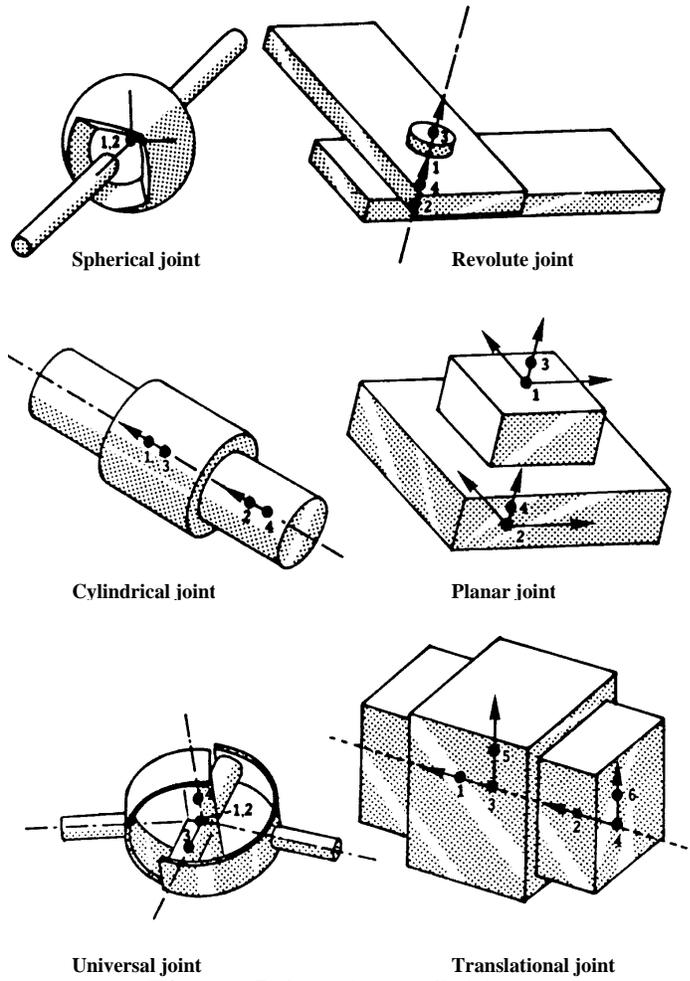


Figure 5.6. Joint definitions 1-6

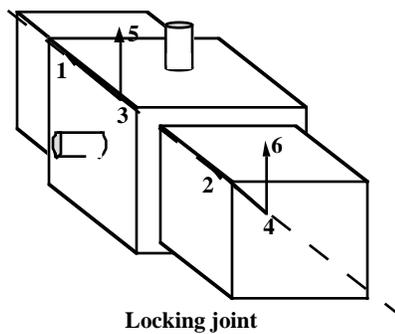


Figure 5.7. Locking joint.

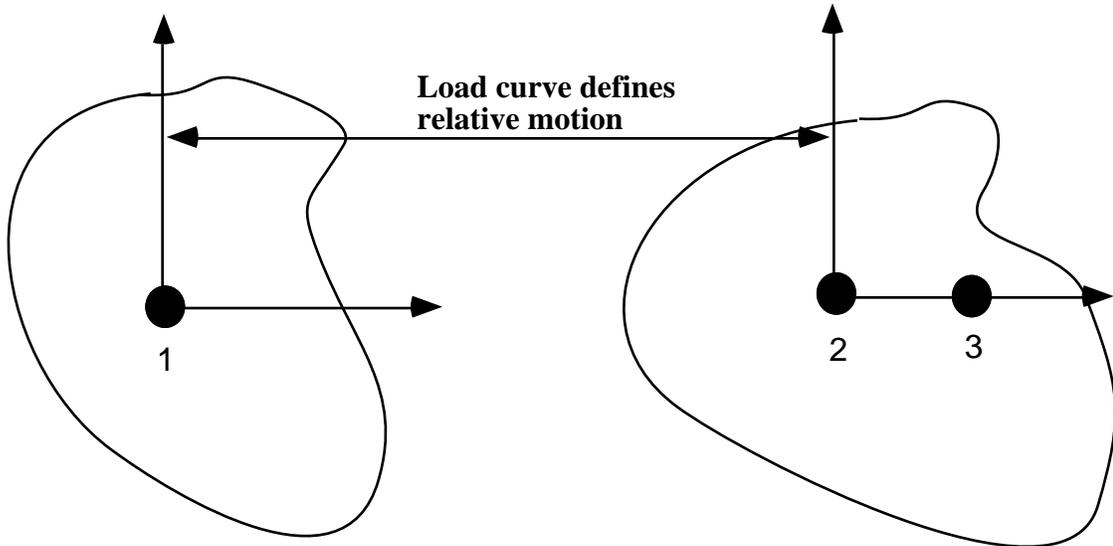


Figure 5.8. Translational motor joint. This joint can be used in combination with the translational or the cylindrical joint.

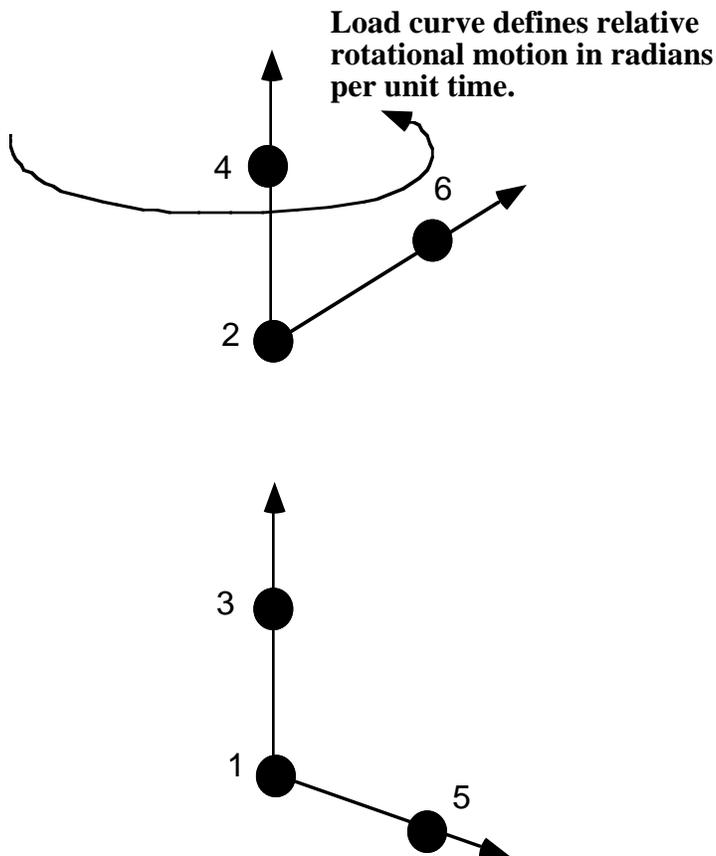


Figure 5.9. Rotational motor joint. This joint can be used in combination with other joints such as the revolute or cylindrical joints.

*CONSTRAINED

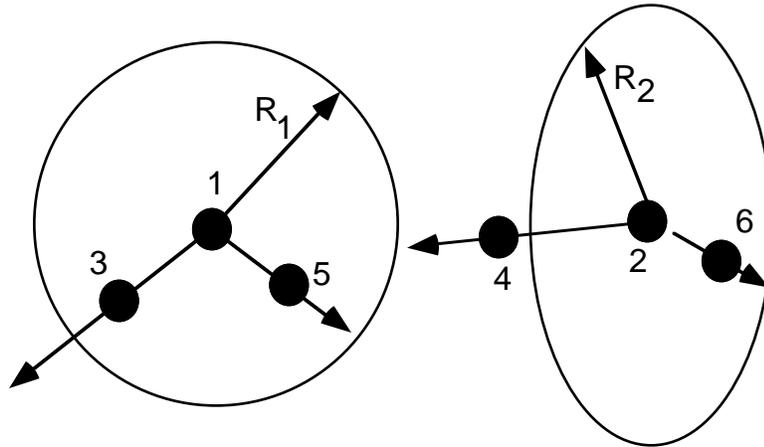


Figure 5.10. Gear joint. Nodal pairs (1,3) and (2,4) define axes that are orthogonal to the gears. Nodal pairs (1,5) and (2,6) define vectors in the plane of the gears. The ratio $\frac{R_2}{R_1}$ is specified.

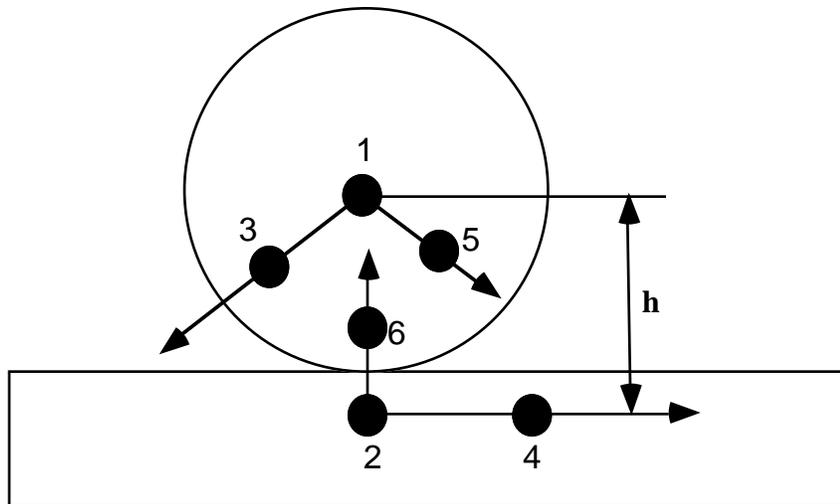


Figure 5.11. Rack and pinion joint. Nodal pair (1,3) defines a vector that is orthogonal to the plane of the gear. Nodal pair (1,5) is a vector in the plane of the gear. Nodal pair (2,4) defines the direction of travel for the second body. The value h is specified.

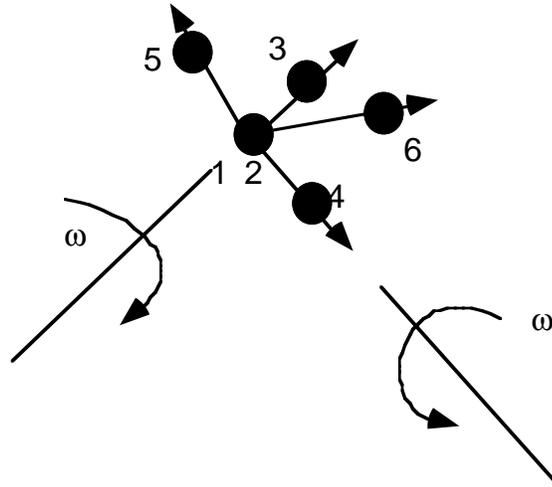


Figure 5.12. Constant velocity joint. Nodal pairs (1,3) and (2,4) define an axes for the constant angular velocity, and nodal pairs (1,5) are orthogonal vectors. Here nodal points 1 and 2 must be coincident.

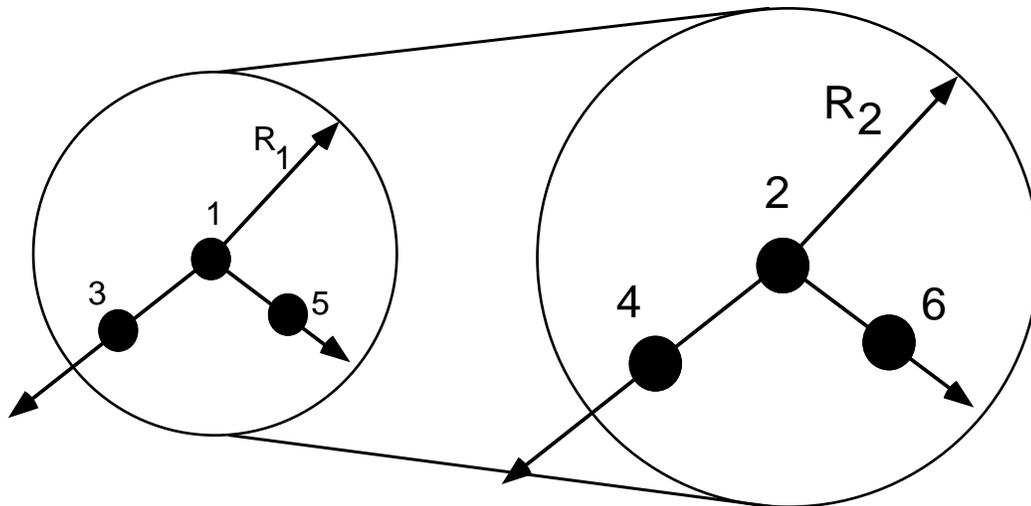


Figure 5.13. Pulley joint. Nodal pairs (1,3) and (2,4) define axes that are orthogonal to the pulleys. Nodal pairs (1,5) and (2,6) define vectors in the plane of the pulleys. The ratio $\frac{R_2}{R_1}$ is specified.

*CONSTRAINED

*CONSTRAINED_JOINT_STIFFNESS_OPTION

Options include:

FLEXION-TORSION

GENERALIZED

TRANSLATIONAL

Purpose: Define optional rotational and translational 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 Formats:

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 6

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

<u>VARIABLE</u>	<u>DESCRIPTION</u>
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. <i>For the translational stiffness the local coordinate system must be defined by nodal points, *DEFINE_COORDINATE_NODES, since the first nodal point in each coordinate system is used to track the motion.</i>

*CONSTRAINED

VARIABLE	DESCRIPTION
CIDB	Coordinate ID for rigid body B. If zero, the coordinate ID for rigid body A is used, see *DEFINE_COORDINATE_OPTION. <i>For the translational stiffness the local coordinate system must be defined by nodal points, *DEFINE_COORDINATE_NODES, since the first nodal point in each coordinate system is used to track the motion.</i>
JID	Joint ID for the joint reaction forces. If zero, tables can't be used in place of load curves for defining the frictional moments.

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

Card 2 1 2 3 4 5 6

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

Card 3 1 2 3 4

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

Card 4 1 2 3

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

*CONSTRAINED

<u>VARIABLE</u>	<u>DESCRIPTION</u>
LCIDAL	Load curve ID for α -moment versus rotation in radians. See Figure 5.15 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 γ versus a scale factor which scales the bending moment due to the α rotation. This load curve should be defined in the interval $-\pi \leq \gamma \leq \pi$. If zero the scale factor defaults to 1.0. See *DEFINE_CURVE.
LCIDBT	Load curve ID for β -torsion moment versus twist in radians. If zero the applied twist is set to zero. See *DEFINE_CURVE.
DLCIDAL	Load curve ID for α -damping moment versus rate of rotation in radians per unit time. If zero, damping is not considered. See *DEFINE_CURVE.
DLCIDG	Load curve ID for γ -damping scale factor versus rate of rotation in radians per unit time. This scale factor scales the α -damping moment. If zero, the scale factor defaults to one. See *DEFINE_CURVE.
DLCIDBT	Load curve ID for β -damping torque versus rate of twist. If zero damping is not considered. See *DEFINE_CURVE.
ESAL	Elastic stiffness per unit radian for friction and stop angles for α rotation, see Figure 5.15. If zero, friction and stop angles are inactive for α rotation..
FMAL	Frictional moment limiting value for α rotation. If zero, friction is inactive for α rotation. This option may also be thought of as an elastic-plastic spring. If a negative value is input then the absolute value is taken as the load curve or table ID defining the yield moment versus α rotation, see Figure 5.15. A table permits the moment to also be a function of the joint reaction force and requires the specification of JID on card 1.
ESBT	Elastic stiffness per unit radian for friction and stop angles for β twist, see Figure 5.15. If zero, friction and stop angles are inactive for β twist.
FMBT	Frictional moment limiting value for β twist. If zero, friction is inactive for β twist. This option may also be thought of as an elastic-plastic spring. If a negative value is input then the absolute value is taken as the load curve or table ID defining the yield moment versus β rotation, see Figure 5.15 A table permits the moment to also be a function of the joint reaction force and requires the specification of JID on card 1.
SAAL	Stop angle in degrees for α rotation where $0 \leq \alpha \leq \pi$. Ignored if zero.
NSABT	Stop angle in degrees for negative β rotation. Ignored if zero.
PSABT	Stop angle in degrees for positive β rotation. Ignored if zero.

Remarks:

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

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

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

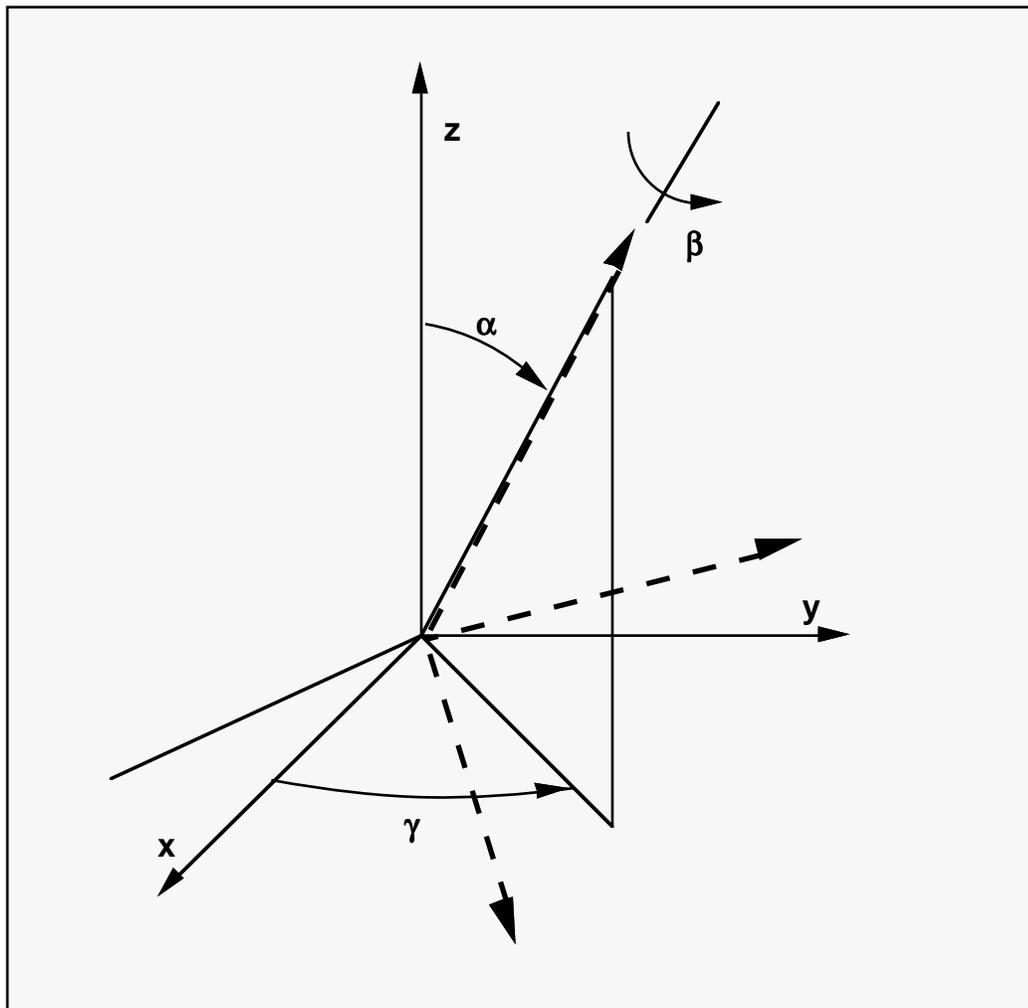


Figure 5.15 Flexion-torsion joint angles. If the initial positions of the local coordinate axes of the two rigid bodies connected by the joint do not coincide, the angles, α and γ , are initialized and torques will develop instantaneously based on the defined load curves. The angle β is also initialized but no torque will develop about the local axis on which β is measured. Rather, β will be measured relative to the computed offset.

*CONSTRAINED

Card 2-4 - 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		

Card 3

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

Card 4

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

VARIABLE

DESCRIPTION

LCIDPH	Load curve ID for ϕ -moment versus rotation in radians. See Figure 5.16. If zero, the applied moment is set to 0.0. See *DEFINE_CURVE.
LCIDT	Load curve ID for θ -moment versus rotation in radians. If zero, the applied moment is set to 0.0. See *DEFINE_CURVE.
LCIDPS	Load curve ID for ψ -moment versus rotation in radians. If zero, the applied moment is set to 0.0. See *DEFINE_CURVE.
DLCIDPH	Load curve ID for ϕ -damping moment versus rate of rotation in radians per unit time. If zero, damping is not considered. See *DEFINE_CURVE.

VARIABLE	DESCRIPTION
DLCIDT	Load curve ID for θ -damping moment versus rate of rotation in radians per unit time. If zero, damping is not considered. See *DEFINE_CURVE.
DLCIDPS	Load curve ID for ψ -damping torque versus rate of rotation in radians per unit time. If zero, damping is not considered. See *DEFINE_CURVE.
ESPH	Elastic stiffness per unit radian for friction and stop angles for ϕ rotation. See Figure 5.17. If zero, friction and stop angles are inactive for ϕ rotation.
FMPH	Frictional moment limiting value for ϕ rotation. If zero, friction is inactive for ϕ rotation. This option may also be thought of as an elastic-plastic spring. If a negative value is input then the absolute value is taken as the load curve or table ID defining the yield moment versus ϕ rotation. See Figure 5.17. A table permits the moment to also be a function of the joint reaction force and requires the specification of JID on card 1.
EST	Elastic stiffness per unit radian for friction and stop angles for θ rotation. See Figure 5.17. If zero, friction and stop angles are inactive for θ rotation.
FMT	Frictional moment limiting value for θ rotation. If zero, friction is inactive for θ rotation. This option may also be thought of as an elastic-plastic spring. If a negative value is input then the absolute value is taken as the load curve or table ID defining the yield moment versus θ rotation. See Figure 5.17. A table permits the moment to also be a function of the joint reaction force and requires the specification of JID on card 1.
ESPS	Elastic stiffness per unit radian for friction and stop angles for ψ rotation. See Figure 5.17. If zero, friction and stop angles are inactive for ψ rotation..
FMPS	Frictional moment limiting value for ψ rotation. If zero, friction is inactive for ψ rotation. This option may also be thought of as an elastic-plastic spring. If a negative value is input then the absolute value is taken as the load curve or table ID defining the yield moment versus ψ rotation. See Figure 5.17. A table permits the moment to also be a function of the joint reaction force and requires the specification of JID on card 1.
NSAPH	Stop angle in degrees for negative ϕ rotation. Ignored if zero.
PSAPH	Stop angle in degrees for positive ϕ rotation. Ignored if zero.
NSAT	Stop angle in degrees for negative θ rotation. Ignored if zero.
PSAT	Stop angle in degrees for positive θ rotation. Ignored if zero.
NSAPS	Stop angle in degrees for negative ψ rotation. Ignored if zero.
PSAPS	Stop angle in degrees for positive ψ rotation. Ignored if zero.

*CONSTRAINED

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.

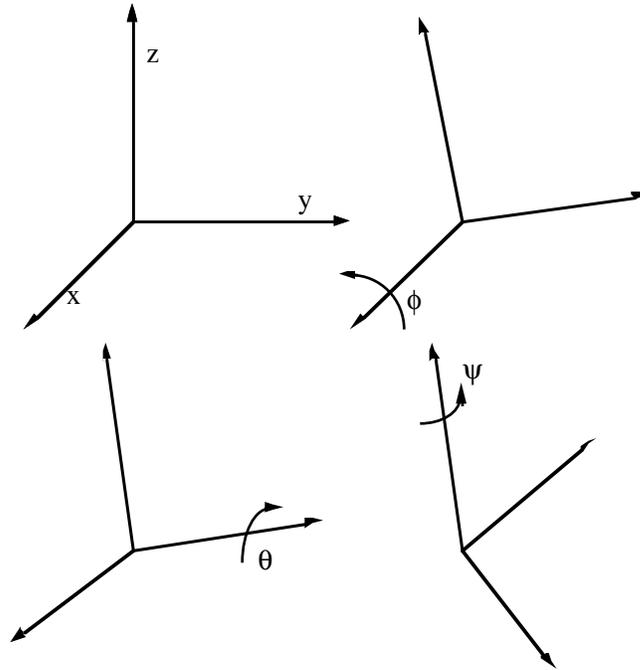


Figure 5.16. Definition of angles for the generalized joint stiffness. The magnitude of the angular rotations are limited by the stop angles defined on Card 4. If the initial local coordinate axes do not coincide, the angles, ϕ , θ , and ψ , will be initialized and torques will develop instantaneously based on the defined load curves.

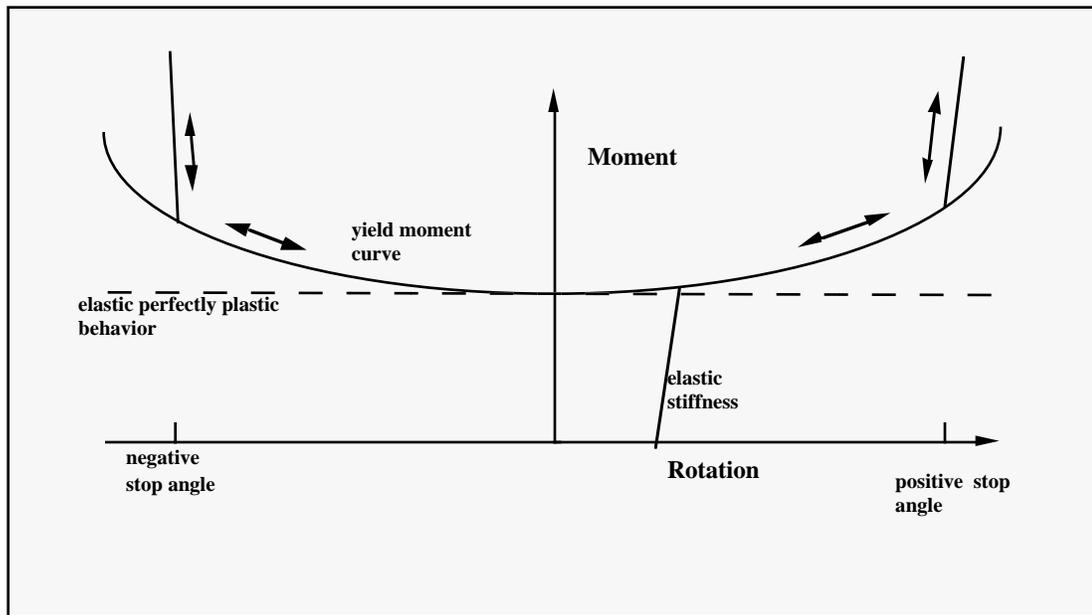


Figure 5.17. Frictional behavior is modeled by a plasticity model. Elastic behavior is obtained once the stop angles are reached. The same elastic stiffness is used to simulate sticking situations.

*CONSTRAINED

Card 2- 4 - Required for TRANSLATIONAL stiffness.

Card 2 1 2 3 4 5 6

Variable	LCIDX	LCIDY	LCIDZ	DLCIDX	DLCIDY	DLCIDZ		
Type	I	I	I	I	I	I		
Default	none	none	none	none	none	none		

Card 3

Variable	ESX	FFX	ESY	FFY	ESZ	FFZ		
Type	F	F	F	F	F	F		
Default	0.0	0.0	0.0	0.0	0.0	0.0		

Card 4

Variable	NSDX	PSDX	NSDY	PSDY	NSDZ	PSDZ		
Type	F	F	F	F	F	F		
Default	not used							

VARIABLE

DESCRIPTION

LCIDX Load curve ID for x–force versus x-translational relative displacement between the origins of CIDA and CIDB based on the x-direction of CIDB. If zero, the applied force is set to 0.0. See *DEFINE_CURVE. See Figure 5.18.

LCIDY Load curve ID for y–force versus y-translational relative displacement between the origins of CIDA and CIDB based on the y-direction of CIDB. If zero, the applied force is set to 0.0. See *DEFINE_CURVE.

VARIABLE	DESCRIPTION
LCIDZ	Load curve ID for z–force versus z-translational relative displacement between the origins of CIDA and CIDB based on the z-direction of CIDB. If zero, the applied force is set to 0.0. See *DEFINE_CURVE.
DLCIDX	Load curve ID for x–damping force versus rate of x-translational displacement per unit time between the origins of CIDA and CIDB based on the x-direction of CIDB. If zero, damping is not considered. See *DEFINE_CURVE.
DLCIDY	Load curve ID for y–damping force versus rate of y-translational displacement per unit time between the origins of CIDA and CIDB based on the y-direction of CIDB. If zero, damping is not considered. See *DEFINE_CURVE.
DLCIDZ	Load curve ID for z–damping force versus rate of z-translational displacement per unit time between the origins of CIDA and CIDB based on the z-direction of CIDB. If zero, damping is not considered. See *DEFINE_CURVE.
ESX	Elastic stiffness for friction and stop displacement for x-translation. If zero, friction and stop angles are inactive for x-translation.
FFX	Frictional force limiting value for x-translation. If zero, friction is inactive for x-translation. This option may also be thought of as an elastic-plastic spring. If a negative value is input then the absolute value is taken as the load curve ID defining the yield force versus x-translation.
ESY	Elastic stiffness for friction and stop displacement for y-translation. If zero, friction and stop angles are inactive for y-translation.
FFY	Frictional force limiting value for y-translation. If zero, friction is inactive for y-translation. This option may also be thought of as an elastic-plastic spring. If a negative value is input then the absolute value is taken as the load curve ID defining the yield force versus y-translation.
ESZ	Elastic stiffness for friction and stop displacement for z-translation. If zero, friction and stop angles are inactive for z-translation..
FMZ	Frictional force limiting value for z-translation. If zero, friction is inactive for z-translation. This option may also be thought of as an elastic-plastic spring. If a negative value is input then the absolute value is taken as the load curve ID defining the yield force versus z-translation.
NSDX	Stop displacement for negative x-translation. Ignored if zero.
PSDX	Stop displacement for positive x-translation. Ignored if zero.
NSDY	Stop displacement for negative y-translation. Ignored if zero.

*CONSTRAINED

VARIABLE	DESCRIPTION
PSDY	Stop displacement for positive y-translation. Ignored if zero.
NSDZ	Stop displacement for negative z-translation. Ignored if zero.
PSDZ	Stop displacement for positive z-translation. Ignored if zero.

Remarks:

After the stop displacements are reached the force increase linearly to resist further translational motion using the stiffness values on Card 3. Reasonable stiffness values must be chosen. If the stiffness values are too low or zero, the stop will be violated.

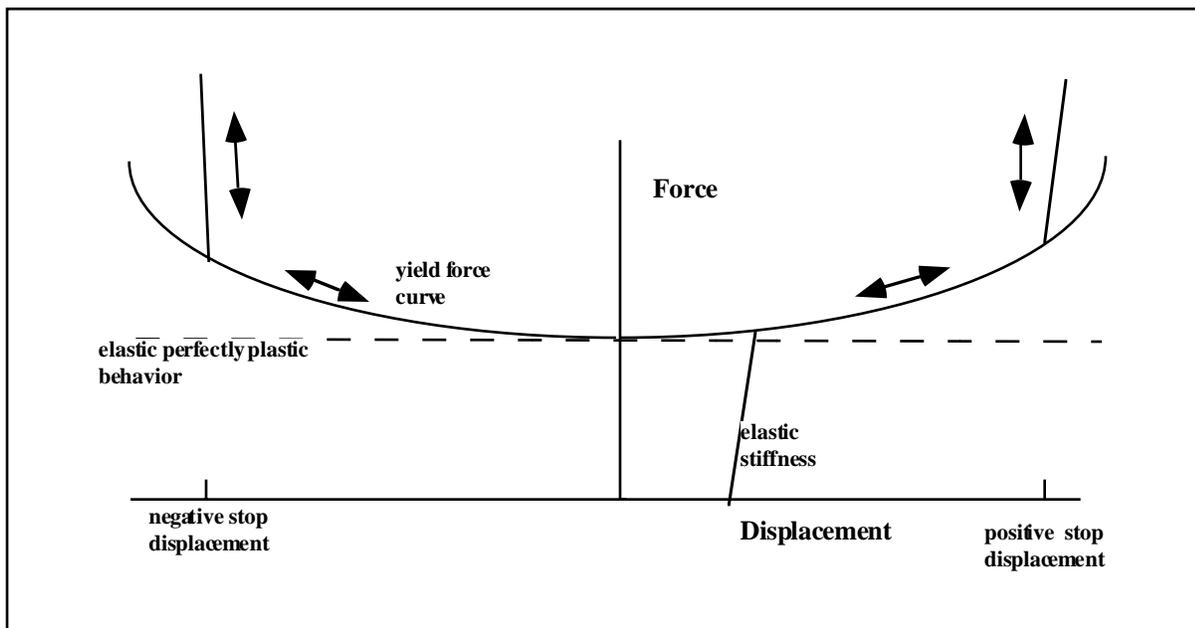


Figure 5.18. Frictional behavior is modeled by a plasticity model. Elastic behavior is obtained once the stop displacements are reached. The same elastic stiffness is used to simulate sticking situations.

*CONSTRAINED

*CONSTRAINED_LAGRANGE_IN_SOLID

Purpose: This command provides the mechanism for coupling interaction between a (slave) Lagrangian geometric entity (i.e. a Lagrangian mesh of shells, solids or beams) to a (master) ALE or Eulerian geometric entity (i.e. an ALE or Eulerian mesh).

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

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 1 2 3 4 5 6 7 8

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

Card 3 1 2 3 4 5 6 7 8

Variable	CQ	HMIN	HMAX	ILEAK	PLEAK	LCIDPOR		
Type	F	F	F	I	F			
Default	0.0	none	none	0	0.1			

<u>VARIABLE</u>	<u>DESCRIPTION</u>
SLAVE	Part, part set ID or Segment set ID of slaves see *PART , *SET_PART or *SET_SEGMENT. The Lagrangian parts are the slave entities.
MASTER	Part or part set ID of master solid elements, see *PART or *SET_PART. The Eulerian or ALE parts are the master entities.
SSTYP	Slave type: EQ.0: part set ID (PSID), EQ.1: part ID (PID), EQ.2: segment set ID (SGSID),
MSTYP	Master type: EQ.0: part set ID (PSID), EQ.1: part ID (PID),.
NQUAD	Quadrature rule for coupling slaves to solids. If NQUAD=n, then n is the number of coupling points assigned to a surface of each Lagrangian slave segment. There will be n-by-n couplings points distributed over each Lagrangian segment surface (see remark 3). 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 without erosion (Shell and Solid Elements), EQ.5: penalty coupling allowing erosion in the Lagrangian entities (Solid Elements). EQ.6: penalty coupling designed for airbag modeling (testing). DIREC is automatically reset to DIREC=1.
DIREC	Coupling direction (CTYPE 4 and 5, see remark 4). EQ.1: normal direction, compression and tension (default), EQ.2: normal direction, compression only, EQ.3: all directions.
MCOUP	Multi-material option (CTYPE 4 and 5, see remark 5). EQ.0: couple with all multi-material groups, EQ.1: couple with material with highest density. EQ.-n: refers to a set ID of an ALE multi-materal groups defined in *SET_MULTI-MATERIAL_GROUP card in which its set ID=n.
START	Start time for coupling.
END	End time for coupling.

*CONSTRAINED

VARIABLE	DESCRIPTION
PFAC	Penalty factor (CTYPE 4 and 5 only). PFAC is a scale factor for scaling the estimated stiffness of the interacting (coupling) system. It is used to compute the coupling forces to be distributed on the slave and master parts. If positive real: Fraction of estimated critical stiffness. If negative integer, -n: Refers to load curve ID n. The curve defines the contact pressure (y-axis) as a function of the penetration (x-axis). (See Remark 6)
FRIC	Coefficient of friction (DIREC 2 only).
FRCMIN	Minimum volume fraction to activate coupling (MCOUP=1)
NORM	A flag indicating the rule for defining which side of the segment the coupling is to be activated. The coupling direction may be computed based on the normal vector orientations of the Lagrangian shell part(s) (or segment set). The fluid(s) pointed to by the segment normals are to be coupled to, by default (NORM=0). But this direction can also be reversed by setting NORM=1 (see remark 7): EQ.0: normals are defined according to the right hand rule (default), EQ.1: normals are defined according to the left hand rule.
NORMTYP	Penalty coupling spring direction (DIREC 1 and 2): EQ.0: normal vectors are interpolated from nodal normals (default), EQ.1: normal vectors are interpolated from segment normals.
DAMP	Damping factor for penalty type 4. This is a coupling-damping-frequency in terms of a percentage (%) of the critical system frequency.
CQ	Heat transfer coefficient, C_q .
HMIN	Minimum air gap in heat transfer, h_{\min} .
HMAX	Maximum air gap in heat transfer, h_{\max} . There is no heat transfer above this value.
ILEAK	Coupling leakage control flag: EQ.0: none (default), EQ.1: weak, leakage control is turned off if volfrac > FRCMIN+0.1 (FRCMIN=0.3) EQ.2: strong, leakage control is turned off if volfrac > FRCMIN+0.3 (FRCMIN=0.3)
PLEAK	Leakage control penalty factor, $0 < \text{PLEAK} < 0.2$. This factor influences the additional coupling force magnitude to prevent leakage.
LCIDPOR	A load curve ID (LCID) defining porous flow through coupling segment: abscissa=x=coupling_ΔP ordinate=y=relative_porous_fluid_velocity The coupling ΔP is defined as

$$\Delta P_{coupling} \equiv (P_{in} - P_{out}) = \frac{\Delta F_{coupling}}{Area_{coupling}}$$

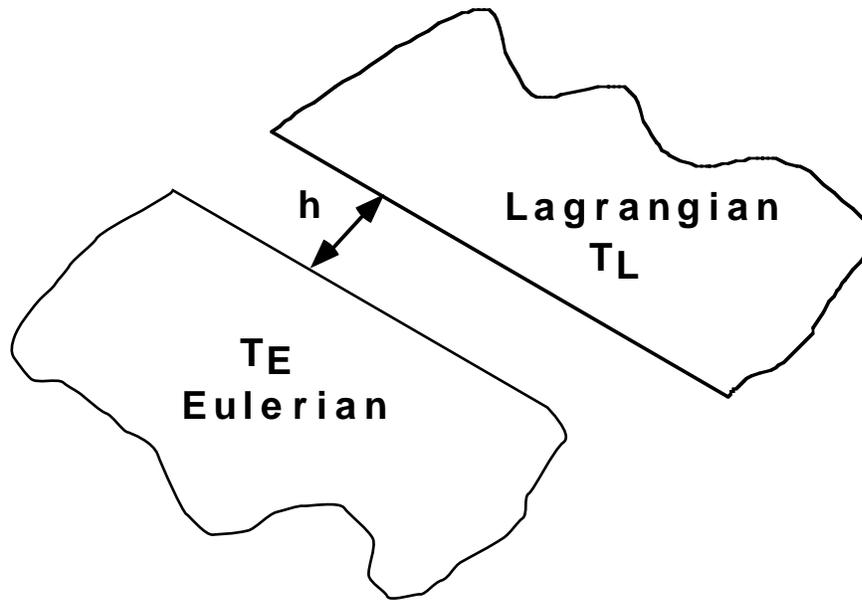
where P_{in} and P_{out} are the pressures inside and outside of the coupling segment. The porous velocity is the ALE fluid velocity relative to the moving Lagrangian segment. This curve is typically obtained from experiment.

Remark:

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

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

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



CQ, HMIN, HMAX are defined for a heat transfer interface between the Eulerian master part(s) and the Lagrangian slave part(s) assuming there is a gap at their interface(s). This is a preliminary means for heat transfer "coupling" effect. They must be derived from data in general.

2. 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.
3. NQUAD should generally be 2 or 3 per each Eulerian/ALE element. Consider case 1 where 1 Lagrangian slave segment spans, say, 2 ALE elements. Then NQUAD for each Lagrangian segment should be 4 or 6. Consider case 2 where 2 or 3 Lagrangian segments spanning 1 ALE elements, then maybe NQUAD=1 should be adequate.

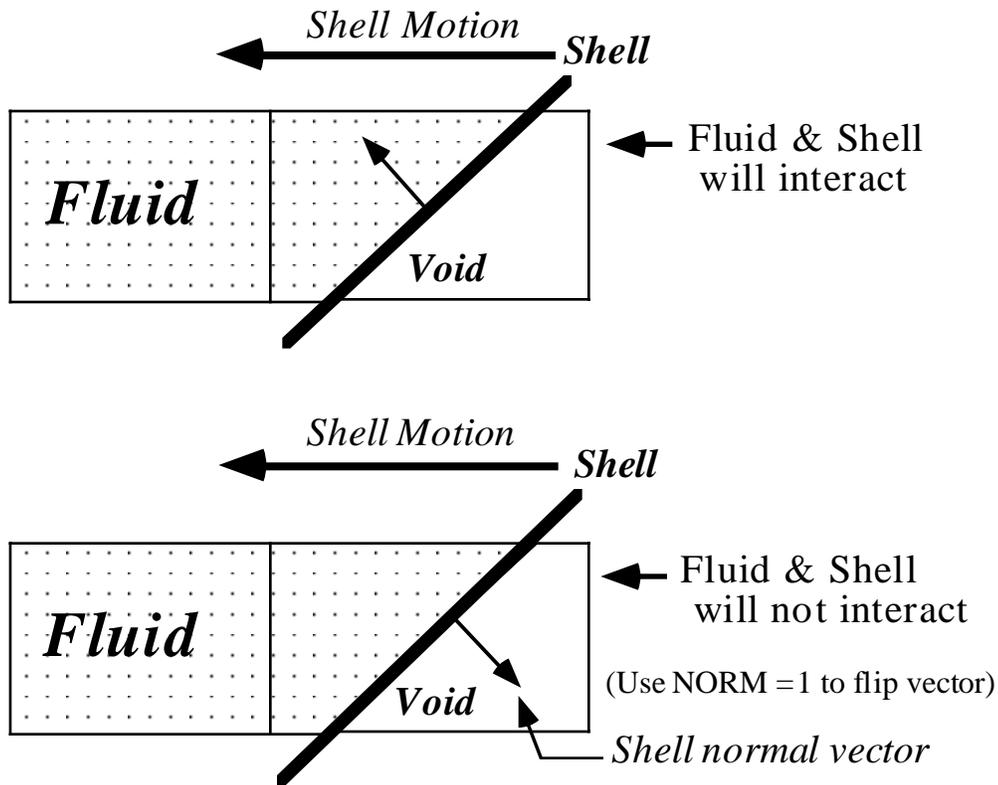
*CONSTRAINED

4. DIREC=2 may be generally a more stable and robust choice for coupling direction. However a choice of coupling direction should be made based on the physical set-up of the problem.
5. When MCOUP is a negative integer, say for example MCOUP= -123, then an ALE multi-material set-ID (AMMSID) of 123 must exist, which is defined by a *SET_MULTI-MATERIAL_GROUP_LIST card.
6. When PFAC is a negative integer, say for example PFAC= -321, then a load curve ID (LCID) of 321 must exist, which is defined by a *DEFINE_CURVE card. The x-axis of this curve is the amount of fluid penetration across the Lagrangian segment allowed for a corresponding coupling pressure value (y-axis). We may be able to allow a small penetration for an estimated maximum pressure in the fluid. In the example below, a penetration of 1.0E-3 m is associated with a maximum pressure range in the fluid of 4 atm or about 405300 Pascals.

For example, consider a coupling between an airbag with the gas from the inflator canister. This example demonstrates the usage of both MCOUP and PFAC when they are negative integers.

```
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|....8
$ PID 21 = ALE inflator gas; PID 22 = air mesh surrounding the airbag
*ALE_MULTI-MATERIAL_GROUP
$      SID      IDTYPE
        21         1
        22         1
$ ALEMMGID = 1 <= PID 21 <== see 1st line of the *ALE_MULTI-MATERIAL_GROUP
card.
*SET_MULTI-MATERIAL_GROUP_LIST
$      AMMSID
        123
$      ALEMMGID
        1
$ Lagrangian = slave = PSID 1 contains all airbag parts
*SET_PART
$      SID      DA1      DA2      DA3      DA4
        1         0.         0.         0.         0.
$      PID1      PID2      PID3      PID4      PID5      PID6      PID7
PID8
        3         4         5         6         7         8         9
0
$ Eulerian = master = PSID 11 contains all fluid mesh (geometrical space to
search).
*SET_PART_LIST
        11
        2
*CONSTRAINED_LAGRANGE_IN_SOLID
$      SLAVE      MASTER      SSTYP      MSTYP      NQUAD      CTYPE      DIREC
MCOUP
        1         11         0         0         4         4         2
-123
$      START      END      PFAC      FRIC      FRCMIN      NORM
        0.0        0.0      -321      0.00      0.3         1
$      CQ      HMIN      HMAX      ILEAK      PLEAK      VLK_PLCID
        0         0         0         2         0.10
$ We should couple to only the inflator gas (example unit for kg-m-s-K)
$ fluid penetration ~ 1 mm <====> Pmax = 405300 pascal ~ 4 atm
*DEFINE_CURVE
321
0.0000, 0.0
1.0e-3, 405300.0
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|....8
```

7. The normal vectors of the Lagrangian segments should point in the correct direction, i.e. toward the fluid(s) to be coupled. Sometimes a shell part or mesh is generated such that its normal vectors do not point uniformly in a consistent direction (all toward the inside or outside of a container, etc.) A normal of a segment is defined by going around the 4 node ID's according to the right hand rule. So to switch the normal direction is equivalent to switching the order of the node IDs of a segment or shell element. The user should always check for the normal vectors of any Lagrangian shell part interacting with any fluid. The NORM parameter may be used to flip the normal direction of all the segments included in the Lagrangian slave set.



*CONSTRAINED

*CONSTRAINED_LINEAR_GLOBAL

Purpose: Define linear constraint equations between displacements and rotations, which can be defined in global coordinate systems. For a newer and for a more general constraint see *CONSTRAINED_INTERPOLATION

Card 1 - Required

Card 1 1 2 3 4 5 6 7 8

Variable	LCID								
Type	I								
Default	none								

Card 2 - Define one card for each constrained degree-of-freedom. Input is terminated when a "*" card is found.

Card 2 1 2 3 4 5 6 7 8

Variable	NID	DOF	COEF						
Type	I	I	I						
Default	none	0	0						
Remark	1								

<u>VARIABLE</u>	<u>DESCRIPTION</u>
LCID	Linear constraint definition ID. This ID can be used to identify a set to which this constraint is a member.
NID	Node ID
DOF	Degree of freedom in the local coordinate system; EQ.1: displacement along global x-direction EQ.2: displacement along global y-direction EQ.3: displacement along global z-direction EQ.4: global rotation about global x-axis EQ.5: global rotation about global y-axis EQ.6: global rotation about global z-axis

<u>VARIABLE</u>	<u>DESCRIPTION</u>
COEF	Nonzero coefficient, C_k

Remarks:

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

In this section linear constraint equations of the form:

$$\sum_{k=1}^n C_k u_k = C_0$$

can be defined, where u_k are the displacements and C_k are user defined coefficients. Unless LS-DYNA is initialized by linking to an implicit code to satisfy this equation at the beginning of the calculation, the constant C_0 is assumed to be zero. The first constrained degree-of-freedom is eliminated from the equations-of-motion:

$$u_1 = C_0 - \sum_{k=2}^n \frac{C_k}{C_1} u_k$$

Its velocities and accelerations are given by

$$\begin{aligned} \dot{u}_1 &= - \sum_{k=2}^n \frac{C_k}{C_1} \dot{u}_k \\ \ddot{u}_1 &= - \sum_{k=2}^n \frac{C_k}{C_1} \ddot{u}_k, \end{aligned}$$

respectively. In the implementation a transformation matrix, L , is constructed relating the unconstrained, u , and constrained, u_c , degrees-of-freedom. The constrained accelerations used in the above equation are given by:

$$\ddot{u}_c = \left[\tilde{L}^t \tilde{M} \tilde{L} \right]^{-1} \tilde{L}^t \tilde{F}$$

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

***CONSTRAINED_LINEAR_LOCAL**

Purpose: Define linear constraint equations between displacements and rotations, which can be defined in a local coordinate system. Each node may have a unique coordinate ID.

Card 1 - Required

Card 1 1 2 3 4 5 6 7 8

Variable	LCID							
Type	I							
Default	none							

Card 2 - Define one card for each constrained degree-of-freedom. Input is terminated when a "*" card is found.

Card 2 1 2 3 4 5 6 7 8

Variable	NID	DOF	CID	COEF				
Type	I	I	I	I				
Default	none	0	0	0				
Remark	1							

VARIABLE**DESCRIPTION**

LCID	LCID for linear constraint definition. This ID can be used to identify a set to which this constraint is a member.
NID	Node ID
DOF	Degree of freedom in the local coordinate system; EQ.1: displacement along local x-direction EQ.2: displacement along local y-direction EQ.3: displacement along local z-direction EQ.4: local rotation about local x-axis EQ.5: local rotation about local y-axis EQ.6: local rotation about local z-axis

*CONSTRAINED

<u>VARIABLE</u>	<u>DESCRIPTION</u>
CID	Local coordinate system ID number. If the number is zero, the global coordinate system is used.
COEF	Nonzero coefficient, C_k

Remarks:

In this section linear constraint equations of the form:

$$\sum_{k=1}^n C_k u_k^L = C_0$$

can be defined, where u_k^L are the displacements in the local coordinate systems and C_k are user defined coefficients. Unless LS-DYNA is initialized by linking to an implicit code to satisfy this equation at the beginning of the calculation, the constant C_0 is assumed to be zero. The first constrained degree-of-freedom is eliminated from the equations-of-motion:

$$u_1^L = C_0 - \sum_{k=2}^n \frac{C_k}{C_1} u_k^L$$

Its velocities and accelerations are given by

$$\dot{u}_1^L = - \sum_{k=2}^n \frac{C_k}{C_1} \dot{u}_k^L$$

$$\ddot{u}_1^L = - \sum_{k=2}^n \frac{C_k}{C_1} \ddot{u}_k^L$$

respectively. The local displacements are calculated every time step using the local coordinate systems defined by the user. More than one degree of freedom for a node can be constrained by specifying a card for each degree of freedom.

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

***CONSTRAINED_NODAL_RIGID_BODY_{OPTION}_{OPTION}**

If the center of mass is constrained, the option:

SPC

can be used and 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.

The first node in the nodal rigid body definition is treated as the master for the case where DRFLAG and RRFLAG are nonzero. The first node always has six degrees-of-freedom. The release conditions applied in the global system are sometimes convenient in small displacement linear analysis, but, otherwise, are not recommended. It is strongly recommended, especially for implicit calculations, that release conditions are only used for a two noded nodal rigid body.

Card Format:

Card 1 is required.

Cards 2 - 4 are required for the INERTIA option.

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

Remarks:

1. Unlike the *CONSTRAINED_NODE_SET which permits only constraints on translational motion, here the equations of rigid body dynamics are used to update the motion of the nodes and therefore rotations of the nodal sets are admissible. Mass properties are determined from the nodal masses and coordinates. Inertial properties are defined if and only if the INERTIA option is specified.

Card 1 - Required.

Card 1 1 2 3 4 5 6 7 8

Variable	PID	CID	NSID	PNODE	IPRT	DRFLAG	RRFLAG	
Type	I	I	I	I	I	I	I	
Default	none	none	none	0	0	0	0	

*CONSTRAINED

Define if and only if SPC is specified in the keyword.

Card 1 1 2 3 4 5 6 7 8

Variable	CMO	CON1	CON2						
Type	F	F	F						
Default	0	0	0						

<u>VARIABLE</u>	<u>DESCRIPTION</u>
PID	Part ID of the nodal rigid body.
CID	Optional coordinate system ID for the rigid body local system, see *DEFINE_COORDINATE_OPTION. Output of the rigid body data and the degree-of- freedom releases are done in this local system. This local system rotates with the rigid body.
NSID	Nodal set ID, see *SET_NODE_OPTION. This nodal set defines the rigid body. If NSID=0, then NSID=PID, i.e., the node set ID and the part ID are assumed to be identical.
PNODE	An optional, possibly massless, nodal point located at the mass center of the nodal rigid body. The initial nodal coordinates will be reset if necessary to ensure that they lie at the mass center. In the output files, the coordinates, accelerations, velocities, and displacements of this node will correspond to the mass center of the nodal rigid body. If CID is defined, the velocities and accelerations of PNODE will be output in the local system in the D3PLOT and D3THDT files unless PNODE is specified as a negative number in which case the global system is used.
IPRT	Print flag. For nodal rigid bodies the following values apply: EQ.1: write data into RBDOUT EQ.2: do not write data into RBDOUT Printing is suppressed for two noded rigid bodies unless IPRT is set to unity. This is to avoid excessively large RBDOUT files when many, two-noded welds are used.
DRFLAG	Displacement release flag for all nodes except the first node in the definition. EQ.-7: release x, y, and z displacement in global system EQ.-6: release z and x displacement in global system EQ.-5: release y and z displacement in global system EQ.-4: release x and y displacement in global system

<u>VARIABLE</u>	<u>DESCRIPTION</u>
	EQ.-3: release z displacement in global system EQ.-2: release y displacement in global system EQ.-1: release x displacement in global system EQ. 0: off for rigid body behavior EQ. 1: release x displacement in rigid body local system EQ. 2: release y displacement in rigid body local system EQ. 3: release z displacement in rigid body local system EQ. 4: release x and y displacement in rigid body local system EQ. 5: release y and z displacement in rigid body local system EQ. 6: release z and x displacement in rigid body local system EQ. 7: release x, y, and z displacement in rigid body local system
RRFLAG	Rotation release flag for all nodes except the first node in the definition. EQ.-7: release x, y, and z rotations in global system EQ.-6: release z and x rotations in global system EQ.-5: release y and z rotations in global system EQ.-4: release x and y rotations in global system EQ.-3: release z rotation in global system EQ.-2: release y rotation in global system EQ.-1: release x rotation in global system EQ. 0: off for rigid body behavior EQ. 1: release x rotation in rigid body local system EQ. 2: release y rotation in rigid body local system EQ. 3: release z rotation in rigid body local system EQ. 4: release x and y rotations in rigid body local system EQ. 5: release y and z rotations in rigid body local system EQ. 6: release z and x rotations in rigid body local system EQ. 7: release x, y, and z rotations in rigid body local system
CMO	Center of mass constraint option, CMO: EQ.+1.0: constraints applied in global directions, EQ. 0.0: no constraints, EQ. -1.0: constraints applied in local directions (SPC constraint).
CON1	First constraint parameter: <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. <u>If CMO=-1.0, then specify local coordinate system ID.</u> See *DEFINE_ COORDINATE_OPTION: This coordinate system is fixed in time.

*CONSTRAINED

VARIABLE	DESCRIPTION
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>

Card 2 of 4 - Required for the INERTIA option.

Card 2	1	2	3	4	5	6	7	8
Variable	XC	YC	ZC	TM	IRCS	NODEID		
Type	F	F	F	F	I	I		
Default	0	0	0	0	0	0		

VARIABLE	DESCRIPTION
XC	x-coordinate of center of mass. If nodal point, NODEID, is defined XC, YC, and ZC are ignored and the coordinates of the nodal point, NODEID, are taken as the center of mass.
YC	y-coordinate of center of mass
ZC	z-coordinate of center of mass
TM	Translational mass

<u>VARIABLE</u>	<u>DESCRIPTION</u>
IRCS	Flag for inertia tensor reference coordinate system: EQ.0: global inertia tensor, EQ.1: principal moments of inertias with orientation vectors as given below.
NODEID	Optional nodal point defining the CG of the rigid body. If this node is not a member of the set NSID above, its motion will not be updated to correspond with the nodal rigid body after the calculation begins. PNODE and NODEID can be identical if and only if PNODE physically lies at the mass center at time zero.

Card 3 of 4 - Required for the INERTIA option.

Card 3 1 2 3 4 5 6 7 8

Variable	IXX	IXY	IXZ	IYY	IYZ	IZZ		
Type	F	F	F	F	F	F		
Default	none	0	0	none	0	0		

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

Card 4 of 4 - Required for the INERTIA option.

Card 4 1 2 3 4 5 6 7 8

Variable	VTX	VTY	VTZ	VRX	VRY	VRZ		
Type	F	F	F	F	F	F		
Default	0	0	0	0	0	0		

*CONSTRAINED

<u>VARIABLE</u>	<u>DESCRIPTION</u>
VTX	x-rigid body initial translational velocity in global coordinate system.
VTY	y-rigid body initial translational velocity in global coordinate system.
VTZ	z-rigid body initial translational velocity in global coordinate system.
VRX	x-rigid body initial rotational velocity in global coordinate system.
VRY	y-rigid body initial rotational velocity in global coordinate system.
VRZ	z-rigid body initial rotational velocity in global coordinate system.

Remark:

The velocities defined above can be overwritten by the *INITIAL_VELOCITY card.

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

Card 5 1 2 3 4 5 6 7 8

Variable	XL	YL	ZL	XLIP	YLIP	ZLIP	CID2	
Type	F	F	F	F	F	F	I	
Default	none							

<u>VARIABLE</u>	<u>DESCRIPTION</u>
XL	x-coordinate of local x-axis. Origin lies at (0,0,0).
YL	y-coordinate of local x-axis
ZL	z-coordinate of local x-axis
XLIP	x-coordinate of local in-plane vector
YLIP	y-coordinate of local in-plane vector
ZLIP	z-coordinate of local in-plane vector
CID2	Local coordinate system ID, see *DEFINE_COORDINATE_.... With this option leave fields 1-6 blank.

*CONSTRAINED

*CONSTRAINED_NODE_SET_{OPTION}

To define an ID for the constrained node set the following option is available:

ID

If the ID is defined an additional card is required.

Purpose: Define nodal constraint sets for translational motion in global coordinates. No rotational coupling. See Figure 5.19. Nodal points included in the sets should not be subjected to any other constraints including prescribed motion, e.g., with the *BOUNDARY_PRESCRIBED_MOTION options.

ID Card - Required if the option `_ID` is active on the keyword card.

Card 1 1 2 3 4 5 6 7 8

Variable	CNSID								
Type	I								
Default	0								

Card 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

CNSID	Optional constrained node set ID.
NSID	Nodal set ID, see *SET_NODE_OPTION.

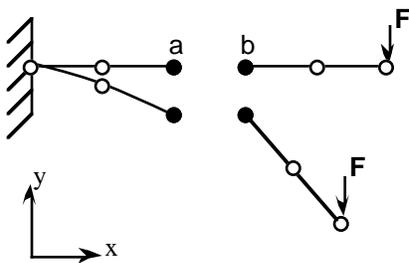
VARIABLE	DESCRIPTION
DOF	Applicable degrees-of-freedom: EQ. 1: x-translational degree-of-freedom, EQ. 2: y-translational degree-of-freedom, EQ. 3: z-translational degree-of-freedom, EQ. 4: x and y-translational degrees-of-freedom, EQ. 5: y and z-translational degrees-of-freedom, EQ. 6: z and x-translational degrees-of-freedom, EQ. 7: x, y, and z-translational degrees-of-freedom.
TF	Failure time for nodal constraint set.

Remarks:

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

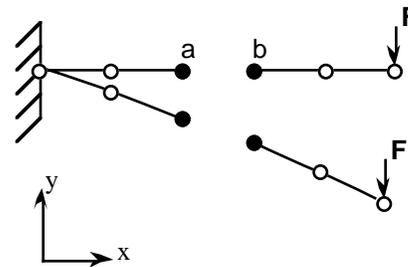
***CONSTRAINED_NODE_SET**

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



***CONSTRAINED_NODAL_RIGID_BODY**
***CONSTRAINED_SPOTWELD**

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



Offset nodes a and b are constrained to move together.

Figure 5.19. `*CONSTRAINED_NODE_SET` can lead to nonphysical responses.

***CONSTRAINED_POINTS**

Purpose: Constrain two points with the specified coordinates connecting two shell elements at locations other than nodal points. In this option, the penalty method is used to constrain the translational and rotational degrees-of-freedom of the points. Force resultants are written into the SWFORC ASCII file for post-processing.

Card Format (I10)

Card 1 1 2 3 4 5 6 7 8

Variable	CID								
Type	I								
Default	none								

Card Format (I8,3E16.0)

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

Variable	EID1	X1	Y1	Z1					
Type	I	F	F	F					
Default	none	0.	0.	0.					

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

Variable	EID2	X2	Y2	Z2					
Type	I	F	F	F					
Default	none	0.	0.	0.					

*CONSTRAINED

Card Format (4E10.0)

Card 4 1 2 3 4 5 6 7 8

Variable	PSF	FAILA	FAILS	FAILM				
Type	F	F	F	F				
Default	1.0	0.0	0.0	0.0				

<u>VARIABLE</u>	<u>DESCRIPTION</u>
CID	Constrained points ID.
Xi, Yi, Zi	Coordinates of the constrained points, i=1,2.
EIDi	Shell element ID, i=1,2.
PSF	Penalty scale factor (Default=1.0).
FAILA	Axial force resultant failure value (Skip if zero.).
FAILS	Shear force resultant failure value (Skip if zero.).
FAILM	Moment resultant failure value (Skip if zero.).

***CONSTRAINED_RIGID_BODIES**

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

Card Format

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

VARIABLE

DESCRIPTION

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

Remarks:

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

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

*CONSTRAINED_RIGID_BODY_STOPPERS

Purpose: Rigid body stoppers provide a convenient way of controlling the motion of rigid tooling in metalforming applications. The motion of a “master” rigid body is limited by load curves. This option will stop the motion based on a time dependent constraint. The stopper overrides prescribed motion boundary conditions (except relative displacement) operating in the same direction for both the master and slaved rigid bodies. See Figure 5.20.

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 ²¹						

VARIABLE

DESCRIPTION

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

*CONSTRAINED

VARIABLE	DESCRIPTION
LCMIN	Load curve ID defining the minimum coordinate or displacement as a function of time. See *DEFINE_CURVE: LT.0: Load Curve ID LCMIN defines a lower bound for the displacement of the rigid body EQ.0: no limitation of the minimum displacement. GT.0: Load Curve ID LCMIN defines a lower bound for the position of the rigid body center of mass
PSIDMX	Optional part set ID of rigid bodies that are slaved in the maximum coordinate direction to the master rigid body. The part set definition, (see *SET_PART_COLUMN) may be used to define the closure distance (D_1 and D_2 in Figure 5.20) which activates the constraint. The constraint does not begin to act until the master rigid body stops. If the distance between the master rigid body is greater than or equal to the closure distance, the slave rigid body motion away from the master rigid body also stops. However, the slaved rigid body is free to move towards the master. If the closure distance is input as zero (0.0) then the slaved rigid body stops when the master stops.
PSIDMN	Optional part set ID of rigid bodies that are slaved in the minimum coordinate direction to the master rigid body. The part set definition, (see *SET_PART_COLUMN) may be used to define the closure distance (D_1 and D_2 in Figure 5.20) which activates the constraint. The constraint does not begin to act until the master rigid body stops. If the distance between the master rigid body is less than or equal to the closure distance, the slave rigid body motion towards the master rigid body also stops. However, the slaved rigid body is free to move away from the master. If the closure distance is input as zero (0.0) then the slaved rigid body stops when the master stops.
LCVMX	Load curve ID which defines the maximum absolute value of the velocity as a function of time that is allowed for the master rigid body. See *DEFINE_CURVE: EQ.0: no limitation on the velocity.
DIR	Direction stopper acts in: EQ.1: x-translation, EQ.2: y-translation, EQ.3: z-translation, EQ.4: arbitrary, defined by vector VID (see below), EQ.5: x-axis rotation , EQ.6: y-axis rotation, EQ.7: z-axis rotation, EQ.8: arbitrary, defined by vector VID (see below).
VID	Vector for arbitrary orientation of stopper, see *DEFINE_VECTOR.
TB	Time at which stopper is activated.
TD	Time at which stopper is deactivated.

Remark:

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

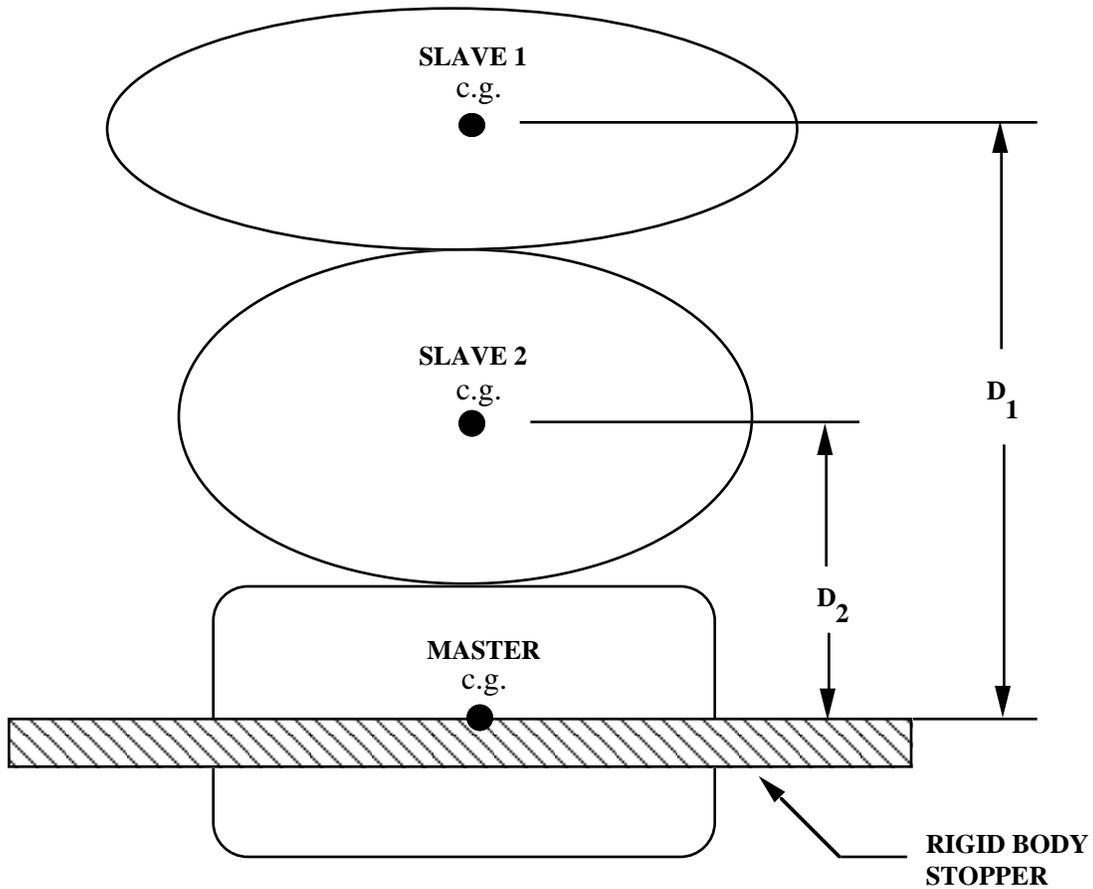


Figure 5.20 When the master rigid body reaches the rigid body stopper, the velocity component into the stopper is set to zero. Slave rigid bodies 1 and 2 also stop if the distance between their mass centers and the master rigid body is less than or equal to the input values D_1 and D_2 , respectively. (c.g. \equiv center of gravity).

*CONSTRAINED

*CONSTRAINED_RIVET_{OPTION}

To define an ID for the rivet, the following option is available:

ID

If the ID is defined an additional card is required.

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

ID Card - Required if the option ID is active on the keyword card.

Card 1 1 2 3 4 5 6 7 8

Variable	RID								
Type	I								
Default	0								

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

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

*CONSTRAINED

*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								

<u>VARIABLE</u>	<u>DESCRIPTION</u>
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 5.21. During the calculation, the brick nodes thus constrained must lie along the fiber but can move relative to each other in the fiber direction. The shell node stays on the fiber at the same relative spacing between the first and last brick node. The brick nodes must be input in the order in which they occur, in either the plus or minus direction, as one moves along the shell node fiber.

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

*CONSTRAINED

*CONSTRAINED_SPOTWELD_{OPTION}_{OPTION}

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

FILTERED_FORCE

and one additional card must be defined below. To define an ID for the spot weld the following option is available:

ID

If the ID is defined an additional card is required. The ordering of the options is arbitrary.

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

Warning: The accelerations of spot welded nodes are output as zero into the various databases, but if the acceleration of spotwelded nodes are required, use either the *CONSTRAINED_GENERALIZED_WELD or the *CONSTRAINED_NODAL_RIGID_BODY input. However, if the output interval is frequent enough accurate acceleration time histories can be obtained from the velocity time history by differentiation in the post-processing phase.

ID Card - Required if the option _ID is active on the keyword card.

Card 1 1 2 3 4 5 6 7 8

Variable	WID								
Type	I								
Default	0								

*CONSTRAINED

Card 1 Format

Card 1 1 2 3 4 5 6 7 8

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

Card 2 Format Define if and only if the option `_FILTERED_FORCE` is specified.

Card 2 1 2 3 4 5 6 7 8

Variable	NF	TW						
Type	I	F						
Default	none	none						
Remarks								

VARIABLE

DESCRIPTION

WID	Optional weld ID.
N1	Node ID
N2	Node ID
SN	Normal force at spotweld failure (see Remark 2 below).
SS	Shear force at spotweld failure (see Remark 2 below).
N	Exponent for normal spotweld force (see Remark 2 below).
M	Exponent for shear spotweld force (see Remark 2 below).

*CONSTRAINED

<u>VARIABLE</u>	<u>DESCRIPTION</u>
TF	Failure time for nodal constraint set.
EP	Effective plastic strain at failure.
NF	Number of force vectors stored for filtering.
TW	Time window for filtering.

Remarks:

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

2. Failure of the spot welds occurs when:

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

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

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

*CONSTRAINED

*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					

<u>VARIABLE</u>	<u>DESCRIPTION</u>
SNSID	Slave node set ID, see *SET_NODE_OPTION.
MNSID	Master node set ID, see *SET_NODE_OPTION.
EPPF	Plastic strain at failure

Remarks:

- Nodes in the master node set must be given in the order they appear as one moves along the edge of the surface.
- Tie-breaks may not cross.
- 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.
- Tie-breaks may be used to simulate the effect of failure along a predetermined line, such as a seam or structural joint. When the failure criterion is reached in the adjoining elements, nodes along the slideline will begin to separate. As this effect propagates, the tie-breaks will appear to “unzip,” thus simulating failure of the connection.

***CONSTRAINED_TIED_NODES_FAILURE**

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

Card Format

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

VARIABLE**DESCRIPTION**

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

Remarks:

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

***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_AUTO_MOVE**

***CONTACT_COUPLING**

***CONTACT_ENTITY**

***CONTACT_GEBOD_OPTION**

***CONTACT_INTERIOR**

***CONTACT_RIGID_SURFACE**

***CONTACT_1D**

***CONTACT_2D_OPTION1_{OPTION2}_{OPTION3}**

The first, ***CONTACT_...**, is the general 3D contact algorithms. The second, ***CONTACT_COUPLING**, provides a means of coupling to deformable surfaces to MADYMO. The third, ***CONTACT_ENTITY**, treats contact using mathematical functions to describe the surface geometry for the master surface. The fourth, ***CONTACT_GEBOD** is a specialized form of the contact entity for use with the rigid body dummies (see ***COMPONENT_GEBOD**). The fifth, ***CONTACT_INTERIOR**, is under development and is used with soft foams where element inversion is sometimes a problem. Contact between layers of brick elements is treated to eliminate negative volumes. The sixth, ***CONTACT_RIGID_SURFACE** is for modeling road surfaces for durability and NVH calculations. The seventh, ***CONTACT_1D**, remains in LS-DYNA for historical reasons, and is sometimes still used to model rebars which run along edges of brick elements. The last, ***CONTACT_2D**, is the general 2D contact algorithm based on those used previously in LS-DYNA2D.

***CONTACT**

***CONTACT_OPTION1_{OPTION2}_{OPTION3}_{OPTION4}**

Purpose: Define a contact interface.

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

AIRBAG_SINGLE_SURFACE
AUTOMATIC_GENERAL
AUTOMATIC_GENERAL_INTERIOR
AUTOMATIC_NODES_TO_SURFACE
AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE
AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE_TIEBREAK
AUTOMATIC_SINGLE_SURFACE
AUTOMATIC_SURFACE_TO_SURFACE
AUTOMATIC_SURFACE_TO_SURFACE_TIEBREAK
CONSTRAINT_NODES_TO_SURFACE
CONSTRAINT_SURFACE_TO_SURFACE
DRAWBEAD
ERODING_NODES_TO_SURFACE
ERODING_SINGLE_SURFACE
ERODING_SURFACE_TO_SURFACE
FORCE_TRANSDUCER_CONSTRAINT
FORCE_TRANSDUCER_PENALTY
FORMING_NODES_TO_SURFACE
FORMING_ONE_WAY_SURFACE_TO_SURFACE
FORMING_SURFACE_TO_SURFACE
NODES_TO_SURFACE
NODES_TO_SURFACE_INTERFERENCE
ONE_WAY_SURFACE_TO_SURFACE
ONE_WAY_SURFACE_TO_SURFACE_INTERFERENCE
RIGID_NODES_TO_RIGID_BODY
RIGID_BODY_ONE_WAY_TO_RIGID_BODY
RIGID_BODY_TWO_WAY_TO_RIGID_BODY
SINGLE_EDGE
SINGLE_SURFACE
SLIDING_ONLY
SLIDING_ONLY_PENALTY

SPOTWELD
SPOTWELD_WITH_TORSION
SURFACE_TO_SURFACE
SURFACE_TO_SURFACE_INTERFERENCE
TIEBREAK_NODES_TO_SURFACE
TIEBREAK_NODES_ONLY
TIEBREAK_SURFACE_TO_SURFACE
TIED_NODES_TO_SURFACE
TIED_SHELL_EDGE_TO_SURFACE
TIED_SURFACE_TO_SURFACE
TIED_SURFACE_TO_SURFACE_FAILURE

OPTION2 specifies a thermal contact and takes the single option:

THERMAL

Only the SURFACE_TO_SURFACE contact type may be used with this option.

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

ID

OPTION4 specifies that offsets may be used with the tied contacts types. If one of these three offset options is set, then offsets are permitted for these contact types, and, if not, the nodes are projected back to the contact surface during the initialization phase and a constraint formulation is used. Note that in a constraint formulation the nodes of rigid bodies are not permitted in the definition.

OFFSET

Contact types TIED_NODES_TO_SURFACE, TIED_SHELL_EDGE_TO_SURFACE, and TIED_SURFACE_TO_SURFACE may be used with this option. The OFFSET option switches the formulation from a constraint type formulation to one that is penalty based where the force and moment (if applicable) resultants are transferred discrete spring elements between the slave nodes and master segments. For the TIED_SHELL_EDGE_TO_SURFACE contact the BEAM_OFFSET option may be preferred. Rigid bodies can be used with this option.

BEAM_OFFSET

This option applies only to contact type TIED_SHELL_EDGE_TO_SURFACE. If this option is set, then offsets are permitted for this contact type. The BEAM_OFFSET option switches the formulation from a constraint type formulation to one that is penalty based. Beam like springs are used to transfer force and moment resultants between the slave nodes and the master segments. Rigid bodies can be used with this option.

*CONTACT

CONSTRAINED_OFFSET

Contact types TIED_NODES_TO_SURFACE, TIED_SHELL_EDGE_TO_SURFACE, and TIED_SURFACE_TO_SURFACE may be used with this option. If this option is set, then offsets are permitted for these contact types. The CONSTRAINED_OFFSET option is a constraint type formulation. The nodal points in the TIED_NODES_TO_SURFACE option and the TIED_SURFACE_TO_SURFACE may not be connected to structural nodes, i.e., nodes with rotational degrees-of-freedom, since the rotational degrees-of-freedom are not affected, which will lead to an instability since the translational motions due to rotation are imposed on the slave nodes.

Remarks:

1. *OPTION1*, *OPTION2*, *OPTION3* and *OPTION4* may appear in any order in the keyword command line. The data must be in the order specified below.
2. *OPTION1* is mandatory.
3. *OPTION2*, *OPTION3* and *OPTION4* are optional.
4. The following contact types are available for implicit calculations:

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

DISCUSSION AND EXAMPLES:

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

Card ordering is important in this section:

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

Define the ID and heading card first.

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

- *CONTACT_CONSTRAINT_type**
- *CONTACT_DRAWBEAD**
- *CONTACT_ERODING_type**
- *CONTACT_..._INTERFERENCE**
- *CONTACT_RIGID_type**
- *CONTACT_TIEBREAK_type**

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

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

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

- **Optional Card A**

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

- **Optional Card B**

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

Card 1 is mandatory for all contact types.

Card 1 1 2 3 4 5 6 7 8

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

VARIABLE

DESCRIPTION

SSID	Slave segment, node set ID, partset ID, part ID, or shell element set ID, see *SET_SEGMENT, *SET_NODE_OPTION, *PART, *SET_PART or *SET_SHELL_OPTION. For eroding contact use either a part ID or a partset ID. EQ.0: all part IDs are included for single surface contact, automatic single surface, and eroding single surface.
MSID	Master segment set ID, partset ID, part ID, or shell element set ID, see *SET_SEGMENT, *SET_NODE_OPTION, *PART, *SET_PART, or *SET_SHELL_OPTION: EQ.0: for single surface contact, automatic single surface, and eroding single surface.
SSTYP	Slave segment or node set type. The type must correlate with the number specified for SSID: EQ.0: segment set ID for surface to surface contact, EQ.1: shell element set ID for surface to surface contact, EQ.2: part set ID, EQ.3: part ID, EQ.4: node set ID for node to surface contact, EQ.5: include all for single surface 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.

*CONTACT

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.

Card 2 is mandatory for all contact types.

Card 2 1 2 3 4 5 6 7 8

Variable	FS	FD	DC	VC	VDC	PENCHK	BT	DT
Type	F	F	F	F	F	I	F	F
Default	0.	0.	0.	0.	0.	0	0.	1.0E20
Remarks								

VARIABLE

DESCRIPTION

FS

Static coefficient of friction if FS is >0. and not equal to 2. The frictional coefficient is assumed to be dependent on the relative velocity v_{rel} of the surfaces in contact $\mu_c = FD + (FS - FD)e^{-DC \cdot |v_{rel}|}$. The two other possibilities are:

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

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

EQ. 2: For contact types SURFACE_TO_SURFACE and ONE_WAY_SURFACE_TO_SURFACE, the dynamic coefficient of friction points to the table, see DEFINE_TABLE (The table ID is give by FD below.), giving the coefficient of friction as a function of the relative velocity and pressure. This option must be used in combination with the thickness offset option. See Figure 6.1.

FD

Dynamic coefficient of friction. The frictional coefficient is assumed to be dependent on the relative velocity v_{rel} of the surfaces in contact $\mu_c = FD + (FS - FD)e^{-DC \cdot |v_{rel}|}$. Give table ID if FS=2.

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

*CONTACT

VARIABLE	DESCRIPTION
	failure occurs if $\left[\frac{\max(0.0, \sigma_{normal})}{FS} \right]^2 + \left[\frac{\sigma_{shear}}{FD} \right]^2 - 1 > 0$ where σ_{normal} and σ_{shear} are the interface normal and shear stresses.
FS	Normal tensile stress at failure
FD	Shear stress at failure
DC	Exponential decay coefficient. The frictional coefficient is assumed to be dependent on the relative velocity v_{rel} of the surfaces in contact $\mu_c = FD + (FS - FD)e^{-DC v_{rel} }$
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 σ_o is the yield stress of the contacted material.
VDC	Viscous damping coefficient in percent of critical. In order to avoid undesirable oscillation in contact, e.g., for sheet forming simulation, a contact damping perpendicular to the contacting surfaces is applied. Damping coefficient $\xi = \frac{VDC}{100} \xi_{wd}$, eg VDC = 20. ξ_{crit} is determined in the following fashion by LS-DYNA. $\xi_{crit} = 2mw; \quad m = \min(m_{slave}, m_{master}) \quad \begin{matrix} \text{mass of master} \\ \text{resp. slave node} \end{matrix}$ $w = \sqrt{k \cdot \frac{m_{slave} + m_{master}}{m_{slave} \cdot m_{master}}} \quad k \text{ interface stiffness}$
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).

Card 3 is mandatory for all contact types.

Card 3 1 2 3 4 5 6 7 8

Variable	SFS	SFM	SST	MST	SFST	SFMT	FSF	VSF
Type	F	F	F	F	F	F	F	F
Default	1.	1.	element thickness	element thickness	1.	1.	1.	1.

VARIABLE

DESCRIPTION

SFS	Scale factor on default slave penalty stiffness, see also *CONTROL_CONTACT.
SFM	Scale factor on default master penalty stiffness, see also *CONTROL_CONTACT.
SST	Optional thickness for slave surface (overrides true thickness). This option applies only to contact with shell elements. True thickness is the element thickness of the shell elements. For the *CONTACT_TIED... options, SST and MST below can be defined as negative values, which will cause the determination of whether or not a node is tied to depend only on the separation distance relative to the absolute value of these thicknesses. More information is given under <u>General Remarks on *CONTACT</u> following Optional Card C.
MST	Optional thickness for master surface (overrides true thickness). This option applies only to contact with shell elements. True thickness is the element thickness of the shell elements. For the TIED options see SST above.
SFST	Scale factor for slave surface thickness (scales true thickness). This option applies only to contact with shell elements. True thickness is the element thickness of the shell elements.
SFMT	Scale factor for master surface thickness (scales true thickness). This option applies only to contact with shell elements. True thickness is the element thickness of the shell elements.
FSF	Coulomb friction scale factor. The Coulomb friction value is scaled as $\mu_{sc} = FSF \cdot \mu_c$, see above.
VSF	Viscous friction scale factor. If this factor is defined then the limiting force becomes: $F_{lim} = VSF \cdot VC \cdot A_{cont}$, see above.

*CONTACT

Remarks:

The variables FSF and VSF above can be overridden segment by segment on the *SET_SEGMENT or *SET_SHELL_OPTION cards for the **slave surface only** as A3 and A4, and for the **master surface only** as A1 and A2. See *SET_SEGMENT and *SET_SHELL_OPTION.

This Card 4 is mandatory for:

***CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_TIEBREAK**
***CONTACT_AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE_TIEBREAK**

Card 4 1 2 3 4 5 6 7 8

Variable	OPTION	NFLS	SFLS	PARAM				
Type	I	F	F	F				
Default	required	required	required	option=2,6				

VARIABLE

DESCRIPTION

OPTION

Response:

EQ.1: slave nodes in contact and which come into contact will permanently stick. Tangential motion is inhibited.

EQ.2: tiebreak is active for nodes which are initially in contact. Until failure, tangential motion is inhibited. If PARAM is set to unity, (1.0) shell thickness offsets are ignored, and the orientation of the shell surfaces is required such that the outward normals point to the opposing contact surface.

EQ.3: as 1 above but with failure after sticking.

EQ.4: tiebreak is active for nodes which are initially in contact but tangential motion with frictional sliding is permitted.

EQ.5: tiebreak is active for nodes which are initially in contact. Stress is limited by the yield condition described in remark 5 below. Damage is a function of the crack width opening. The damage function is defined by a load curve which starts at unity for a crack width of zero and decays in some way to zero at a given value of the crack opening.

This interface can be used to represent deformable glue bonds.

<u>VARIABLE</u>	<u>DESCRIPTION</u>
	EQ.6: This option is for use with solids and thick shells only. Tiebreak is active for nodes which are initially in contact. Failure stress must be defined for tiebreak to occur. After the failure stress tiebreak criterion is met, damage is a linear function of the (maximum over time) distance C between points initially in contact. When the distance is equal to PARAM damage is fully developed and interface failure occurs. After failure, this contact option behaves as a surface to surface contact.
NFLS	Normal failure stress for OPTION=2, 3, 4, or 6. For OPTION=5 NFLS becomes the plastic yield stress as defined in remark 5.
SFLS	Shear failure stress for OPTION=2, 3, or 6. For OPTION=4, SFLS is a frictional stress limit if PARAM=1. This frictional stress limit is independent of the normal force at the tie. For OPTION=5 SFLS becomes the load curve ID of the damage model.
PARAM	For OPTION=2, setting PARAM=1 causes the shell thickness offsets to be ignored. For OPTION=4, setting PARAM=1 causes SFLS to be a frictional stress limit. For OPTION=6, PARAM is the critical distance at which the interface failure is complete.

Remarks:

1. After failure, this contact option behaves as a surface-to-surface contact with thickness offsets. After failure, no interface tension is possible.
2. The soft constraint option with SOFT=2 is not implemented for the tiebreak option.
3. For options 2, 3, and 6 the tiebreak failure criterion has normal and shear components:

$$\left(\frac{|\sigma_n|}{NFLS}\right)^2 + \left(\frac{|\sigma_s|}{SFLS}\right)^2 \geq 1$$

4. For option 4, the tiebreak failure criterion has only a normal stress component:

$$\frac{|\sigma_n|}{NFLS} \geq 1$$

5. For option 5, the stress is limited by a perfectly plastic yield condition. For ties in tension, the yield condition is

$$\frac{\sqrt{\sigma_n^2 + 3|\sigma_s|^2}}{NFLS} \leq 1$$

For ties in compression, the yield condition is

*CONTACT

$$\frac{\sqrt{3|\sigma_s|^2}}{NLFS} \leq 1$$

The stress is also scaled by the damage function which is obtained from the load curve. For ties in tension, both normal and shear stress are scaled. For ties in compression, only shear stress is scaled.

6. For option 6, damage initiates when the stress meets the failure criterion. The stress is then scaled by the damage function. Assuming no load reversals, the energy released due to the failure of the interface is approximately $0.5 * S * CCRIT$, where

$$S = \sqrt{\max(\sigma_n, 0)^2 + |\sigma_s|^2}$$

at initiation of damage. This interface may be used for simulating crack propagation. For the energy release to be correct, the contact penalty stiffness must be much larger than

$$\frac{MIN(NFLF, SFLS)}{CCRIT}$$

This Card 4 is mandatory for:

***CONTACT_CONSTRAINT_NODES_TO_SURFACE
*CONTACT_CONSTRAINT_SURFACE_TO_SURFACE**

Card 4 1 2 3 4 5 6 7 8

Variable	KPF								
Type	F								
Default	0.0								

VARIABLE

DESCRIPTION

KPF

Kinematic partition factor for constraint:
 EQ. 0.0: fully symmetric treatment.
 EQ. 1.0: one way treatment with slave nodes constrained to master surface. Only the slave nodes are checked against contact.
 EQ. -1.0: one way treatment with master nodes constrained to slave surface. Only the master nodes are checked against contact.

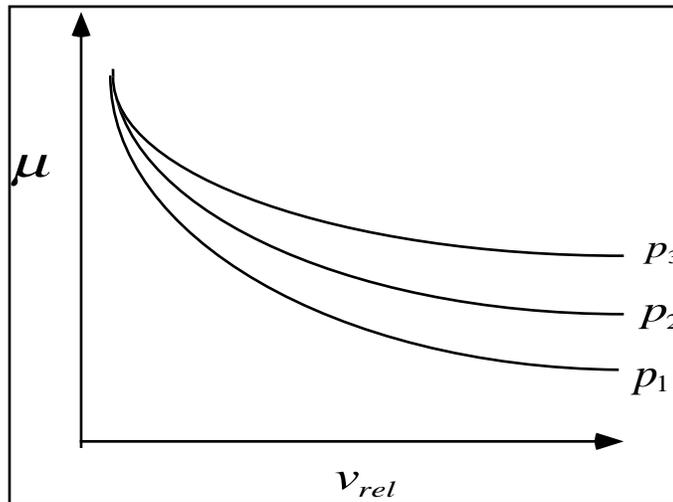


Figure 6.1. Friction coefficient, μ , can be a function of relative velocity and pressure. Specify a flag for the static coefficient of friction, FS, and a table ID for the dynamic coefficient. This option only works with SURFACE_TO_SURFACE and ONE_WAY_SURFACE_TO_SURFACE with thickness offsets.

*CONTACT

This Card 4 is mandatory for:

*CONTACT_DRAWBEAD
 *CONTACT_DRAWBEAD_INITIALIZE

Card 4 1 2 3 4 5 6 7 8

Variable	LCIDRF	LCIDNF	DBDTH	DFSCL	NUMINT	DBPID	ELOFF	
Type	I	I	F	F	I	I	I	
Default	required	none	0.0	1.0	0	0	0	

If the option INITIALIZE is active, then define the following card to initialize the plastic strain and thickness of elements that pass under the drawbead.

Optional 1 2 3 4 5 6 7 8

Variable	LCEPS	TSCALE	LCEPS2	OFFSET				
Type	I	F	I	F				
Default	required	1.0	optional	optional				

VARIABLE

DESCRIPTION

LCIDRF

If LCIDRF is positive then it defines the load curve ID giving the bending component of the restraining force, $F_{bending}$, per unit draw bead length as a function of displacement, δ , see Figure 6.2. This force is due to the bending and unbending of the blank as it moves through the drawbead. The total restraining force is the sum of the bending and friction components.

If LCIDRF is negative then the absolute value gives the load curve ID defining max bead force versus normalized drawbead length. The abscissa values is between zero and 1 and is the normalized drawbead length. The ordinate gives the maximum allowed drawbead retaining force when the bead is in the fully closed position. If the drawbead is not fully closed linear interpolation is used to compute the drawbead force.

VARIABLE	DESCRIPTION
LCIDNF	Load curve ID giving the normal force per unit draw bead length as a function of displacement, δ , see Figure 6.2. This force is due to the bending of the blank into the draw bead as the binder closes on the die and represents a limiting value. The normal force begins to develop when the distance between the die and binder is less than the draw bead depth. <i>As the binder and die close on the blank this force should diminish or reach a plateau</i> , see the explanation below.
DBDTH	Draw bead depth, see Figure 6.2. Necessary to determine correct δ displacement from contact displacements.
DFSCCL	Scale factor for load curve. Default=1.0. This factor scales load curve ID, LCIDRF above.
NUMINT	Number of equally spaced integration points along the draw bead: EQ.0: Internally calculated based on element size of elements that interact with draw bead. This is necessary for the correct calculation of the restraining forces. More integration points may increase the accuracy since the force is applied more evenly along the bead.
DBPID	Optional part ID for the automatically generated truss elements for the draw bead display in the post-processor. If undefined LS-DYNA assigns a unique part ID.
ELOFF	Option to specify and element ID offset for the truss elements that are automatically generated for the draw bead display. If undefined LS-DYNA chooses a unique offset.
LCEPS	Load curve ID defining the plastic strain versus the parametric coordinate through the shell thickness. The parametric coordinate should be defined in the interval between -1 and 1 inclusive. The value of plastic strain at the integration point is interpolated from this load curve. If the plastic strain at an integration point exceeds the value of the load curve at the time initialization occurs, the plastic strain at the point will remain unchanged.
TSCALE	Scale factor that multiplies the shell thickness as the shell element moves under the draw bead.
LCEPS2	Optional load curve ID defining the plastic strain versus the parametric coordinate through the shell thickness, which is used after an element has traveled a distance equal to OFFSET. The parametric coordinate should be defined in the interval between -1 and 1 inclusive. The value of plastic strain at the integration point is interpolated from this load curve. If the plastic strain at an integration point exceeds the value of the load curve at the time initialization occurs, the plastic strain at the point will remain unchanged. Input parameters LCEPS2 and OFFSET provides a way to model the case where a material moves under two draw beads. In this latter case the curve would be the sum of the plastic strains generate by moving under two consecutive beads.

*CONTACT

VARIABLE	DESCRIPTION
OFFSET	If the center of an element has moved a distance equal to OFFSET, the load curve ID, LCEPS2 is used to reinitialize the plastic strain. The TSCALE scale factor is also applied.

Remarks:

The draw bead is defined by a *consecutive* list of *slave* nodes that lie along the draw bead. For straight draw beads only two nodes need to be defined, i.e., one at each end, but for curved beads sufficient nodes are required to define the curvature of the bead geometry. The integration points along the bead are equally spaced and are independent of the nodal spacing used in the definition of the draw bead. By using the capability of tying extra nodes to rigid bodies (see *CONSTRAINED_EXTRA_NODES_OPTION) the draw bead nodal points do not need to belong to the element connectivities of the die and binder. The blank makes up the master surface. IT IS HIGHLY RECOMMENDED TO DEFINE A BOXID AROUND THE DRAWBEAD TO LIMIT THE SIZE OF THE MASTER SURFACE CONSIDERED FOR THE DRAW BEAD. THIS WILL SUBSTANTIALLY REDUCE COST AND MEMORY REQUIREMENTS.

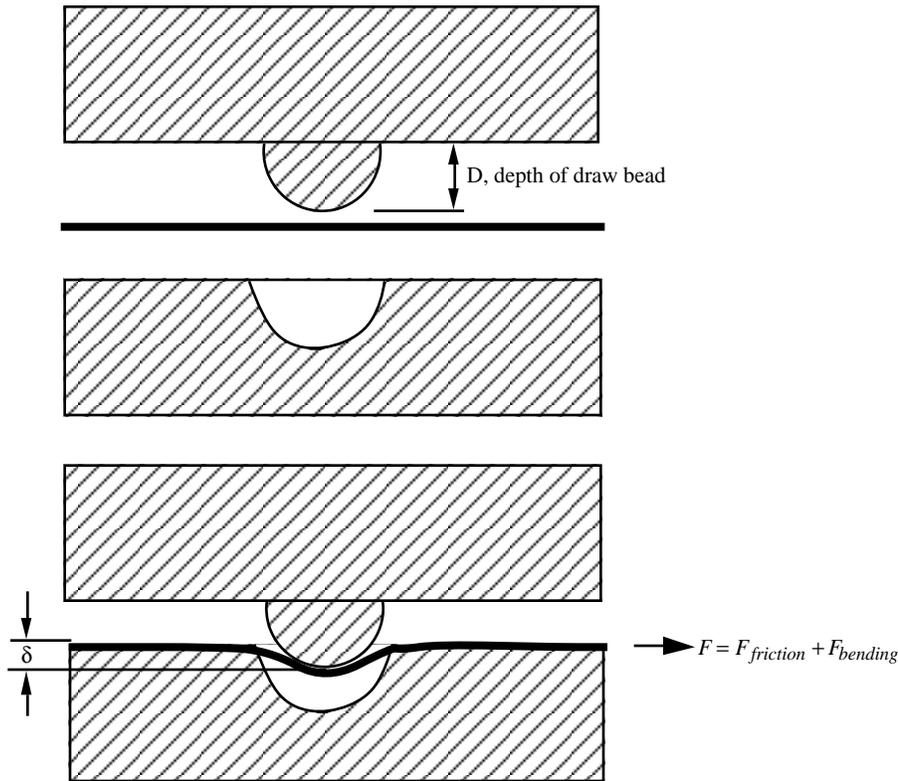


Figure 6.2. Draw bead contact model defines a resisting force as a function of draw bead displacement. The friction force is computed from the normal force in the draw bead and the given friction coefficient.

This Card 4 is mandatory for:

*CONTACT_ ERODING_NODES_TO_SURFACE
*CONTACT_ ERODING_SINGLE_SURFACE
*CONTACT_ ERODING_SURFACE_TO_SURFACE

Card 4 1 2 3 4 5 6 7 8

Variable	ISYM	EROSOP	IADJ					
Type	I	I	I					
Default	0	0	0					

VARIABLE

DESCRIPTION

ISYM

Symmetry plane option:
EQ.0: off,
EQ.1: do not include faces with normal boundary constraints (e.g., segments of brick elements on a symmetry plane).
This option is important to retain the correct boundary conditions in the model with symmetry.

EROSOP

Erosion/Interior node option:
EQ.0: only exterior boundary information is saved,
EQ.1: storage is allocated so that eroding contact can occur. Otherwise, no contact is assumed after erosion of the corresponding element.

IADJ

Adjacent material treatment for solid elements:
EQ.0: solid element faces are included only for free boundaries,
EQ.1: solid element faces are included if they are on the boundary of the material subset. This option also allows the erosion within a body and the subsequent treatment of contact.

Remarks:

Eroding contact may control the timestep (see ECDT in *CONTROL_CONTACT).

For ERODING_NODES_TO_SURFACE, define the slave side using a node set, not a part ID or part set ID.

*CONTACT

This Card 4 is mandatory for:

- *CONTACT_NODES_TO_SURFACE_INTERFERENCE
- *CONTACT_ONE_WAY_SURFACE_TO_SURFACE_INTERFERENCE
- *CONTACT_SURFACE_TO_SURFACE_INTERFERENCE

Purpose: This contact option provides of means of modeling parts which are shrink fitted together and are, therefore, prestressed in the initial configuration. This option turns off the nodal interpenetration checks (which changes the geometry by moving the nodes to eliminate the interpenetration) at the start of the simulation and allows the contact forces to develop to remove the interpenetrations. The load curves defined in this section scale the interface stiffness constants such that the stiffness can increase slowly from zero to a final value with effect that the interface forces also increase gradually to remove the overlaps.

Card 4 1 2 3 4 5 6 7 8

Variable	LCID1	LCID2						
Type	I	I						
Default	0	0						

VARIABLE	DESCRIPTION
LCID1	Load curve ID which scales the interface stiffness during dynamic relaxation. This curve must originate at (0,0) at time=0 and gradually increase.
LCID2	Load curve ID which scales the interface stiffness during the transient calculation. This curve is generally has a constant value of unity for the duration of the calculation if LCID1 is defined. If LCID1=0, this curve must originate at (0,0) at time=0 and gradually increase to a constant value.

Remarks:

Extreme caution must be used with this option. First, shell thickness offsets are taken into account for deformable shell elements. Furthermore, SEGMENT ORIENTATION FOR SHELL ELEMENTS AND INTERPENETRATION CHECKS ARE SKIPPED. Therefore, it is necessary in the problem setup to ensure that all contact segments which belong to shell elements are properly oriented, i.e., the outward normal vector of the segment based on the right hand rule relative to the segment numbering, must point to the opposing contact surface; consequently, automatic contact generation should be avoided for parts composed of shell elements unless automatic generation is used on the slave side of a nodes to surface interface.

This Card 4 is mandatory for:

***CONTACT_RIGID_NODES_TO_RIGID_BODY**

***CONTACT_RIGID_BODY_ONE_WAY_TO_RIGID_BODY**

***CONTACT_RIGID_BODY_TWO_WAY_TO_RIGID_BODY**

Card 4 1 2 3 4 5 6 7 8

Variable	LCID	FCM	US					
Type	I	I	F					
Default	required	required	from LCID					

VARIABLE

DESCRIPTION

LCID	Load curve ID giving force versus penetration behavior for RIGID_ contact. See also the definition of FCM below.
FCM	Force calculation method for RIGID_contact: EQ.1: Load curve gives total normal force on surface versus maximum penetration of any node (RIGID_BODY_ONE_WAY only). EQ.2: Load curve gives normal force on each node versus penetration of node through the surface (all RIGID_contact types). EQ.3: Load curve gives normal pressure versus penetration of node through the surface (RIGID_BODY_TWO_WAY and RIGID_BODY_ONE_WAY only). EQ.4: Load curve gives total normal force versus maximum soft penetration. In this case the force will be followed based on the original penetration point. (RIGID_BODY_ONE_WAY only).
US	Unloading stiffness for RIGID_contact. The default is to unload along the loading curve. This should be equal to or greater than the maximum slope used in the loading curve.

*CONTACT

This Card 4 is mandatory for:

***CONTACT_TIEBREAK_NODES_TO_SURFACE and**

***CONTACT_TIEBREAK_NODES_ONLY**

Card 4 1 2 3 4 5 6 7 8

Variable	NFLF	SFLF	NEN	MES				
Type	F	F	F	F				
Default	required	required	2.	2.				

VARIABLE	DESCRIPTION
-----------------	--------------------

- NFLF Normal failure force. Only tensile failure, i.e., tensile normal forces, will be considered in the failure criterion.
- SFLF Shear failure force
- NEN Exponent for normal force
- MES Exponent for shear force. Failure criterion:

$$\left(\frac{|f_n|}{NFLF} \right)^{NEN} + \left(\frac{|f_s|}{SFLF} \right)^{MES} \geq 1.$$

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.

Both NFLF and SFLF must be defined. If failure in only tension or shear is required then set the other failure force to a large value (1E+10).

After failure, the contact_tiebreak_nodes_to_surface behaves as a nodes-to-surface contact with no thickness offsets (no interface tension possible) whereas the contact_tiebreak_nodes_only stops acting altogether. Prior to failure, the two contact types behave identically.

This Card 4 is mandatory for:

***CONTACT_ TIEBREAK_SURFACE_TO_SURFACE**

Card 4 1 2 3 4 5 6 7 8

Variable	NFLS	SFLS	TBLCID					
Type	F	F	I					
Default	required	required	0					

VARIABLE

DESCRIPTION

NFLS Tensile failure stress. See remark below.

SFLS Shear failure stress. Failure criterion:

$$\left(\frac{|\sigma_n|}{NFLS} \right)^2 + \left(\frac{|\sigma_s|}{SFLS} \right)^2 \geq 1.$$

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.

Both NFLS and SFLS must be defined. If failure in only tension or shear is required then set the other failure stress to a large value (1E+10). When used with shells, contact segment normals are used to establish the tension direction (as opposed to compression). Compressive stress does not contribute to the failure equation.

After failure, this contact option behaves as a surface-to-surface contact with no thickness offsets. After failure, no interface tension is possible.

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

VARIABLE

DESCRIPTION

CF

Thermal conductivity (k) of fluid between the slide surfaces. If a gap with a thickness l_{gap} exists between the slide surfaces, then the conductance due to thermal conductivity between the slide surfaces is

$$h_{cond} = \frac{k}{l_{gap}}$$

Note that LS- DYNA calculates l_{gap} based on deformation.

FRAD

Radiation factor, f , between the slide surfaces. A radiant-heat-transfer coefficient (h_{rad}) is calculated (see *BOUNDARY_RADIATION). If a gap exists between the slide surfaces, then the contact conductance is calculated by

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

HTC

Heat transfer conductance (h_{cont}) for closed gaps. Use this heat transfer conductance for gaps in the range

$$0 \leq l_{gap} \leq l_{min}$$

where l_{min} is GCRIT defined below.

GCRIT

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

<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:

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

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

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

*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	SOFSCS	LCIDAB	MAXPAR	SBOPT	DEPTH	BSORT	FRCFRQ
Type	I	F	I	F	F	I	I	I
Default	0	.1	0	1.025.	0.	2	10-100	1
Remarks			type a13			type 13		

VARIABLE

DESCRIPTION

SOFT

Soft constraint option:

EQ.0: penalty formulation,

EQ.1: soft constraint formulation,

EQ.2: pinball segment-based contact.

EQ.4: constraint approach for FORMING contact option.

The soft constraint may be necessary if the material constants of the elements which make up the surfaces in contact have a wide variation in the elastic bulk moduli. In the soft constraint option, the interface stiffness is based on the nodal mass and the global time step size. This method of computing the interface stiffness will typically give much higher stiffness value than would be obtained by using the bulk modulus; therefore, this method the preferred approach when soft foam materials interact with metals. See the remark below for the segment-based penalty formulation.

SOFSCS

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 for most contact. options. Other defaults are:

EQ.1.006:SPOTWELD and

EQ.1.006:TIED_SHELL_..._CONSTRAINED_OFFSET,

EQ.1.006:TIED_SHELL_..._OFFSET

EQ.1.006:TIED_SHELL_..._:BEAM_OFFSET.

EQ.1.100:AUTOMATIC_GENERAL

Optional Card A (continued)

<u>VARIABLE</u>	<u>DESCRIPTION</u>
	This factor allows an increase in the size of the segments which may be useful at sharp corners. For the SPOTWELD and ..._OFFSET options larger values can sometimes lead to numerical instabilities; however, a larger value is sometimes necessary to ensure that all nodes of interest are tied.
SBOPT	Segment-based contact options (SOFT=2). EQ.0: defaults to 2. EQ.1: pinball edge-edge contact (not recommended) EQ.2: assume planer segments (default) EQ.3: warped segment checking EQ.4: sliding option EQ.5: do options 3 and 4
DEPTH	Search depth in automatic contact. Value of 1 is sufficiently accurate for most crash applications and is much less expensive. LS-DYNA for improved accuracy sets this value to 2. If zero, the default is set to 2. LT.0: DEPTH is the load curve ID defining searching depth versus time. See remarks below for segment-based contact options controlled by DEPTH.
BSORT	Number of cycles between bucket sorts. Values of 25 and 100 are recommended for contact types 4 and 13 (SINGLE_SURFACE), respectively. Values of 10-15 are okay for the surface to surface and node to surface contact. If zero, LS-DYNA determines the interval. LT.0: BSORT load curve ID defining bucket sorting frequency versus time.
FRCFRQ	Number of cycles between contact force updates for penalty contact formulations. This option can provide a significant speed-up of the contact treatment. If used, values exceeding 3 or 4 are dangerous. Considerable care must be exercised when using this option, as this option assumes that contact does not change FRCFRG cycles. EQ.0: FRCFRG is set to 1 and force calculations are performed each cycle-strongly recommended.

Remark:

Setting SOFT=1 or 2 on optional contact card A will cause the contact stiffness to be determined based on stability considerations, taking into account the time step and nodal masses. This approach is generally more effective for contact between materials of dissimilar stiffness or dissimilar mesh densities.

SOFT=2 is for general shell and solid element contact. This option is available for all SURFACE_TO_SURFACE, ONE_WAY_SURFACE_TO_SURFACE, and SINGLE_SURFACE options including eroding and airbag contact. When the AUTOMATIC option is used, orientation of shell segment normals is automatic. When the AUTOMATIC option is not used, the segment or element orientations are used as input. The segment-based penalty formulation contact algorithm checks for segments vs. segment penetration rather than node vs. segment. After penetrating segments are found, an automatic judgment is made as to which is the master segment, and penalty

*CONTACT

forces are applied normal to that segment. The user may override this automatic judgment by using the ONE_WAY options in which case the master segment normals are used as input by the user. All parameters on the first three cards are active except for VC, and VSF. On optional card A, some parameters have different meanings than they do for the default contact.

For SOFT=2, the SBOPT parameter on optional card A controls several options. The pinball edge to edge checking is not recommended and is included only for back compatibility. For edge to edge checking setting DEPTH=5 is recommended instead (see below). The warped segment option more accurately checks for penetration of warped surfaces. The sliding option uses neighbor segment information to improve sliding behavior. It is primarily useful for preventing segments from incorrectly catching nodes on a sliding surface.

For SOFT=2, the DEPTH parameter controls several additional options for segment based contact. When DEPTH=2 (default), surface penetrations measured at nodes are checked. When DEPTH=3, surface penetration may also be measured at the edge. This options is useful mainly for airbags. When DEPTH=5, both surface penetrations and edge to edge penetration is checked. The airbag contact has two additional options, DEPTH=1 and 4. DEPTH=4 activates additional airbag logic that uses neighbor segment information when judging if contact is between interior or exterior airbag surfaces. This option is not recommended and is maintained only for backward compatibility. Setting DEPTH=1 suppresses all airbag logic.

For SOFT=2 contact, the MAXPAR has a totally different use. Positive values of MAXPAR are ignored. If a negative value is input for MAXPAR, the absolute value of MAXPAR will be used as an assumed time step for scaling the contact stiffness. This option is useful for maintaining consistent contact behavior of an airbag deployment when a validated airbag is inserted into an automobile model. For the new run, setting MAXPAR=the negative of the solution time step of the validated run will cause the airbag contact stiffness to be unchanged in the new run even if the solution time step of the new run is smaller.

For SOFT=2 contact, only the ISYM, I2D3D, SLDTHK, and SLDSTF parameters are active on optional card B.

Optional Card B

Reminder: If Optional Card B is used, then Optional Card A must be defined. (Optional Card A may be a blank line).

Optional Card B	1	2	3	4	5	6	7	8
Variable	PENMAX	THKOPT	SHLTHK	SNLOG	ISYM	I2D3D	SLDTHK	SLDSTF
Type	F	I	I	I	I	I	F	F
Default	0	0	0	0	0	0	0	0
Remarks		Old types 3, 5, 10	Old types 3, 5, 10					

VARIABLE

DESCRIPTION

PENMAX

Maximum penetration distance for old type 3, 5, 8, 9, and 10 contact or the segment thickness multiplied by PENMAX defines the maximum penetration allowed (as a multiple of the segment thickness) for contact types a 3, a 5, a10, 13, 15, and 26. (see discussion at end of section, including Table 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.

*CONTACT

<u>VARIABLE</u>	<u>DESCRIPTION</u>
SNLOG	<p>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.</p> <p>EQ.0: logic is enabled (default), EQ.1: logic is skipped (sometimes recommended for metalforming calculations or for contact involving foam materials).</p>
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).</p> <p>This option is important to retain the correct boundary conditions in the model with symmetry. For the <code>_ERODING_</code> contacts this option may also be defined on card 4.</p>
I2D3D	<p>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.</p>
SLDTHK	<p>Optional solid element thickness. A nonzero positive value will activate the contact thickness offsets in the contact algorithms where offsets apply. The contact treatment will then be equivalent to the case where null shell elements are used to cover the brick elements. The contact stiffness parameter below, <code>SLDSTF</code>, may also be used to override the default value.</p>
SLDSTF	<p>Optional solid element stiffness. A nonzero positive value overrides the bulk modulus taken from the material model referenced by the solid element.</p>

Optional Card C

Reminder: If Optional Card C is used, then Optional Cards A and B must be defined. (Optional Cards A and B may be blank lines).

Optional Card C	1	2	3	4	5	6	7	8
Variable	IGAP	IGNORE						
Type	I	I						
Default	1	0						
Remarks								

VARIABLE

DESCRIPTION

IGAP

Flag to improve implicit convergence behavior at the expense of creating some sticking if parts attempt to separate. This option is not available for AUTOMATIC contact types. (IMPLICIT ONLY)
EQ. 1: apply method to improve convergence (DEFAULT)
EQ. 2: do not apply method

IGNORE

Ignore initial penetrations in the *CONTACT_AUTOMATIC options.
EQ.0: Take the default value from the fourth card of the CONTROL_CONTACT input.
EQ.1: Allow initial penetrations to exist by tracking the initial penetrations.
EQ.2: Move nodes to eliminate initial penetrations in the model definition.

*CONTACT

General Remarks on *CONTACT:

1. Modeling airbag interactions with structures and occupants using the actual fabric thickness, which is approximate 0.30 mm, may result in a contact breakdown that leads to inconsistent occupant behavior between different machines. Based on our experience, using a two-way automatic type contact definition, i.e., AUTOMATIC_SURFACE_TO_SURFACE, between any airbag to structure/occupant interaction and setting the airbag fabric contact thickness to at least 10 times the actual fabric thickness has helped improved contact behavior and eliminates the machine inconsistencies. Due to a large stiffness difference between the airbag and the interacting materials, the soft constraint option (SOFT=1) or the segment based pinball option (SOFT=2) is recommended. It must be noted that with the above contact definition, only the airbag materials should be included in any *AIRBAG_SINGLE_SURFACE definitions to avoid duplicate contact treatment that can lead to numerical instabilities.
2. TIED_NODES_TO_SURFACE
TIED_SHELL_EDGE_TO_SURFACE
SPOTWELD
SPOTWELD_WITH_TORSION
TIED_SURFACE_TO_SURFACE

These contact definitions are based on constraint equations and will not work with rigid bodies. However, tied interfaces with the offset option can be used with rigid bodies, i.e.,

TIED_NODES_TO_SURFACE_OFFSET
TIED_SHELL_EDGE_TO_SURFACE_OFFSET
TIED_SHELL_EDGE_TO_SURFACE_BEAM_OFFSET
TIED_SURFACE_TO_SURFACE_OFFSET
TIED_SURFACE_TO_SURFACE_BEAM_OFFSET

Also, it may sometimes be advantageous to use the CONSTRAINED_EXTRA_NODE_OPTION instead for tying deformable nodes to rigid bodies since in this latter case the tied nodes may be an arbitrary distance away from the rigid body.

Tying will only work if the surfaces are near each other. The criteria used to determine whether a slave node is tied down is that it must be "close". For shell elements "close" is defined as distance, δ , less than:

$$\delta_1 = 0.60 * (\text{thickness_slave_node} + \text{thickness_master_segment})$$

$$\delta_2 = 0.05 * \min(\text{master_segment_diagonals})$$

$$\delta = \max(\delta_1, \delta_2)$$

If a node is further away it will not be tied and a warning message will be printed. For solid elements the slave node thickness is zero; otherwise, the same procedure is used.

If there is a large difference in element areas between the master and slave side, the distance, δ_2 , may be too large and may cause the unexpected projection of nodes that should not be tied. This can occur during calculation when adaptive remeshing is used. To avoid this difficulty the slave and master thickness can be specified as negative values on Card 3 in which case

$$\delta = \text{abs}(\delta_1)$$

3. The contact algorithm for tying spotwelds with torsion, SPOTWELD_WITH_TORSION, must be used with care. Parts that are tied by this option should be subjected to stiffness proportional damping of approximately ten percent, i.e., input a coefficient of 0.10. This can be defined for each part on the *DAMPING_PART_STIFFNESS input. Stability problems may arise with this option if damping is not used.

4. CONSTRAINT_NODES_TO_SURFACE
CONSTRAINT_SURFACE_TO_SURFACE

These contact definitions must be used with care. The surface and the nodes which are constrained to a surface are not allowed to be used in any other CONSTRAINT_... contact definition. If, however, contact has to be defined from both sides as in 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.

5. AIRBAG_SINGLE_SURFACE
AUTOMATIC_GENERAL
AUTOMATIC_GENERAL_INTERIOR
AUTOMATIC_NODES_TO_SURFACE
AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE
AUTOMATIC_SINGLE_SURFACE
AUTOMATIC_SURFACE_TO_SURFACE
SINGLE_SURFACE

These contact definitions require thickness to be taken into account for rigid bodies modeled with shell elements. Therefore, care should be taken to ensure that realistic thicknesses are specified for the rigid body shells. A thickness that is too small may result in loss of contact and an unrealistically large thickness may result in a degradation in speed during the bucket sorts as well as nonphysical behavior. The SHLTHK option on the *CONTROL_CONTACT card is ignored for these contact types.

6. Two methods are used in LS-DYNA for projecting the contact surface to account for shell thicknesses. The choice of methods can influence the accuracy and cost of the calculation. Segment based projection is used in contact types:

AIRBAG_SINGLE_SURFACE
AUTOMATIC_GENERAL
AUTOMATIC_NODES_TO_SURFACE
AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE
AUTOMATIC_SINGLE_SURFACE
AUTOMATIC_SURFACE_TO_SURFACE
FORMING_NODES_TO_SURFACE
FORMING_ONE_WAY_SURFACE_TO_SURFACE
FORMING_SURFACE_TO_SURFACE

The remaining contact types use nodal normal projections if projections are used. The main advantage of nodal projections is that a continuous contact surface is obtained which is much more accurate in applications such as metal forming. The disadvantages of nodal projections are the higher costs due to the nodal normal calculations, difficulties in treating T-intersections and other geometric complications, and the need for consistent orientation of contact surface segments. The contact type:

SINGLE_SURFACE

*CONTACT

uses nodal normal projections and consequently is slower than the alternatives.

7. FORCE_TRANSDUCER_PENALTY FORCE_TRANSDUCER_CONSTRAINT

This contact allows the total contact forces applied by all contacts to be picked up. This contact does not apply any force to the model. 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.

8. FORMING_...

These contacts are mainly used for metal forming applications. A connected mesh is not required for the master (tooling) side but the orientation of the mesh **must** be in the same direction. These contact types are based on the `AUTOMATIC` type contacts and consequently the performance is better than the original two surface contacts.

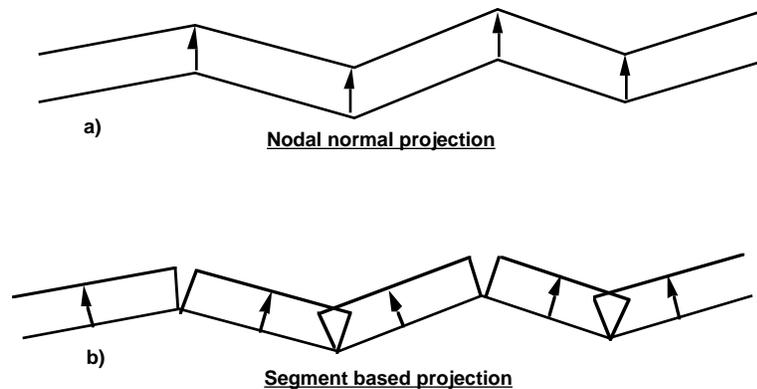


Figure 6.3. Nodal normal and segment based projection is used in the contact options.

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	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, 95X type numbers are:

STRUCTURED INPUT TYPE ID	KEYWORD NAME
a13	AIRBAG_SINGLE_SURFACE
26	AUTOMATIC_GENERAL
i26	AUTOMATIC_GENERAL_INTERIOR
a 5	AUTOMATIC_NODES_TO_SURFACE
a 5	AUTOMATIC_NODES_TO_SURFACE_TIEBREAK
a10	AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE
13	AUTOMATIC_SINGLE_SURFACE
a 3	AUTOMATIC_SURFACE_TO_SURFACE
a 3	AUTOMATIC_SURFACE_TO_SURFACE_TIEBREAK
18	CONSTRAINT_NODES_TO_SURFACE
17	CONSTRAINT_SURFACE_TO_SURFACE
23	DRAWBEAD
16	ERODING_NODES_TO_SURFACE
14	ERODING_SURFACE_TO_SURFACE
15	ERODING_SINGLE_SURFACE
27	FORCE_TRANSDUCER_CONSTRAINT
25	FORCE_TRANSDUCER_PENALTY
m 5	FORMING_NODES_TO_SURFACE
m10	FORMING_ONE_WAY_SURFACE_TO_SURFACE
m 3	FORMING_SURFACE_TO_SURFACE
5	NODES_TO_SURFACE
5	NODES_TO_SURFACE_INTERFERENCE
10	ONE_WAY_SURFACE_TO_SURFACE
20	RIGID_NODES_TO_RIGID_BODY
21	RIGID_BODY_ONE_WAY_TO_RIGID_BODY
19	RIGID_BODY_TWO_WAY_TO_RIGID_BODY
22	SINGLE_EDGE
4	SINGLE_SURFACE
1	SLIDING_ONLY
p 1	SLIDING_ONLY_PENALTY
3	SURFACE_TO_SURFACE
3	SURFACE_TO_SURFACE_INTERFERENCE

STRUCTURED INPUT TYPE ID	KEYWORD NAME
8	TIEBREAK_NODES_TO_SURFACE
9	TIEBREAK_SURFACE_TO_SURFACE
6	TIED_NODES_TO_SURFACE
o 6	TIED_NODES_TO_SURFACE_OFFSET
7	TIED_SHELL_EDGE_TO_SURFACE
7	SPOTWELD
s 7	SPOTWELD_WITH_TORSION
2	TIED_SURFACE_TO_SURFACE
o 2	TIED_SURFACE_TO_SURFACE_OFFSET

*CONTACT_AUTO_MOVE

Purpose: Move the master surface in a contact definition to close an initial gap between the slave and master surfaces. The contact surfaces will then start in contact thereby saving calculational cost. The master surface in metalforming applications will typically be the punch and the blank will be the slave surface.

Define one card. Card Format (I0)

Cards 1 1 2 3 4 5 6 7 8

Variable	ID	CID	VID	LCID	ATIME			
Type	I	I	I	I	F			
Default	required	required	required	0	0.0			

VARIABLE

DESCRIPTION

ID	ID for this auto positioning input.
CID	Contact ID.
VID	Vector ID for a vector oriented in the direction of the movement of the master surface. See *DEFINE_VECTOR. The origin of this vector is unimportant since the direction cosines of the vector are computed and used.
LCID	Optional load curve ID defining velocity versus time. The load curve will be adjusted to account for the movement of the master surface. The load curve should be defined by four points, and its shape should resemble a trapezoid with the longest parallel side along the abscissa. The abscissa is adjusted (shortened) in the flat part of the curve where the velocity is constant to account for the movement.
ATIME	Activation time. At this time the master surface is moved.

*CONTACT

*CONTACT_COUPLING

Purpose: Define a coupling surface for MADYMO to couple LS-DYNA with deformable and rigid parts within MADYMO. In this interface, MADYMO computes the contact forces acting on the coupling surface, and LS-DYNA uses these forces in the update of the motion of the coupling surface for the next time step. Contact coupling can be used with other coupling options in LS-DYNA.

Cards 1 1 2 3 4 5 6 7 8

Variable	ID								
Type	I								
Default	required								

Cards 2, 3, 4, ... Define as cards as necessary. The next "*" card terminates this input.

Cards 2,3,.. 1 2 3 4 5 6 7 8

Variable	SID	STYPE							
Type	I	I							
Default	required	0							

VARIABLE

DESCRIPTION

SID	Set ID for coupling, See remark 1 below.
STYPE	Set type: EQ.0: part set EQ.1: shell element set EQ.2: solid element set EQ.3: thick shell element set

Remarks:

1. Only one coupling surface can be defined. If additional surfaces are defined, the coupling information will be added to the first definition.
2. The units and orientation can be converted by using the CONTROL_COUPLING keyword. It is not necessary to use the same system of units in MADYMO and in LS-DYNA if unit conversion factors are defined.

*CONTACT

*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

<u>VARIABLE</u>	<u>DESCRIPTION</u>
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,

<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.

*CONTACT

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				

<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.
GO	Flag for mesh generation of the contact entity for entity types 1-5 and 10-11. This is used for visualization in post-processing only. EQ.0: mesh is not generated, EQ.1: mesh is generated.

Cards 3 and 4 Format

Card 3 1 2 3 4 5 6 7 8

Variable	XC	YC	ZC	AX	AY	AZ		
Type	F	F	F	F	F	F		
Default	0.	0.	0.	0.	0.	0		

Card 4 1 2 3 4 5 6 7 8

Variable	BX	BY	BZ					
Type	F	F	F					
Default	0.	0.	0.					

VARIABLE**DESCRIPTION**

XC	x-center, x_c , see remarks below.
YC	y-center, y_c , see remarks below.
ZC	z-center, z_c . See remarks below.
AX	x-direction for local axis A, A_x , see remarks below.
AY	y-direction for local axis A, A_y , see remarks below.
AZ	z-direction for local axis A, A_z , see remarks below.
BX	x-direction for local axis B, B_x , see remarks below.
BY	y-direction for local axis B, B_y , see remarks below.
BZ	z-direction for local axis B, B_z , see remarks below.

*CONTACT

Remarks:

1. The coordinates, (x_c, y_c, z_c) , are the positions of the local origin of the geometric entity in global coordinates. The entity's local A-axis is determined by the vector (A_x, A_y, A_z) and the local B-axis by the vector (B_x, B_y, B_z) .
2. Cards 3 and 4 define a local to global transformation. The geometric contact entities are defined in a local system and transformed into the global system. For the ellipsoid, this is necessary because it has a restricted definition for the local position. For the plane, sphere, and cylinder, the entities can be defined in the global system and the transformation becomes $(x_c, y_c, z_c)=(0,0,0)$, $(A_x, A_y, A_z)=(1,0,0)$, and $(B_x, B_y, B_z)=(0,1,0)$.

Card 5 Format

Card 5 1 2 3 4 5 6 7 8

Variable	INOUT	G1	G2	G3	G4	G5	G6	G7
Type	I	F	F	F	F	F	F	F
Default	0	0.	0.	0.	0.	0.	0.	0.

VARIABLE

DESCRIPTION

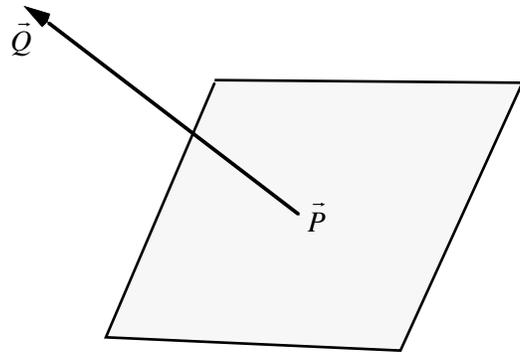
INOUT	In-out flag. Allows contact from the inside or the outside (default) of the entity: EQ.0: slave nodes exist outside of the entity, EQ.1: slave nodes exist inside the entity.
G1	Entity coefficient g_1 (CAL3D/MADYMO plane or ellipse number) for coupled analysis (see Appendix F).
G2	Entity coefficient g_2 , see remarks below.
G3	Entity coefficient g_3 , see remarks below.
G4	Entity coefficient g_4 , see remarks below.
G5	Entity coefficient g_5 , see remarks below.
G6	Entity coefficient g_6 , see remarks below.
G7	Entity coefficient g_7 , see remarks below.

Remarks:

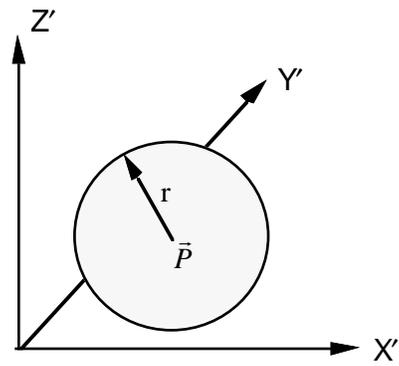
Figures 6.4a and 6.4b show the definitions of the geometric contact entities. The relationships between the entity coefficients and the Figure 6.4a and 6.4b variables are as follows (please note that (P_x, P_y, P_z) is a position vector and that (Q_x, Q_y, Q_z) is a direction vector):

$$\begin{aligned} \text{GEOTYP} = 1: \quad & g_1 = P_x & g_4 & = Q_x \\ & g_2 = P_y & g_5 & = Q_y \\ & g_3 = P_z & g_6 & = Q_z \\ & & g_7 & = L \end{aligned}$$

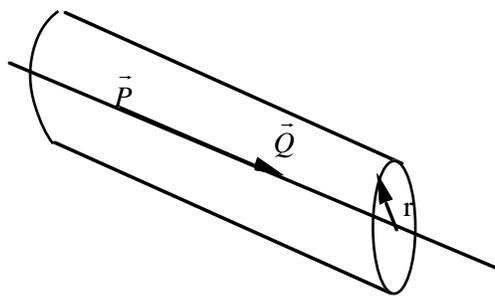
If automatic generation is used, a square plane of length L on each edge is generated which represents the infinite plane. If generation is inactive, then g_7 may be ignored.



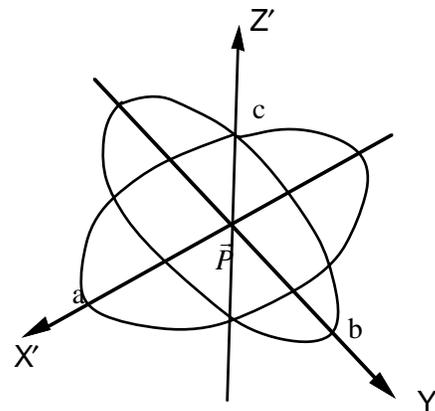
IGTYPE= 1: Infinite Plane



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.

*CONTACT

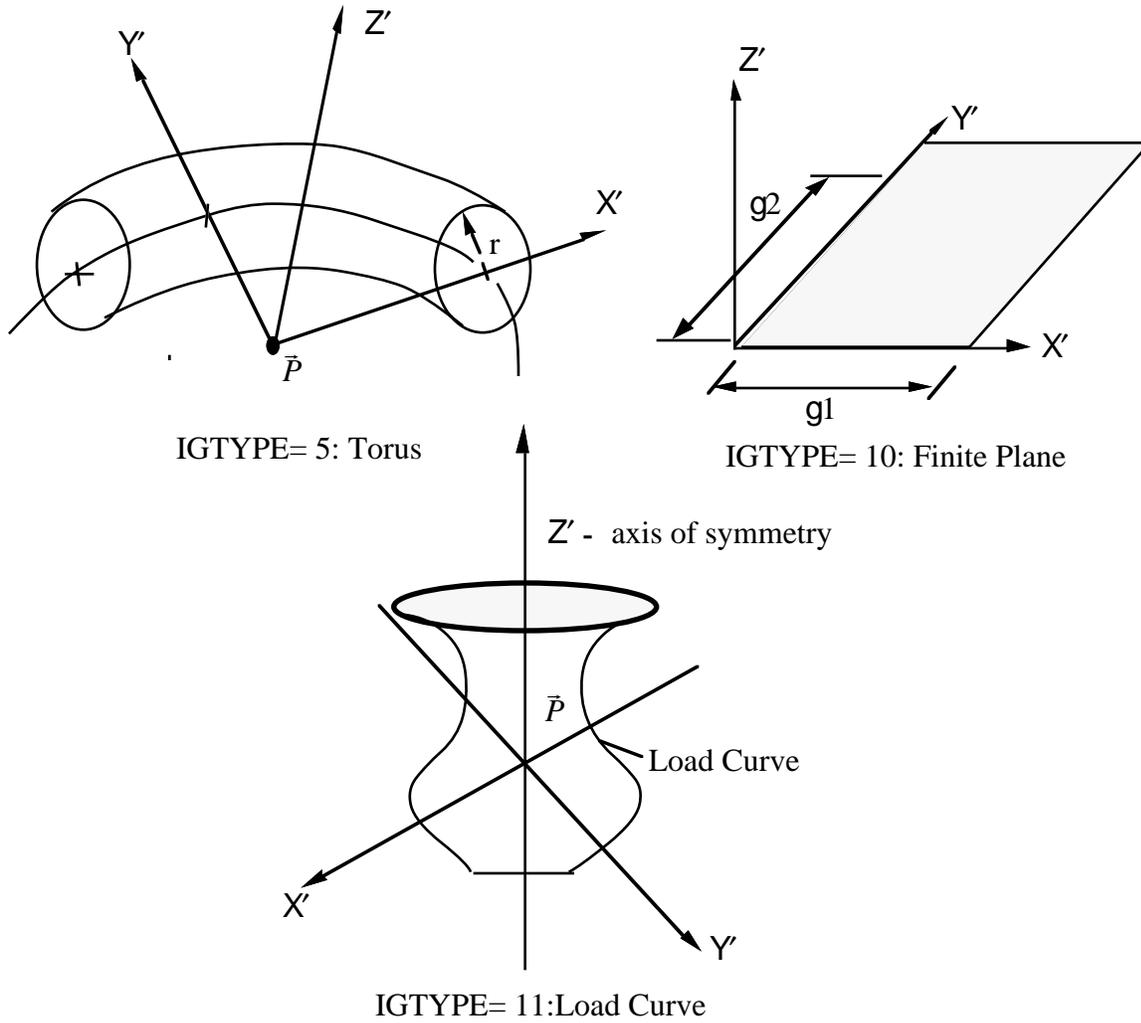


Figure 6.4b. Contact Entities.

*CONTACT_GEBOD_OPTION

Purpose: Define contact interaction between the segment of a GEBOD dummy and parts or nodes of the finite element model. This implementation follows that of the contact entity, however, it is specialized for the dummies. Forces may be output using the *DATABASE_GCEOUT command. See *COMPONENT_GEBOD and Appendix K for further details.

Conventional *CONTACT_OPTION treatment (surface-to-surface, nodes-to-surface, etc.) can also be applied to the segments of a dummy. To use this approach it is first necessary to determine part ID assignments by running the model through LSDYNA's initialization phase.

The following options are available and refer to the ellipsoids which comprise the dummy. Options involving **HAND** are not applicable for the child dummy since its lower arm and hand share a common ellipsoid.

LOWER_TORSO

MIDDLE_TORSO

UPPER_TORSO

NECK

HEAD

LEFT_SHOULDER

RIGHT_SHOULDER

LEFT_UPPER_ARM

RIGHT_UPPER_ARM

LEFT_LOWER_ARM

RIGHT_LOWER_ARM

LEFT_HAND

RIGHT_HAND

LEFT_UPPER_LEG

RIGHT_UPPER_LEG

LEFT_LOWER_LEG

RIGHT_LOWER_LEG

LEFT_FOOT

RIGHT_FOOT

*CONTACT

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	

<u>VARIABLE</u>	<u>DESCRIPTION</u>
DID	Dummy ID, see *COMPONENT_GEBOD_OPTION.
SSID	Slave set ID, see *SET_NODE_OPTION, *PART, or *SET_PART.
SSTYP	Slave set type: EQ.0: node set, EQ.1: part ID, EQ.2: part set ID.
SF	Penalty scale factor. Useful to scale maximized penalty.
DF	Damping option, see description for *CONTACT_OPTION: EQ..0: no damping, GT..0: viscous damping in percent of critical, e.g., 20 for 20% damping, EQ.-n: n is the load curve ID giving the damping force versus relative normal velocity (see remark 1 below).
CF	Coulomb friction coefficient (see remark 2 below). Assumed to be constant.
INTORD	Integration order (slaved materials only). EQ.0: check nodes only, EQ.1: 1 point integration over segments, EQ.2: 2×2 integration, EQ.3: 3×3 integration, EQ.4: 4×4 integration, EQ.5: 5×5 integration. This option allows a check of the penetration of the dummy segment into the deformable (slaved) material. Then virtual nodes at the location of the integration points are checked.

Card 2 Format

Card 2 1 2 3 4 5 6 7 8

Variable	BT	DT	SO					
Type	F	F	I					
Default	0.	1.E+20	0					

VARIABLE

DESCRIPTION

BT	Birth time
DT	Death time
SO	Flag to use penalty stiffness as in surface to surface contact: EQ.0: contact entity stiffness formulation, EQ.1: surface to surface contact method, EQ.-n: n is the load curve ID giving the force versus the normal penetration.

Remarks:

1. The optional load curves that are defined for damping versus relative normal velocity and for force versus normal penetration should be defined in the positive quadrant. The sign for the damping force depends on the direction of the relative velocity and the treatment is symmetric if the damping curve is in the positive quadrant. If the damping force is defined in the negative and positive quadrants, the sign of the relative velocity is used in the table look-up.
2. Insofar as these ellipsoidal contact surfaces are continuous and smooth it may be necessary to specify Coulomb friction values larger than those typically used with faceted contact surfaces.

*CONTACT

*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.

Four attributes should be defined for the part set:

Attribute 1:

PSF, penalty scale factor (Default=1.00).

Attribute 2:

Activation factor, F_a (Default=0.10). When the crushing of the element reaches F_a times the initial thickness the contact algorithm begins to act.

Attribute 3:

ED, Optional modulus for interior contact stiffness.

Attribute 4:

TYPE, Optional modulus for interior contact stiffness.
EQ.1.0: Default, recommended for uniform compression
EQ.2.0: Designed to control the combined modes of shear and compression. Works for type 1 brick formulation.

Remarks:

The interior penalty is determined by the formula:

$$K = \frac{SLSFAC \cdot PSF \cdot Volume^{\frac{2}{3}} \cdot E}{Min. Thickness}$$

where SLSFAC is the value specified on the *CONTROL_CONTACT card , volume is the volume of the brick element, E is a constitutive modulus, and min. thickness is approximately the thickness of the solid element through its thinnest dimension. If ED, is defined above the interior penalty is then given instead by:

$$K = \frac{Volume^{\frac{2}{3}} \cdot ED}{Min. Thickness}$$

where the scaling factors are ignored. Generally, ED should be taken as the locking modulus specified for the foam constitutive model.

Caution should be observed when using this option since if the time step size is too large an instability may result. The time step size is not affected by the use of interior contact.

*CONTACT

*CONTACT_RIGID_SURFACE

Purpose: Define rigid surface contact. The purpose of rigid surface contact is to model large rigid surfaces, e.g., road surfaces, with nodal points and segments that require little storage and are written out at the beginning of the binary databases. The rigid surface motion, which can be optionally prescribed, is defined by a displacement vector which is written with each output state. The nodal points defining the rigid surface must be defined in the *NODE_RIGID_SURFACE section of this manual. These rigid nodal points do not contribute degrees-of-freedom.

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	CID	PSID	BOXID	SSID	FS	FD	DC	VC
Type	I	I	I	I	F	F	F	F
Default	none	none	0	none	0.	0.	0.	0.

Card 2 1 2 3 4 5 6 7 8

Variable	LCIDX	LCIDY	LCIDZ	FSLCID	FDLCID			
Type	I	I	I	I	I			
Default	0	0	0	0	0			

Card 3 1 2 3 4 5 6 7 8

Variable	SFS	STTHK	SFTHK	XPENE	BSORT			
Type	F	F	F	F	I			
Default	1.0	0.0	1.0	4.0	10			

VARIABLE	DESCRIPTION
CID	Contact interface ID. This must be a unique number.
PSID	Part set ID of all parts that may contact the rigid surface. See *SET_PART.
BOXID	Include only nodes of the part set that are within the specified box, see *DEFINE_BOX, in contact. If BOXID is zero, all nodes from the part set, PSID, will be included in the contact.
SSID	Segment set ID defining the rigid surface. See *SET_SEGMENT.
FS	Static coefficient of friction. The frictional coefficient is assumed to be dependent on the relative velocity v_{rel} of the surfaces in contact $\mu_c = FD + (FS - FD)e^{-DC v_{rel} }$. If FSLCID is defined, see below, then FS is overwritten by the value from the load curve.
FD	Dynamic coefficient of friction. The frictional coefficient is assumed to be dependent on the relative velocity v_{rel} of the surfaces in contact $\mu_c = FD + (FS - FD)e^{-DC v_{rel} }$. If FDLCID is defined, see below, then FD is overwritten by the value from the load curve.
DC	Exponential decay coefficient. The frictional coefficient is assumed to be dependent on the relative velocity v_{rel} of the surfaces in contact $\mu_c = FD + (FS - FD)e^{-DC v_{rel} }$.
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 σ_o is the yield stress of the contacted material.
LCIDX	Load curve ID defining x-direction motion. If zero, there is no motion in the x-coordinate system.
LCIDY	Load curve ID defining y-direction motion. If zero, there is no motion in the y-coordinate system.
LCIDZ	Load curve ID defining z-direction motion. If zero, there is no motion in the z-coordinate system.
FSLCID	Load curve ID defining the static coefficient of friction as a function of interface pressure. This option applies to shell segments only.
FDLCID	Load curve ID defining the dynamic coefficient of friction as a function of interface pressure. This option applies to shell segments only.
SFS	Scale factor on default slave penalty stiffness, see also *CONTROL_CONTACT.

*CONTACT

<u>VARIABLE</u>	<u>DESCRIPTION</u>
STTHK	Optional thickness for slave surface (overrides true thickness). This option applies to contact with shell, solid, and beam elements. True thickness is the element thickness of the shell elements. Thickness offsets are not used for solid element unless this option is specified.
SFTHK	Scale factor for slave surface thickness (scales true thickness). This option applies only to contact with shell elements. True thickness is the element thickness of the shell elements.
XPENE	Contact surface maximum penetration check multiplier. If the penetration of a node through the rigid surface exceeds the product of XPENE and the slave node thickness, the node is set free. EQ.0: default is set to 4.0.
BSORT	Number of cycles between bucket sorts. The default value is set to 10 but can be much larger, e.g., 50-100, for fully connected surfaces.

Remarks:

Thickness offsets do not apply to the rigid surface. There is no orientation requirement for the segments in the rigid surface, and the surface may be assembled from disjoint, but contiguous, arbitrarily oriented meshes. With disjoint meshes, the global searches must be done frequently, about every 10 cycles, to ensure a smooth movement of a slave node between mesh patches. For fully connected meshes this frequency interval can be safely set to 50-200 steps between searches.

The modified binary database (D3PLOT) contains the road surface information prior to the state data. This information contains:

NPDS	=	Total number of rigid surface points in problem.
NRSC	=	Total number of rigid surface contact segments summed over all definitions.
NSID	=	Number of rigid surface definitions.
NVELQ	=	Number of words at the end of each binary output state defining the rigid surface motion. This equals 6 x NSID if any rigid surface moves or zero if all rigid surfaces are stationary.
PIDS	=	An array equal in length to NPDS. This array defines the ID for each point in the road surface.
XC	=	An array equal in length to 3 x NPDS. This array defines the global x, y, and z coordinates of each point.

For each road surface define the following NSID sets of data.

ID	=	Rigid surface ID.
NS	=	Number of segments in rigid surface.
IXRS	=	An array equal in length to 4 x NS. This is the connectivity of the rigid surface in the internal numbering system.

At the end of each state, 6 x NVELQ words of information are written. For each road surface the x, y, and z displacements and velocities are written. If the road surface is fixed, a null vector should be output. Skip this section if NVELQ=0. LS-PREPOST currently displays rigid surfaces and animates their motion.

*CONTACT_1D

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

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	NSIDS	NSIDM	ERR	SIGC	GB	SMAX	EXP	
Type	I	I	F	F	F	F	F	
Default	none	none	0.	0.	0.	0.	0.	

VARIABLE

DESCRIPTION

NSIDS	Nodal set ID for the slave nodes, see *SET_NODE.
NSIDM	Nodal set ID for the master nodes, see *SET_NODE.
ERR	External radius of rebar
SIGC	Compressive strength of concrete
GB	Bond shear modulus
SMAX	Maximum shear strain
EXP	Exponent in damage curve

Remarks:

With this option the concrete is defined with solid elements and the rebar with truss elements, each with their own unique set of nodal points. A string of consecutive nodes, called slave nodes, related to the truss elements may slide along a string of consecutive nodes, called master nodes, related to the solid elements. The sliding commences after the rebar debonds.

The bond between the rebar and concrete is assumed to be elastic perfectly plastic. The maximum allowable slip strain is given as:

$$u_{max} = SMAX \cdot e^{-EXP \cdot D}$$

where D is the damage parameter $D_{n+1} = D_n + \Delta u$. The shear force, acting on area A_s , at time $n+1$ is given as:

$$f_{n+1} = \min(f_n - GB \cdot A_s \cdot \Delta u, GB \cdot A_s \cdot u_{max})$$

*CONTACT

*CONTACT_2D_OPTION1_{OPTION2}_{OPTION3}

Purpose : Define a 2-dimensional contact or slide line. This option is to be used with 2D solid and shell elements using the plane_stress, plane_strain or axisymmetric formulations, see *SECTION_SHELL,

OPTION1 specifies the contact type. The following options should be used with deformable materials only (i.e., not rigid):

SLIDING_ONLY

TIED_SLIDING

SLIDING_VOIDS

since these methods are based on the imposition of constraints. The constraint methods may be used with rigid bodies if the rigid body is the master surface and all rigid body motions are prescribed. The following options may be used with rigid materials as well:

PENALTY_FRICTION

PENALTY

AUTOMATIC_SINGLE_SURFACE

AUTOMATIC_SURFACE_TO_SURFACE

AUTOMATIC_NODE_TO_SURFACE

AUTOMATIC_SURFACE_IN_CONTINUUM

OPTION2 specifies a thermal contact and takes the single option:

THERMAL

Only the AUTOMATIC types: SINGLE_SURFACE, SURFACE_TO_SURFACE, and NODE_TO_SURFACE may be used with this option.

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

TITLE

Note: *OPTION2* and *OPTION3* may appear in any order.

At present, the contact ID number and title are ignored by LS-DYNA but are included for extension in the near future. The title card is picked up by some of the peripheral LS-DYNA codes to aid in post-processing.

Single surface contact in two dimensions is accomplished by the AUTOMATIC_SURFACE_TO_SURFACE option when the master surface part set is set to zero. The SINGLE_SURFACE option in version 940 has been removed.

*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					

<u>VARIABLE</u>	<u>DESCRIPTION</u>
SSID	Nodal set ID for the slave nodes, see *SET_NODE. The slave surface must be to the left of the master surface.
MSID	Nodal set ID for the master nodes, see *SET_NODE.
TBIRTH	Birth time for contact.
TDEATH	Death time for contact.
EXT_PAS	Slideline extension bypass option. EQ:0 extensions are use EQ:1 extensions are not used
THETA1	Angle in degrees of slideline extension at first master node. EQ:0 extension remains tangent to first master segment.
THETA2	Angle in degrees of slideline extension at last master node. EQ:0 extension remains tangent to 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
TOLOFF	Tolerance for stiffness insertion for implicit solution only. The contact stiffness is inserted when a node approaches a segment a distance equal to the segment length multiplied by TOLOFF. The stiffness is increased as the node moves closer with the full stiffness being used when the nodal point finally makes contact. EQ:0.0 default set to 0.025.
FRCSCL	Scale factor for the interface friction. EQ:0.0 default set to 0.010
ONEWAY	Flag for one way treatment. If set to 1.0 the nodal points on the slave surface are constrained to the master surface. This option is generally recommended if the master surface is rigid. EQ:1.0 activate one way treatment.

<u>VARIABLE</u>	<u>DESCRIPTION</u>
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.

*CONTACT

For the AUTOMATIC options define the following two cards:

Card 1 1 2 3 4 5 6 7 8

Variable	SIDS	SIDM	SFACT	FREQ	FS	FD	DC	MEMBS
Type	I	I	F	I	F	F	F	I
Default	none	none	1.0	50	0.	0.	0.	12
Remarks	1,2	1,2						

Card 2 1 2 3 4 5 6 7 8

Variable	TBIRTH	TDEATH	SOS	SOM	NDS	NDM	COF	INIT
Type	F	F	F	F	I	I	I	I
Default	0.	1.e20	1.0	1.0	0	0	0	0
Remarks			3	3	4	5		8

This Card is mandatory for the THERMAL option, i.e.,:
***CONTACT_ AUTOMATIC_..._THERMAL_.....**

Optional 1 2 3 4 5 6 7 8

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

Optional Card A

Card A	1	2	3	4	5	6	7	8
Variable	VC	VDC	IPF	SLIDE	ISTIFF			
Type	F	F	I	I	I			
Default	0.	10.0	0	0	0			
Remarks				10				

VARIABLE

DESCRIPTION

SIDS	Set ID to define the slave surface. If SIDS>0, a part set is assumed, see *SET_PART. If SIDS<0, a node set with ID equal to the absolute value of SIDS is assumed, see *SET_NODE.
SIDM	Set ID to define the master surface. If SIDM>0, a part set is assumed, see *SET_PART. If SIDM<0, a node set with ID equal to the absolute value of SIDM is assumed, see *SET_NODE. Do not define for single surface contact.
SFACT	Scale factor for the penalty force stiffness.
FREQ	Search frequency. The number of timesteps between bucket sorts. For implicit contact this parameter is ignored and the search frequency is 1. EQ.0: default set to 50.
FS	Static coefficient of friction. The frictional coefficient is assumed to be dependent on the relative velocity v_{rel} of the surfaces in contact according to the relationship given by: $\mu_c = FD + (FS - FD)e^{-DC \cdot 1v_{rel}^1}$.
FD	Dynamic coefficient of friction. The frictional coefficient is assumed to be dependent on the relative velocity v_{rel} of the surfaces in contact $\mu_c = FD + (FS - FD)e^{-DC \cdot v_{rel} }$.
DC	Exponential decay coefficient. The frictional coefficient is assumed to be dependent on the relative velocity v_{rel} of the surfaces in contact $\mu_c = FD + (FS - FD)e^{-DC \cdot v_{rel} }$.
MEMBS	Parameter to allocate memory for bucket sort pair information.

*CONTACT

<u>VARIABLE</u>	<u>DESCRIPTION</u>
TBIRTH	Birth time for contact.
TDEATH	Death time for contact.
SOS	Surface offset from midline for 2D shells of slave surface EQ.0.0: default to 1. GT.0.0: scale factor applied to actual thickness LT.0.0: absolute value is used as the offset
SOM	Surface offset from midline for 2D shells of master surface EQ.0: default to 1. GT.0: scale factor applied to actual thickness LT.0: absolute value is used as the offset
NDS	Normal direction flag for 2D shells of slave surface EQ.0: Normal direction is determined automatically EQ.1: Normal direction is in the positive direction EQ.-1: Normal direction is in the negative direction
NDM	Normal direction flag for 2D shells of master surface EQ.0: Normal direction is determined automatically EQ.1: Normal direction is in the positive direction EQ.-1: Normal direction is in the negative direction
COF	Closing/Opening flag for implicit contact EQ.0: Recommended for most problem where gaps are only closing. EQ.1: Recommended when gaps are opening to avoid sticking.
INIT	Special processing during initializaion EQ.0: No special processing. EQ.1: Forming option.
CF	Thermal conductivity (k) of fluid between the slide surfaces. If a gap with a thickness l_{gap} exists between the slide surfaces, then the conductance due to thermal conductivity between the slide surfaces is $h_{cond} = \frac{k}{l_{gap}}$
	Note that LS- DYNA calculates l_{gap} based on deformation.
FRAD	Radiation factor, f , between the slide surfaces. A radiant-heat-transfer coefficient (h_{rad}) is calculated (see *BOUNDARY_RADIATION). If a gap exists between the slide surfaces, then the contact conductance is calculated by $h = h_{cond} + h_{rad}$

<u>VARIABLE</u>	<u>DESCRIPTION</u>
HTC	Heat transfer conductance (h_{cont}) for closed gaps. Use this heat transfer conductance for gaps in the range $0 \leq l_{gap} \leq l_{min}$ where l_{min} is GCRIT defined below.
GCRIT	Critical gap (l_{min}), use the heat transfer conductance defined (HTC) for gap thicknesses less than this value.
GMAX	No thermal contact if gap is greater than this value (l_{max}).
CD_FACT	Is a multiplier used on the element characteristic distance for the search routine. The characteristic length is the largest interface surface element diagonal. EQ.0: Default set to 1.0
VC	Coefficient for viscous friction. This is used to limit the friction force to a maximum. A limiting force is computed $F_{lim} = VC \cdot A_{cont}$. A_{cont} being the area of contacted between segments. The suggested value for VC is to use the yield stress in shear : $VC = \frac{\sigma_o}{\sqrt{3}}$ where σ_o is the yield stress of the contacted material.
VDC	Viscous damping coefficient in percent of critical for explicit contact.
IPF	Initial penetration flag for explicit contact. EQ.0: Allow initial penetrations to remain EQ.1: Push apart initially penetrated surfaces
SLIDE	Sliding option. EQ.0: Off EQ.1: On
ISTIFF	Stiffness scaling option. EQ.0: Use default option. EQ.1: Scale stiffness using segment masses and explicit time step (default for explicit contact) EQ.2: Scale stiffness using segment stiffness and dimensions (default for implicit contact)

*CONTACT

Remarks:

Remarks 1 through 11 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 ± 1 , flow in the direction of the normal is permitted.
6. When using AUTOMATIC_SURFACE_IN_CONTINUUM contact, there is no need to mesh the continuum around the structure because contact is not with continuum nodes but with material in the interior of the continuum elements. The algorithm works well for Eulerian or ALE elements since the structure does not interfere with remeshing. However, a structure will usually not penetrate the surface of an ALE continuum since the nodes are Lagrangian normal to the surface. Therefore, if using an ALE fluid, the structure should be initially immersed in the fluid and remain immersed throughout the calculation. Penetrating the surface of an Eulerian continuum is not a problem.
7. For all types of 2D_AUTOMATIC contact, eroding materials are treated by default. At present, subcycling is not possible.
8. Currently only one special initialization option is available. The forming option is intended for implicit solutions of thin solid parts when back side segments may interfere with the solution. It automatically removes back side segments during initialization. If slave or master surfaces are defined by node sets, the forming option should be turned off.

9. For the thermal option:
- $$h = h_{cont}, \text{ if the gap thickness is } 0 \leq l_{gap} \leq l_{min}$$
- $$h = h_{cond} + h_{rad}, \text{ if the gap thickness is } l_{min} \leq l_{gap} \leq l_{max}$$
- $$h = 0, \text{ if the gap thickness is } l_{gap} > l_{max}$$
10. When turned on, the sliding option activates additional logic intended to improve sliding when surfaces in contact have kinks or corners. This option is off by default.
11. The ISTIFF option allows control of the equation used in calculating the penalty stiffness. For backward compatibility, the default values are different for implicit and explicit solutions. When ISTIFF=1 is used, the explicit time step appears in the stiffness equation regardless if the calculation is implicit or explicit.

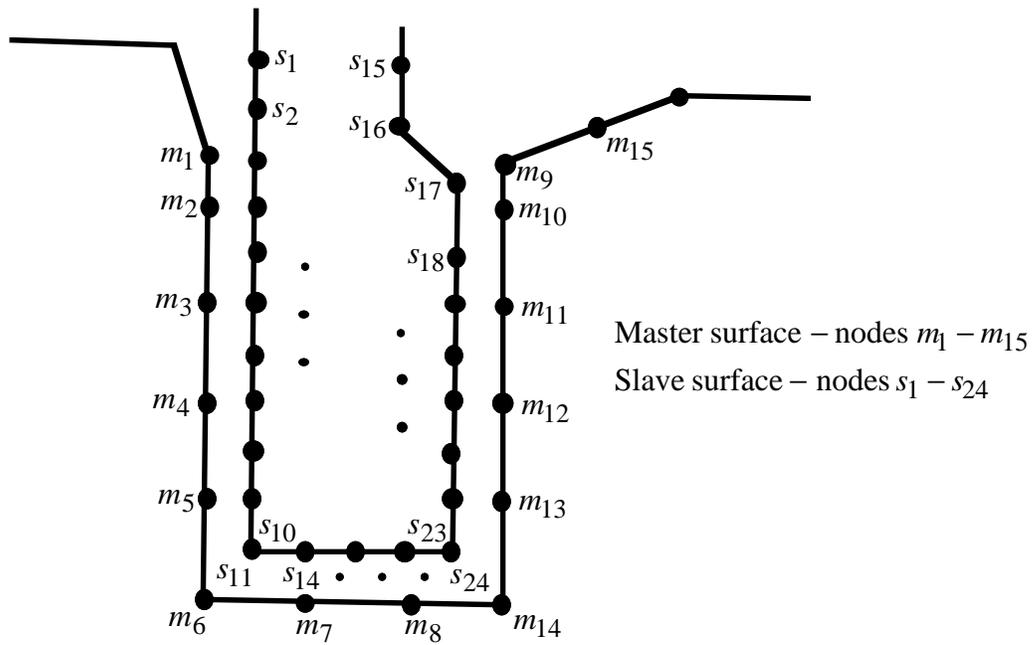
The remaining discussion applies to the SLIDING_ONLY, TIED_SLIDING, SLIDING_VOIDS, PENALTY_FRICTION, and PENALTY options. These options were adopted from LS-DYNA2D and originated in the public domain version of DYNA2D from the Lawrence Livermore National Laboratory. The AUTOMATIC contact options are generally recommended excepted for the TIED option.

Consider two slideline surfaces in contact. It is necessary to designate one as a slave surface and the other as a master surface. Nodal points defining the slave surface are called slave nodes, and similarly, nodes defining the master surface are called master nodes. Each slave-master surface combination is referred to as a slideline.

Many potential problems with the algorithm can be avoided by observing the following precautions:

- Metallic materials should contain the master surface along high explosive-metal interfaces.
- Sliding only type slidelines are appropriate along high explosive-metal interfaces. The penalty formulation is not recommended along such interfaces.
- If one surface is more finely zoned, it should be used as the slave surface. If penalty slidelines are used, PENALTY and PENALTY_FRICTION, the slave-master distinction is irrelevant.
- A slave node may have more than one master segment, and may be included as a member of a master segment if a slideline intersection is defined.
- Angles in the master side of a slideline that approach 90° must be avoided. Whenever such angles exist in a master surface, two or more slidelines should be defined. This procedure is illustrated in Figure 6.5. An exception for the foregoing rule arises if the surfaces are tied. In this case, only one slideline is needed.
- Whenever two surfaces are in contact, the smaller of the two surfaces should be used as the slave surface. For example, in modeling a missile impacting a wall, the contact surface on the missile should be used as the slave surface.
- Care should be used when defining a master surface to prevent the extension from interfering with the solution. In Figures 6.6 and 6.7, slideline extensions are shown.

*CONTACT



1		2		3	
<u>Slaves</u>	<u>Masters</u>	<u>Slaves</u>	<u>Masters</u>	<u>Slaves</u>	<u>Masters</u>
s1	m1	s11	m6	s24	m14
s2	m2	s12	m7	s23	m13
.	.	.	m8	.	.
.	.	.	m14	.	.
.
		s14			m9
s11	m6	s24		s15	m15

Figure 6.5. Proper definition of illustrated slave-master surface requires three slidelines (note that slave surface is to the left of the master surface as one moves along master nodes in order of definition).

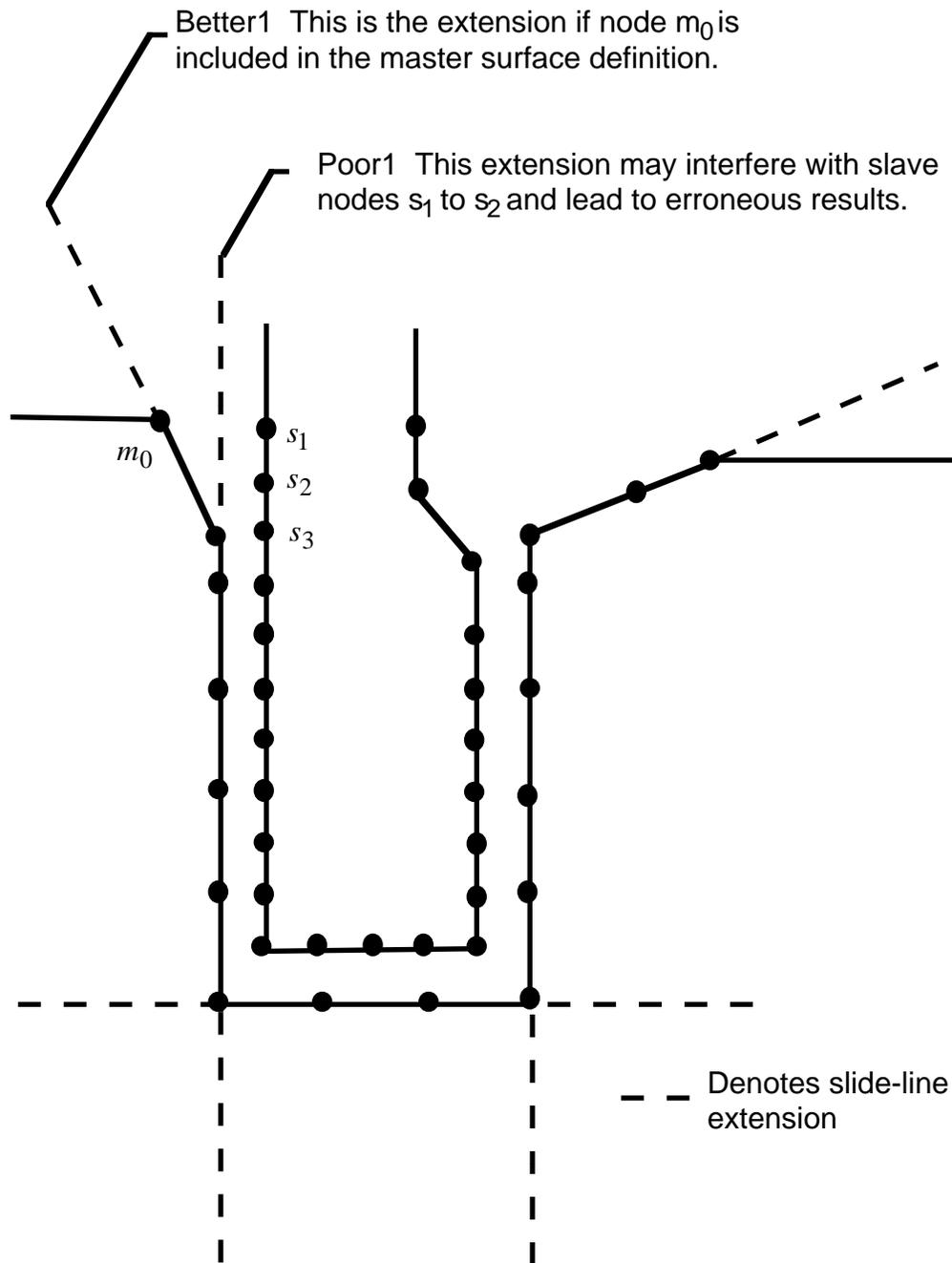


Figure 6.6. Master surface extensions defined automatically by DYNA (extensions are updated every time step to remain tangent to ends of master sides of slidelines unless angle of extension is defined in input).

*CONTACT

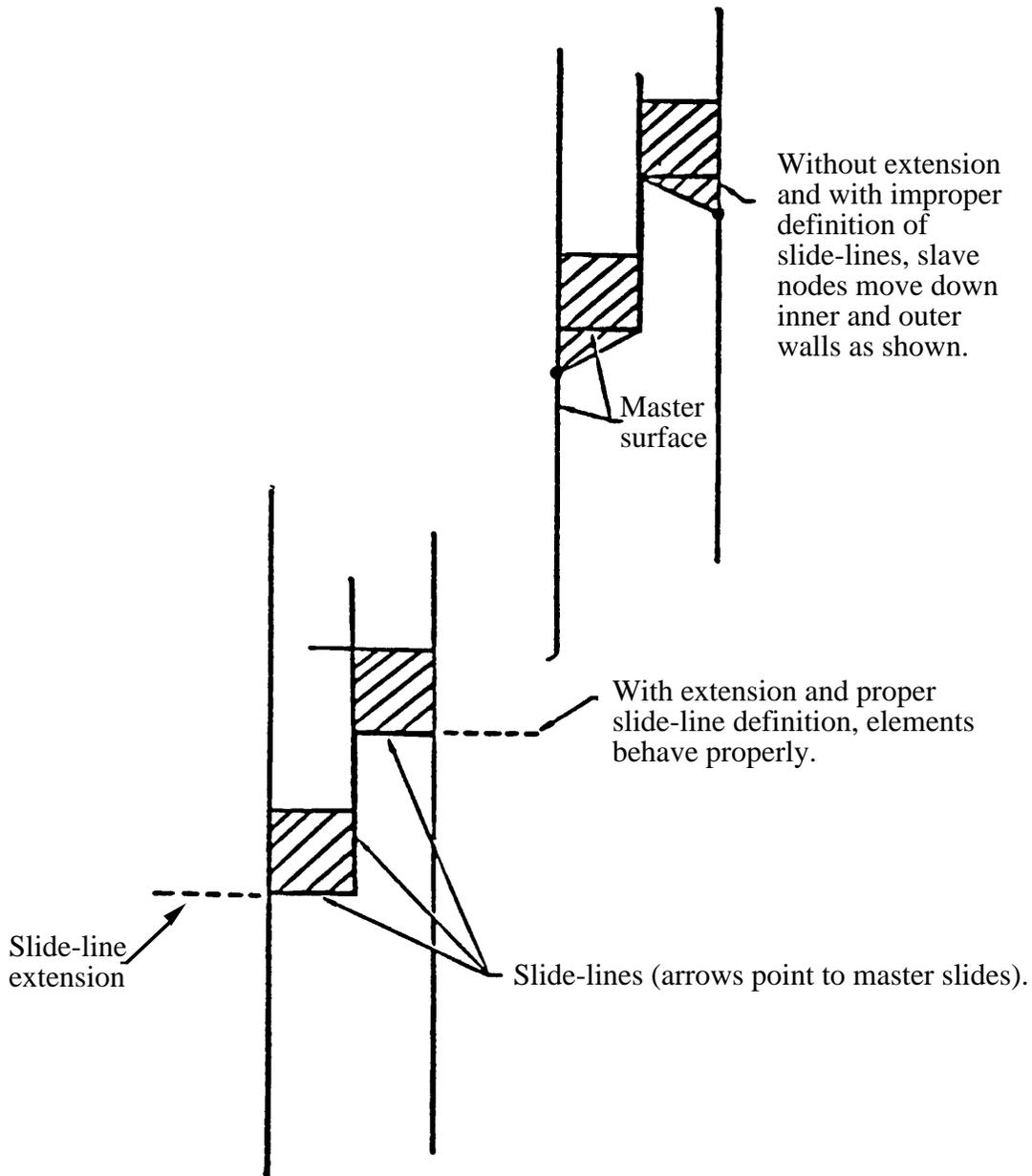


Figure 6.7 Example of slideline extensions helping to provide realistic response.

***CONTROL**

The keyword control cards are optional and can be used to change defaults, activate solution options such as mass scaling, adaptive remeshing, and an implicit solution; however, it is advisable to define the *CONTROL_TERMINATION card. **The ordering of the control cards in the input file is arbitrary. To avoid ambiguities, define no more than one control card of each type.** The following control cards are organized in an alphabetical order:

- *CONTROL_ACCURACY**
- *CONTROL_ADAPSTEP**
- *CONTROL_ADAPTIVE**
- *CONTROL_ALE**
- *CONTROL_BULK_VISCOSITY**
- *CONTROL_CFD_AUTO**
- *CONTROL_CFD_GENERAL**
- *CONTROL_CFD_MOMENTUM**
- *CONTROL_CFD_PRESSURE**
- *CONTROL_CFD_TRANSPORT**
- *CONTROL_CFD_TURBULENCE**
- *CONTROL_CHECK_{OPTION}**
- *CONTROL_COARSEN**
- *CONTROL_CONTACT**
- *CONTROL_COUPLING**
- *CONTROL_CPU**
- *CONTROL_DYNAMIC_RELAXATION**
- *CONTROL_EFG**
- *CONTROL_ENERGY**
- *CONTROL_EXPLOSIVE_SHADOW**
- *CONTROL_HOURLASS_{OPTION}**
- *CONTROL_IMPLICIT_AUTO**
- *CONTROL_IMPLICIT_BUCKLE**
- *CONTROL_IMPLICIT_DYNAMICS**
- *CONTROL_IMPLICIT_EIGENVALUE**
- *CONTROL_IMPLICIT_GENERAL**
- *CONTROL_IMPLICIT_MODES**
- *CONTROL_IMPLICIT_SOLUTION**
- *CONTROL_IMPLICIT_SOLVER**

***CONTROL**

***CONTROL_IMPLICIT_STABILIZATION**
***CONTROL_MPP_DECOMPOSITION_AUTOMATIC**
***CONTROL_MPP_DECOMPOSITION_CHECK_SPEED**
***CONTROL_MPP_DECOMPOSITION_CONTACT_DISTRIBUTE**
***CONTROL_MPP_DECOMPOSITION_CONTACT_ISOLATE**
***CONTROL_MPP_DECOMPOSITION_FILE**
***CONTROL_MPP_DECOMPOSITION_METHOD**
***CONTROL_MPP_DECOMPOSITION_NUMPROC**
***CONTROL_MPP_DECOMPOSITION_RCBLOG**
***CONTROL_MPP_DECOMPOSITION_SHOW**
***CONTROL_MPP_DECOMPOSITION_TRANSFORMATION**
***CONTROL_MPP_IO_NOD3DUMP**
***CONTROL_MPP_IO_NODUMP**
***CONTROL_MPP_IO_NOFULL**
***CONTROL_MPP_IO_SWAPBYTES**
***CONTROL_NONLOCAL**
***CONTROL_OUTPUT**
***CONTROL_PARALLEL**
***CONTROL_REMESHING**
***CONTROL_RIGID**
***CONTROL_SHELL**
***CONTROL_SOLID**
***CONTROL_SOLUTION**
***CONTROL_SPH**
***CONTROL_STRUCTURED_{OPTION}**
***CONTROL_SUBCYCLE**
***CONTROL_TERMINATION**
***CONTROL_THERMAL_NONLINEAR**
***CONTROL_THERMAL_SOLVER**
***CONTROL_THERMAL_TIMESTEP**
***CONTROL_TIMESTEP**

LS-DYNA's implicit mode may be activated in two ways. Using the ***CONTROL_IMPLICIT_GENERAL** keyword, a simulation may be flagged to run entirely in implicit mode. Alternatively, an explicit simulation may be seamlessly switched into implicit mode at the termination time using the ***INTERFACE_SPRINGBACK_SEAMLESS** keyword. The seamless switching feature is intended to simplify metal forming springback calculations, where the forming phase can be run in explicit mode, followed immediately by an implicit static springback

simulation. In case of difficulty, restart capability is supported. Eight keywords are available to support implicit analysis. Default values are carefully selected to minimize input necessary for most simulations. These are summarized below:

- *CONTROL_IMPLICIT_GENERAL
Activates implicit mode, selects time step size.
- *CONTROL_IMPLICIT_SOLVER
Selects parameters for solving system of linear equations $[K]\{x\}=\{f\}$.
- *CONTROL_IMPLICIT_SOLUTION
Selects linear or nonlinear solution method, convergence tolerances.
- *CONTROL_IMPLICIT_AUTO
Activates automatic time step control.
- *CONTROL_IMPLICIT_DYNAMICS
Activates and controls dynamic implicit solution using Newmark method.
- *CONTROL_IMPLICIT_EIGENVALUE
Activates and controls eigenvalue analysis.
- *CONTROL_IMPLICIT_MODES
Activates and controls computation of constraint and attachment modes.
- *CONTROL_IMPLICIT_STABILIZATION
Activates and controls artificial stabilization for multi-step springback.

*CONTROL

*CONTROL_ACCURACY

Purpose: Define control parameters that can improve the accuracy of the calculation.

Card Format

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

<u>VARIABLE</u>	<u>DESCRIPTION</u>
OSU	Global flag for 2nd order objective stress updates (See Remark 1 below). Generally, for explicit calculations only those parts undergoing large rotations, such as rolling tires, need this options. Objective stress updates can be activated for a subset of part IDs by defining the part set in columns 21-30. EQ.0: Off (default) EQ.1: On
INN	Invariant node numbering for shell and solid elements (See Remarks 2 and 3 below). EQ.1: Off (default for explicit) EQ.2: On for shell elements only (default for implicit) EQ.3: On for solid elements only EQ.4: On for both shell and solid elements
PIDOSU	Part set ID for objective stress updates. If this part set ID is given only those part IDs listed will use the objective stress update; therefore, OSU is ignored.

Remarks:

1. Second order objective stress updates are occasionally necessary. Some examples include spinning bodies such as turbine blades in a jet engine, high velocity impacts generating large strains in a few time steps, and large time step sizes due to mass scaling in metal forming. There is a significant added cost which is due in part to the added cost of the second order terms in the stress update when the Jaumann rate is used and the need to compute the strain-displacement matrix at the mid-point geometry. This option is available for one point brick elements, the selective-reduced integrated brick element which uses eight integration points, the fully integrated plane strain and axisymmetric volume weighted (type 15) 2D solid elements, the fully integrated thick shell element, and the following shell elements: Belytschko-Tsay, Belyschko-Tsay with warping stiffness, Belyschko-Chiang-Wong, S/R Hughes-Liu, and the type 16 fully integrated shell element.

2. Invariant node numbering for shell elements affects the choice of the local element shell coordinate system. The orientation of the default local coordinate system is based on the shell normal vector and the direction of the 1-2 side of the element. If the element numbering is permuted, the results will change in irregularly shaped elements. With invariant node numbering, permuting the nodes shifts the local system by an exact multiple of 90 degrees. In spite of its higher costs [$<5\%$], the invariant local system is recommended for several reasons. First, element forces are nearly independent of node sequencing; secondly, the hourglass modes will not substantially affect the material directions; and, finally, stable calculations over long time periods are achievable.
3. Invariant node numbering for solid elements is available for anisotropic materials only. This option has no effect on solid elements of isotropic material. This option is recommended when solid elements of anisotropic material undergo significant deformation.

*CONTROL

*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		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

<u>VARIABLE</u>	<u>DESCRIPTION</u>
	<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.7: 3D r-adaptive remeshing for solid elements. Solid element type 13, a tetrahedron, is used in the adaptive remeshing process. A completely new mesh is generated which is initialized from the old mesh using a least squares approximation. The mesh size is currently based on the minimum and maximum edge lengths defined on the *CONTROL_REMESHING keyword input. This option remains under development, and, we are not sure of its reliability on complex geometries.</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 *NODE, *ELEMENT, *SHELL, and *BOUNDARY_, *CONTACT_NODE_, and *CONSTRAINED_ADAPTIVITY, 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.

VARIABLE	DESCRIPTION
ADPENE	Adapt the mesh when the contact surfaces approach or penetrate the tooling surface depending on whether the value of ADPENE is positive (<i>approach</i>) or negative (<i>penetrates</i>), respectively. The tooling adaptive refinement is based on the curvature of the tooling. If ADPENE is positive the refinement generally occurs before contact takes place; consequently, it is possible that the parameter ADPASS can be set to 1 in invoke the one pass adaptivity.
ADPTH	.EQ.0.0 This parameter is ignored .GT.0.0 Absolute shell thickness level below which adaptive remeshing should began. .LT.0.0 Element thickness ratio. If the ratio of the element thickness to the original element thickness is less than the absolute value of ADPTHK, the element will be refined. This option works only if ADPTOL is nonzero. If thickness based adaptive remeshing is desired without angle changes, then, set ADPTOL to a large angle.
MEMORY	This flag can have two meanings depending on whether the memory environmental variable is or is not set. The command " <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> ". 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. 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.
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.

*CONTROL

Remarks:

1. D3DUMP and RUNRSF files contain all information necessary to restart an adaptive run. This did not work in version 936 of LS-DYNA.
2. Card 2 input is optional and is not required.
3. In order for this control card to work, the flag ADPOPT=1 must be set in the *PART definition. Otherwise, adaptivity will not function.
4. In order for adaptivity to work optimally, the parameter SNLOG=1, must be set on Optional Control Card B in the *CONTACT Section. On disjoint tooling meshes the contact option *CONTACT_FORMING_..... is recommended.
5. A file *adapt.rid* is left on disk after the adaptive run is completed. This file contains the root ID of all elements that are created during the calculation, and it does not need to be kept if it is not used in post-processing.

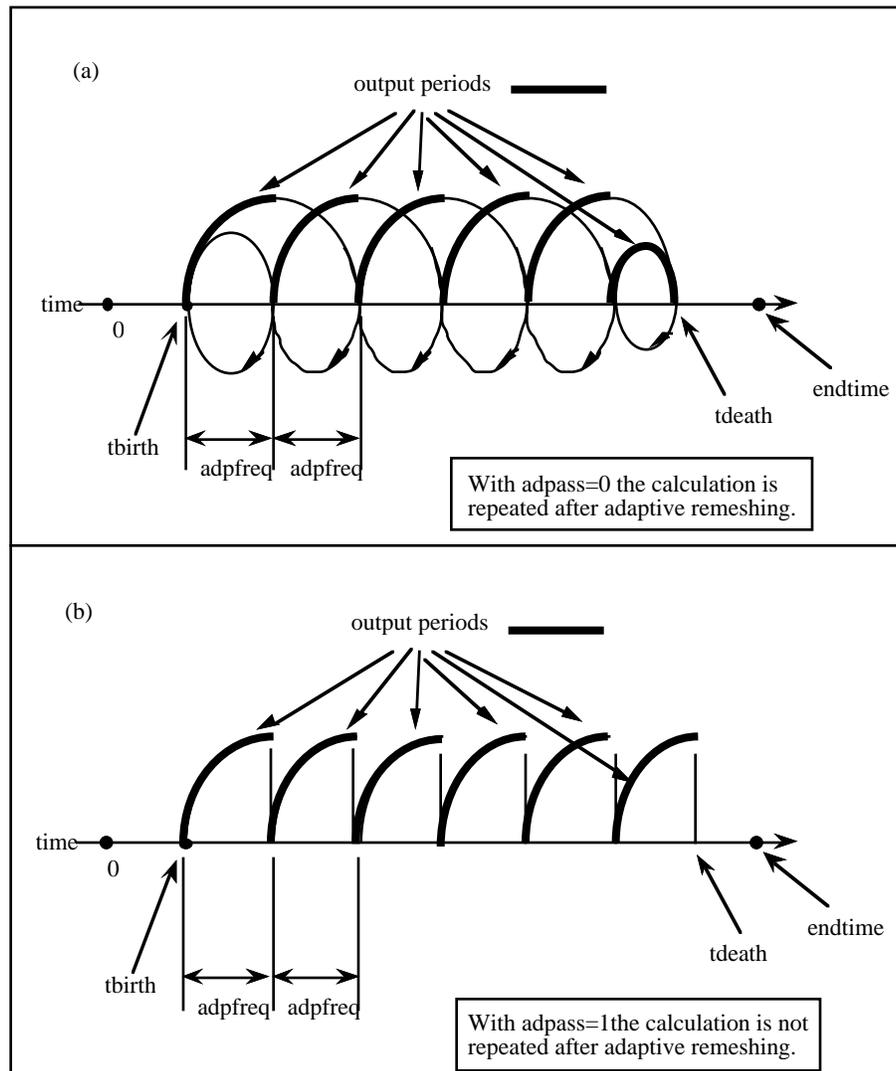


Figure 7.1. At time= t_{birth} the adaptive calculation begins. After computing for a time interval $adpfreq$ error norms are computed. If $ADPASS=0$, then the mesh that existed at time= t_{birth} is refined based on the computed error norms. With the new mesh, the calculation continues to time= $t_{birth}+2 \times adpfreq$ 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

*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	1	0	0	0	0	0

Card 2 1 2 3 4 5 6 7 8

Variable	START	END	AAFAC	VFACT	PRIT	EBC	PREF	NSIDEBC
Type	F	F	F	F	F	I	F	
Default	0	1. 0E+20	1	1.0E-06	0.0	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 advectons.
METH	Advection method: EQ.1: donor cell + HIS (Half-Index-Shift, first order accurate), EQ.2: Van Leer + HIS (Half-Index-Shift, second order).
AFAC	ALE smoothing weight factor - Simple average: EQ.-1: turn smoothing off.
BFAC	ALE smoothing weight factor – Volume weighting

<u>VARIABLE</u>	<u>DESCRIPTION</u>
CFAC	ALE smoothing weight factor – Isoparametric
DFAC	ALE smoothing weight factor – Equipotential
EFAC	ALE smoothing weight factor – Equilibrium
START	Start time for ALE smoothing
END	End time for ALE smoothing
AAFAC	ALE advection factor (donor cell options, default=1.0)
VFACT	Volume fraction limit for stresses in single material and void formulation. All stresses are set to zero for elements with lower volume fraction than VFACT. EQ.0.0: set to default 1.0E-06.
PRIT	A flag to turn on or off the pressure equilibrium iteration option for multi-material elements (See Remark 1). EQ.0. Off (default) EQ.1. On
EBC	Automatic Euler boundary condition EQ.0. Off EQ.1. On with stick condition EQ.2. On with slip condition
PREF	Reference pressure applied to the free surfaces of the ALE mesh boundary. (See Remark 3).
NSIDEBC	A node set ID (NSID) which is to be excluded from the EBC constraint.

Remarks

1. By default, all materials in a multi-material element are assumed to undergo the same element averaged strain rates. This assumption may not be robust when mixing materials with very different compressibilities. In this case, an assumption of pressure equilibrium in the element may be more appropriate.
2. This option, used for ALE and EULER formulations, automatically defines velocity boundary condition constraints for the user. The constraints, once defined, are applied to all nodes on free surfaces of an 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.
3. The PREF definition is equivalent to using the *LOAD_SEGMENT card to provide pressure loading on the free surfaces of all ALE or Eulerian mesh(es) in a model. This cannot be used to initialize the internal pressure of the material (that must be done via the *EOS_ or *BOUNDARY_AMBIENT_EOS cards).

*CONTROL

*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_CFD_AUTO**

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

Card Format

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

VARIABLE

DESCRIPTION

IAUTO

Set the time step control type:
EQ.0: IAUTO=1 for fixed time step size,
EQ.1: Fixed time step based on DTINIT - see
*CONTROL_CFD_GENERAL (default),
EQ.2: Time step based on CFL/stability for INSOL=3/1,
EQ.3: Automatic time step selection based on a second-order Adams-
Bashforth predictor with a trapezoidal rule corrector.

EPSDT

Set the tolerance for local truncation error in time.
EQ.0: EPSDT=1.0e-3 (default).

DTSF

Set the maximum time step scale factor that may be applied at any given time step. This puts the upper limit on the amount that the time step can be increased during any given time step.
EQ.0: DTSF=1.25 (default).

DTMAX

Set the upper limit on the time step size. This value puts a ceiling on how far the time step may be increased for IAUTO=3.
EQ.0: 10*DTINIT (default).

Remarks:

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

*CONTROL

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

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

*CONTROL_CFD_GENERAL

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

Card Format

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

VARIABLE

DESCRIPTION

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

*CONTROL

IACURC

Activate the use of full-quadrature for certain terms in the momentum and transport equations. The accuracy flag improves the accuracy of body force calculations and certain advective/convective terms with a modest increase in computational time.

EQ.0: don't use the increased quadrature rules (default),

EQ.1: use the increased quadrature on advective/convective and body force terms.

Remarks:

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

***CONTROL_CFD_MOMENTUM**

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

Card Format

Card 1 1 2 3 4 5 6 7 8

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

Card 2 1 2 3 4 5 6 7 8

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

VARIABLE

DESCRIPTION

IMASS

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

*CONTROL

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

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

Remarks:

1. The IMASS variable is only active when $INSOL \geq 2$ on the *CONTROL_CFD_GENERAL keyword.
2. The balancing tensor diffusivity should always be used with the explicit forward-Euler treatment of the advection terms. This is the default.
3. The use of the flux limiting procedures is currently restricted to the explicit advection procedures.
4. DIVU sets the ceiling on the discrete divergence that is permitted during a simulation when $INSOL=1$. If the divergence at a given time step exceeds the value set by DIVU, then an intermediate projection is performed to return the velocity to a div-free state.
5. The time weighting variables only apply to the case when $INSOL \geq 2$ on the *CONTROL_CFD_GENERAL keyword.
6. The MSOL keyword for the *CONTROL_CFD_MOMENTUM keyword only applies for $INSOL \geq 2$ on the *CONTROL_CFD_GENERAL keyword.

*CONTROL

*CONTROL_CFD_PRESSURE

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

Card Format

Card 1 1 2 3 4 5 6 7 8

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

Card 2 1 2 3 4 5 6 7 8

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

VARIABLE

DESCRIPTION

IPSOL

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

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

*CONTROL

*CONTROL_CFD_TRANSPORT

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

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	ITEMP		NSPEC					
Type	I		I					
Default	0		0					
Remarks								

Card 2 1 2 3 4 5 6 7 8

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

Card 3 1 2 3 4 5 6 7 8

Variable	ITSOL	MAXIT	ICHKIT	IWRT	IHIST	EPS	IHG	EHG
Type	I	I	I	I	I	F	I	F
Default	20	100	2	0	0	1.0e-5	1	1.0
Remarks	5							

VARIABLE

DESCRIPTION

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

*CONTROL

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

Remarks:

1. The IMASS variable is only active when $INSOL \geq 2$ on the *CONTROL_CFD_GENERAL keyword.
2. The balancing tensor diffusivity should always be used with explicit treatment of the advection terms. (This is the default.)
3. The use of flux limiting procedures is currently restricted to the explicit advective procedures.
4. The time weighting variables only apply to the case when $INSOL \geq 2$ on the *CONTROL_CFD_GENERAL keyword.
5. The ITSOL keyword for the *CONTROL_CFD_TRANSPORT keyword only applies for $INSOL \geq 2$ on the *CONTROL_CFD_GENERAL keyword.

*CONTROL

*CONTROL_CFD_TURBULENCE

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

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	ITRB	SMAGC						
Type	I	F						
Default	0	0.1						
Remarks		1						

VARIABLE

DESCRIPTION

ITRB

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

SMAGC

Smagorinsky Constant:
EQ.0: set to 0.1 (default).

Remarks:

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

***CONTROL_CHECK_{OPTION}**

Options include:

<BLANK>

SHELL

Purpose: Check for various problems in the mesh. Checking is performed during the input phase.

For the SHELL option, shell element integrity checks which have been identified as important in metal forming applications are performed. These checks can improve springback convergence and accuracy. This option will repair bad elements created, for example, during trimming operations.

Card Format (*OPTION = SHELL*) (include one card for each part to be checked)

Card 1 1 2 3 4 5 6 7 8

Variable	PID	IFAUTO	CONVEX	ADPT	ARATIO	ANGLE	SMIN	
Type	I	I	I	I	F	F	F	
Default	0	0	1	1	0.25	150.0	0.0	

VARIABLE**DESCRIPTION**

PID	Part ID to be checked: EQ.0: do not check
IFAUTO	Flag to automatically correct bad elements: EQ.0: write warning message only EQ.1: fix bad element, write message
CONVEX	Check element convexity (internal angles less than 180 degrees) EQ.0: do not check EQ.1: check
ADPT	Check adaptive constraints EQ.0: do not check EQ.1: check
ARATIO	Minimum allowable aspect ratio. Elements which do not meet minimum aspect ratio test will be treated according to IFAUTO above.

*CONTROL

<u>VARIABLE</u>	<u>DESCRIPTION</u>
ANGLE	Maximum allowable internal angle. Elements which fail this test will be treated according to IFAUTO above.
SMIN	Minimum element size. Elements which fail this test will be treated according to IFAUTO above.

Remarks:

1. If the convexity test is activated, all failed elements will be fixed regardless of IFAUTO.
2. In addition to illegal constraint definitions (slave which is also a master), checks are performed for mesh connectivities which have been found to cause convergence trouble in implicit springback applications.

***CONTROL_COARSEN**

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

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	ICOARSE	ANGLE	NSEED	PSID				
Type	I	F	I	I				
Default	0	none	0	0				

Card 2 1 2 3 4 5 6 7 8

Variable	N1	N2	N3	N4	N5	N6	N7	N8
Type	I	I	I	I	I	I	I	I
Default	0	0	0	0	0	0	0	0

VARIABLE

DESCRIPTION

ICOARSE

Coarsening flag:

EQ.0: do not coarsen (default),

EQ.1: coarsen mesh at beginning of simulation for forming model

Eq.2: coarsen mesh at beginning of simulation for crash model

ANGLE

Allowable angle change between neighboring elements. Adjcent elements which are flat to within ANGLE degrees are merged. (Suggested starting value = 8.0 degrees)

NSEED

Number of seed nodes (optional).

EQ.0: use only automatic searching.

EQ.n: also search starting with node IDs given below (maximum = 8 nodes)

PSID

Part set ID. all the parts defined in this set will be prevented from been coarsened.

*CONTROL

<u>VARIABLE</u>	<u>DESCRIPTION</u>
N1...N8	Optional list of seed node IDs for extra searching.

Remarks:

1. Coarsening is performed at the start of a simulation. The first plot state represents the coarsened mesh. By setting the termination time to zero and including the keyword *INTERFACE_SPRINGBACK_LSDYNA a keyword input deck can be generated containing the coarsened mesh.
2. By default, an automatic search is performed to identify elements for coarsening. In some meshes, isolated regions of refinement may be overlooked. Seed nodes can be identified in these regions to assist the automatic search. Seed nodes identify the central node of a four-element group which is coarsened into a single element if the angle criterion is satisfied.
3. The keyword *DEFINE_BOX_COARSEN can be used to indicate regions of the mesh which are protected from coarsening.

***CONTROL_CONTACT**

Purpose: Change defaults for computation with contact surfaces.

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	SLSFAC	RWPNAL	ISLCHK	SHLTHK	PENOPT	THKCHG	ORIEN	ENMASS
Type	F	F	I	I	I	I	I	I
Default	.1	none	1	0	1	0	1	0

Card 2 1 2 3 4 5 6 7 8

Variable	USRSTR	USRFRC	NSBCS	INTERM	XPENE	SSTHK	ECDT	TIEDPRJ
Type	I	I	I	I	F	I	I	I
Default	0	0	10-100	0	4.0	0	0	0

Card 3 is optional. The following parameters are the default values used by parts in automatic contacts. These frictional coefficients apply only to contact types: **SINGLE_SURFACE**, **AUTOMATIC_GENERAL**, **AUTOMATIC_SINGLE_SURFACE**, **AUTOMATIC_NODES_TO_...**, **AUTOMATIC_SURFACE_...**, and **AUTOMATIC_ONE_WAY_...**, and **ERODING_SINGLE_SURFACE**. Also see ***CONTACT** and ***PART**. Note that these default values will override the values specified for these contact types in the ***CONTACT** section.

Card 3 1 2 3 4 5 6 7 8

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	

*CONTROL

Card 4 is optional. If this card is defined, then Card 3 above must be included. A blank card may be inserted for Card 3.

Card 4 1 2 3 4 5 6 7 8

Variable	IGNORE	FRCENG	SKIPRWG	OUTSEG	SPOTSTP	SPOTDEL		
Type	I	I	I	I	I	I		
Default	0	0	0	0	0	0		

<u>VARIABLE</u>	<u>DESCRIPTION</u>
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 files (see remarks below): EQ.0: the default is set to 1, EQ.1: no checking, EQ.2: full check of initial penetration is performed.
SHLTHK	Shell thickness considered in type surface to surface and node to surface type contact options, where options 1 and 2 below activate the new contact algorithms. The thickness offsets are always included in single surface, constraint method, and automatic surface to surface and node to surface contact types (See remarks below.): EQ.0: thickness is not considered, EQ.1: thickness is considered but rigid bodies are excluded, EQ.2: thickness is considered including rigid bodies.
PENOPT	Penalty stiffness value option. For default calculation of the penalty value please refer to the Theoretical Manual. EQ.0: the default is set to 1, EQ.1: minimum of master segment and slave node (default for most contact types),

<u>VARIABLE</u>	<u>DESCRIPTION</u>
	EQ.2: use master segment stiffness (old way), EQ.3: use slave node value, EQ.4: use slave node value, area or mass weighted, EQ.5: same as 4 but inversely proportional to the shell thickness. This may require special scaling and is not generally recommended. Options 4 and 5 are recommended for metalforming calculations.
THKCHG	Shell thickness changes considered in single surface contact: EQ.0: no consideration (default), EQ.1: shell thickness changes are included.
ORIEN	Optional automatic reorientation of contact interface segments during initialization: EQ.0: default is set to 1. EQ.1: active for automated (part) input only. Contact surfaces are given by *PART definitions. EQ.2: active for manual (segment) and automated (part) input. EQ.3: inactive.
ENMASS	Treatment of the mass of eroded nodes in contact. This option effects all contact types where nodes are removed after surrounding elements fail. Generally, the removal of eroded nodes makes the calculation more stable; however, in problems where erosion is important the reduction of mass will lead to incorrect results. EQ.0: eroding nodes are removed from the calculation. EQ.1: eroding nodes of solid elements are retained and continue to be active in contact. EQ.2: the eroding nodes of solid and shell elements are retained and continue to be active in contact.
USRSTR	Storage per contact interface for user supplied interface control subroutine, see Appendix D. If zero, no input data is read and no interface storage is permitted in the user subroutine. This storage should be large enough to accommodate input parameters and any history data. This input data is available in the user supplied subroutine.
USRFRC	Storage per contact interface for user supplied interface friction subroutine, see Appendix E. If zero, no input data is read and no interface storage is permitted in the user subroutine. This storage should be large enough to accommodate input parameters and any history data. This input data is available in the user supplied subroutine.
NSBCS	Number of cycles between contact searching using three dimensional bucket searches. Defaults recommended.
INTERM	Flag for intermittent searching in old surface to surface contact using the interval specified as NSBCS above: EQ.0: off, EQ.1: on.

*CONTROL

<u>VARIABLE</u>	<u>DESCRIPTION</u>
XPENE	Contact surface maximum penetration check multiplier. If the small penetration checking option, PENCHK, on the contact surface control card is active, then nodes whose penetration then exceeds the product of XPENE and the element thickness are set free, see *CONTACT_OPTION_...: EQ.0: default is set to 4.0.
SSTHK	Flag for using actual shell thickness in single surface contact logic-types 4, 13, 15 and 26. See remarks 1 and 2 below. EQ.0: Actual shell thickness is not used in the contacts.(default), EQ.1: Actual shell thickness is used in the contacts. (sometimes recommended for metal forming calculations).
ECDT	Time step size override for eroding contact: EQ.0: contact time size may control Dt. EQ.1: contact is not considered in Dt determination.
TIEDPRJ	Bypass projection of slave nodes to master surface in types: *CONTACT_TIED_NODES_TO_SURFACE, *CONTACT_TIED_SHELL_EDGE_TO_SURFACE, and, *CONTACT_TIED_SURFACE_TO_SURFACE tied interface options: EQ.0: eliminate gaps by projection nodes, EQ.1: bypass projection. Gaps create rotational constraints which can substantially affect results.
SFRIC	Default static coefficient of friction (see *PART_CONTACT)
DFRIC	Default dynamic coefficient of friction (see *PART_CONTACT)
EDC	Default exponential decay coefficient (see *PART_CONTACT)
VFC	Default viscous friction coefficient (see *PART_CONTACT)
TH	Default contact thickness (see *PART_CONTACT)
TH_SF	Default thickness scale factor (see *PART_CONTACT)
PEN_SF	Default local penalty scale factor (see *PART_CONTACT)
IGNORE	Ignore initial penetrations in the *CONTACT_AUTOMATIC options. "Initial" in this context refers to the first timestep that a penetration is encountered. This option can also be specified for each interface on the third optional card under the keyword, *CONTACT. The value defined here will be the default. EQ.0: move nodes to eliminate initial penetrations in the model definition. EQ.1: allow initial penetrations to exist by tracking the initial penetrations. EQ.2: allow initial penetrations to exist by tracking the initial penetrations. However, penetration warning messages are printed with the original coordinates and the recommended coordinates of each slave node given.

VARIABLE	DESCRIPTION
FRCENG	Flag to activate the calculation of frictional sliding energy: EQ.0: do not calculate, EQ.1: calculate frictional energy in contact and store as "Surface Energy Density" in the binary INTFOR file.
SKIPRWG	Flag not to display stationary rigid wall by default. EQ.0: generate 4 extra nodes and 1 shell element to visualize stationary planar rigid wall. EQ.1: do not generate stationary rigid wall.
OUTSEG	Flag to output each spot weld slave node and its master segment for contact type: *CONTACT_SPOTWELD into the D3HSP file. EQ.0: no, do not write out this information. EQ.1: yes, write out this information.
SPOTSTP	If a spot weld node (related to a *MAT_SPOTWELD beam) cannot be found on a master segment, should an error termination occur? EQ.0: no, print warning message and continue calculation. EQ.1: yes, print error message and terminate.
SPOTDEL	If a spot weld node of a spot weld beam is attached to a shell element, which fails and is deleted, then the attached spot weld beam element is deleted if this flag is on. There is a small cost penalty related to this option on non-vector processors. On vector processors, however, this option can significantly slow down the calculation if many weld elements fail since the vector lengths are reduced. EQ.0: no, do not delete the beam element, EQ.1: yes, delete the beam elements when the attached shell fails.

Remarks:

1. The shell thickness change option must be activated in CONTROL_SHELL control input (see ISTUPD) and a nonzero flag specified for SHLTHK above before the shell thickness changes can be included in the surface to surface contact types. An additional flag must be set, see THKCHG above, if thickness changes are included in the single surface contact algorithms. The contact algorithms that include the shell thickness are relatively recent and are now fully optimized and parallelized. The searching in these algorithms is considerably more extensive and therefore slightly more expensive.
2. In the single surface contacts types SINGLE_SURFACE, AUTOMATIC_SINGLE_SURFACE, and ERODING_SINGLE_SURFACE, the default contact thickness is taken as the smaller of two values -- the shell thickness or 40% of the minimum edge length. This may create unexpected difficulties if it is the intent to include thickness effects when the in-plane shell element dimensions are less than the thickness. The default is based on years of experience where it has been observed that sometimes rather large nonphysical thicknesses are specified to achieve high stiffness values. Since the global searching algorithm includes the effects of shell thicknesses, it is possible to slow the searches down considerably by using such nonphysical thickness dimensions.
3. The initial penetration check option is always performed in v. 950 regardless 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.

*CONTROL

4. Automatic reorientation requires offsets between the master and slave surface segments. The reorientation is based on segment connectivity and, once all segments are oriented consistently based on connectivity, a check is made to see if the master and slave surfaces face each other based on the right hand rule. If not, all segments in a given surface are reoriented. This procedure works well for non disjoint surfaces. If the surfaces are disjoint, the AUTOMATIC contact options, which do not require orientation, are recommended. In the FORMING contact options automatic reorientation works for disjoint surfaces.

***CONTROL_COUPLING**

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

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	UNLENG	UNTIME	UNFORC	TIMIDL	FLIPX	FLIPY	FLIPZ	SUBCYL
Type	F	F	F	F	I	I	I	I
Default	1.	1.	1.	0.	0	0	0	1

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

*CONTROL

VARIABLE	DESCRIPTION
SUBCYL	CAL3D/MADYMO3D subcycling interval (# of cycles): EQ.0: Set to 1, EQ.n: number of LS-DYNA time steps between each CAL3D/MADYMO3D step. Then the position of the contacting rigid bodies is assumed to be constant for n LS-DYNA time steps. This may result in some increase in the spikes in contact, thus this option should be used carefully. As the CAL3D/MADYMO3D programs usually work with a very small number of degrees of freedom, not much gain in efficiency can be achieved.

*CONTROL

*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

<u>VARIABLE</u>	<u>DESCRIPTION</u>
NRCYCK	Number of iterations between convergence checks, for dynamic relaxation option (default = 250).
DRTOL	Convergence tolerance for dynamic relaxation option (default = 0.001).
DRFCTR	Dynamic relaxation factor (default = .995).
DRTERM	Optional termination time for dynamic relaxation. Termination occurs at this time or when convergence is attained (default = infinity).
TSSFDR	Scale factor for computed time step during dynamic relaxation. If zero, the value is set to TSSFAC defined on *CONTROL_TIMESTEP. After converging, the scale factor is reset to TSSFAC.
IRELAL	Automatic control for dynamic relaxation option based on algorithm of Papadrakakis [Papadrakakis 1981]: EQ.0: not active, EQ.1: active.
EDTTL	Convergence tolerance on automatic control of dynamic relaxation.

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

Remarks:

1. Stress initialization in LS-DYNA for small strains may be accomplished by linking to an implicit code (option 2). A displacement state is required that gives for each nodal point its label, xyz displacements, xyz rotations and temperature. This data is read from unit 7 (m=) with the format (i8,7e15.0). See also INTRODUCTION, Execution Syntax.
2. If IDRFLG is set to -1 the dynamic relaxation proceeds as normal but time history data is written to the D3THDT file in addition to the normal data being 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

*CONTROL_EFG

Purpose: Define controls for the mesh-free computation.

Card Format

	1	2	3	4	5	6	7	8
Variable	ISPLANE	IDILA	ININT					
Type	I	I	I					
Default	0	0	12					
Remarks			1					

<u>VARIABLE</u>	<u>DESCRIPTION</u>
ISPLINE	Optional choice for the mesh-free kernal functions: EQ.0: Cubic spline function (default) EQ.1: Quadratic spline function
IDILA	Optional choice for the normalized dilation parameter: EQ.0: use individual dilation parameter for each mesh-free particle (default) EQ.1: use the largest dilation parameter; The EFG processor will calculate the largest dilation parameter for each mesh-free part. Every particle associated with that mesh-free part will be assigned to the same value for the mesh-free computation.
ININT	This is the factor needed for the estimation of maximum workspace (MWSPAC) that can be used during the intialization phase.

Remark:

1. The mesh-free computation requires calls to use BCSLIB-EXT solvers during the intialization phase. The maximum workspace (MWSPAC) that can be used during the call is calculated according to:

$$MWSPAC = ININT^{**3} * NUMNEFG$$

where NUMNEFG is the total number of mesh-free nodes. The ININT default value is 12. This value implicitly tells you how many nodes are within the domain of influence per each node in one direction. Increasing this number when the larger normalized dilation parameters are used.

A formula that can automatically compute the required maximum workspace will be updated later.

***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:

<BLANK>

936

which switches the hourglass formulation so that it is identical to that used in version 936 of LS-DYNA. The modification in the hourglass control from version 936 was to ensure that all components of the hourglass force vector are orthogonal to rigid body rotations. However, problems that run under version 936 sometimes lead to different results in versions 940 and later. This difference in results is primarily due to the modifications in the hourglass force vector. Versions released after 936 should be more accurate.

Purpose: Set the default values of the hourglass control to override the default values.

Card Format

	1	2	3	4	5	6	7	8
Variable	IHQ	QH						
Type	I	F						
Default	1	0.1						
Remarks	1							

VARIABLE

DESCRIPTION

IHQ

Default hourglass viscosity type:

- EQ.1: standard LS-DYNA,
- EQ.2: Flanagan-Belytschko integration,
- EQ.3: Flanagan-Belytschko with exact volume integration,
- EQ.4: stiffness form of type 2 (Flanagan-Belytschko),
- EQ.5: stiffness form of type 3 (Flanagan-Belytschko),
- EQ.6: Belytschko-Bindeman [1993] assumed strain co-rotational stiffness form for 2D and 3D solid elements only. This form is available for explicit and IMPLICIT solution methods. In fact, type 6 is mandatory for the implicit options.
- EQ.8: Applicable to the type 16 fully integrated shell element. IHQ=8 activates warping stiffness for accurate solutions. A speed penalty of 25% is common for this option.

*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 except for $IHQ=6$ (see Remark 2).

Remark:

1. Hourglass coefficients and type can be set by part ID in the *HOURGLASS Section.
2. Type 6 hourglass control ($IHQ=6$) is for 2D and 3D solid elements only. Any underintegrated shell element parts that do not have hourglass type defined by *HOURGLASS data will be automatically switched to type 4 hourglass control. If this behavior is not desired, it may be better to use *HOURGLASS to change individual solid parts to type 6 hourglass control. For a more detailed discussion of type 6 hourglass control, please see remark 4 in the *HOURGLASS section.

***CONTROL_IMPLICIT_AUTO**

Purpose: Define parameters for automatic time step control during implicit analysis (see also *CONTROL_IMPLICIT_GENERAL).

Card Format

	1	2	3	4	5	6	7	8
Variable	IAUTO	ITEOPT	ITEWIN	DTMIN	DTMAX	DTEXP		
Type	I	I	I	F	F	F		
Default	0	11	5	DT/1000.	DT*10.	none		

VARIABLE

DESCRIPTION

IAUTO	Automatic time step control flag EQ.0: constant time step size EQ.1: automatically adjusted time step size
ITEOPT	Optimum equilibrium iteration count per time step.
ITEWIN	Allowable iteration window. If iteration count is within ITEWIN iterations of ITEOPT, step size will not be adjusted for the next step.
DTMIN	Minimum allowable time step size. Simulation stops with error termination if time step falls below DTMIN.
DTMAX	Maximum allowable time step size. LT.0: curve ID = (-DTMAX) gives max stepsize as a function of time. Also, the step size is adjusted automatically so that the time value of each point in the curve is reached exactly (see Figure 7.2 and remarks).
DTEXP	Time interval to run in explicit mode before returning to implicit mode. Applies only when automatic implicit-explicit switching is active (IMFLAG= 4 or 5 on *CONTROL_IMPLICIT_GENERAL).

*CONTROL

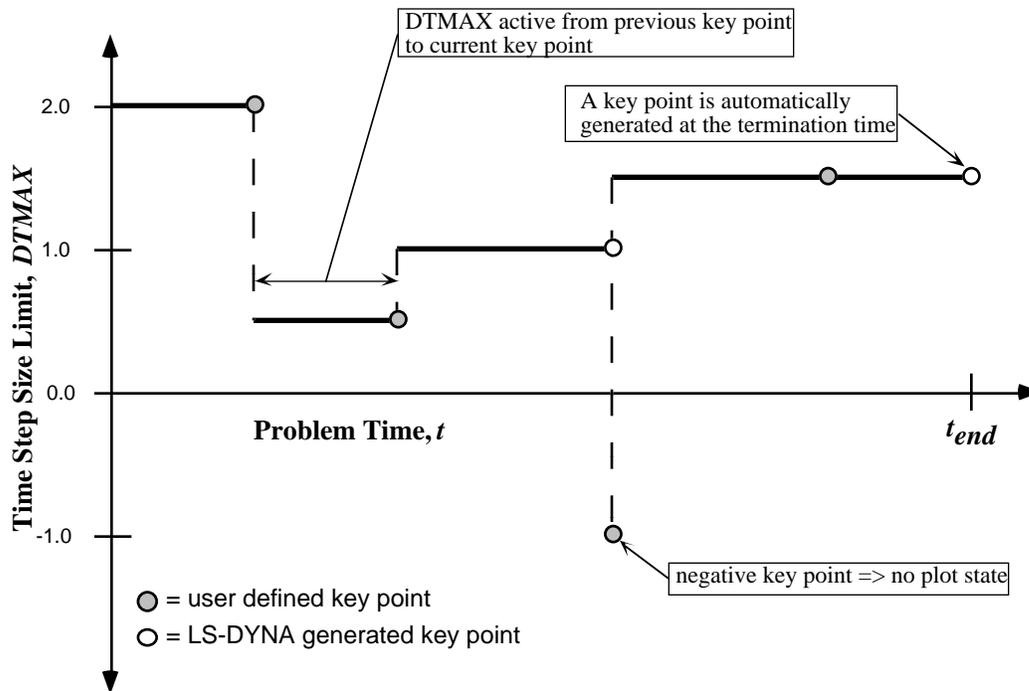


Figure 7.2. A key point curve can be identified using a negative value for $DTMAX$. Function values of each curve point give $DTMAX$. Time values are reached exactly by the automatic step controller, and a plot state is output unless $DTMAX$ is negative.

Remarks:

- IAUTO** The default for IAUTO depends on the analysis type. For “springback” analysis, automatic time step control and artificial stabilization are activated by default.
- ITEOPT** 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=30 or more can be more efficient for highly nonlinear simulations by allowing more iterations in each step, hence fewer total steps.
- ITEWIN** The step size is not adjusted if the iteration count falls within ITEWIN of ITEOPT. Large values of ITEWIN make the controller more tolerant of variations in iteration count.
- DTMAX** To strike a particular simulation time exactly, create a key point curve (Figure 7.2) and enter $DTMAX = -(\text{curve ID})$. This is useful to guarantee that important simulation times, such as when peak load values occur, are reached exactly.
- DTEXP** When the automatic implicit-explicit switching option is activated (IMFLAG = 4 or 5 on *CONTROL_IMPLICIT_GENERAL), the solution method will begin as implicit, and if convergence of the equilibrium iterations fails, automatically switch to explicit for a time interval of DTEXP. A small value of DTEXP should be chosen so that significant dynamic effects do not develop during the explicit phase, since these can make recovery of static equilibrium difficult during the next (static) implicit time step. A reasonable starting value of DTEXP may equal several hundred explicit time steps.

***CONTROL_IMPLICIT_BUCKLE**

Purpose: Activate implicit buckling analysis when termination time is reached (see also *CONTROL_IMPLICIT_GENERAL).

Card Format

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

VARIABLE

DESCRIPTION

NMODE Number of buckling modes to compute
 EQ.0: none (DEFAULT)
 EQ.n: compute n lowest buckling modes

Remarks:

Buckling analysis is performed at the end of a static implicit simulation. The simulation may be linear or nonlinear. After loads have been applied to the model, the buckling eigenproblem is solved:

$$[\mathbf{K}_M + \lambda \mathbf{K}_G] \{u\} = 0$$

where \mathbf{K}_M is the material tangent stiffness matrix, and the geometric or initial stress stiffness matrix \mathbf{K}_G is a function of internal stress in the model. The lowest n eigenvalues and eigenvectors are computed. The eigenvalues, written to text file "eigout", represent multipliers to the applied loads which give buckling loads. The eigenvectors, written to binary database "d3eigv", represent buckling mode shapes. View and animate these modes using LS-PREPOST.

The geometric stiffness terms needed for buckling analysis will be automatically computed when the termination time is reached, regardless of the value of the geometric stiffness flag IGS on *CONTROL_IMPLICIT_GENERAL.

A double precision executable should be used for best accuracy in buckling analysis.

*CONTROL

*CONTROL_IMPLICIT_DYNAMICS

Purpose: Activate implicit dynamic analysis and define time integration constants (see also *CONTROL_IMPLICIT_GENERAL).

Card Format

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

<u>VARIABLE</u>	<u>DESCRIPTION</u>
IMASS	Implicit analysis type EQ.0: static analysis EQ.1: dynamic analysis using Newmark time integration. EQ.2: dynamic analysis by modal superposition
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- β time integration scheme

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

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

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

Here, Δt is the time step size, and β and γ are the free parameters of integration. For $\gamma = \frac{1}{2}$ and $\beta = \frac{1}{4}$ the method reduces to the trapezoidal rule and is energy conserving. If

$$\gamma > \frac{1}{2}$$

$$\beta > \frac{1}{4} \left(\frac{1}{2} + \gamma \right)^2$$

numerical damping is induced into the solution leading to a loss of energy and momentum.

When modal superposition is invoked, NEIGV on *CONTROL_IMPLICIT_EIGENVALUE indicates the number of modes to be used. With modal superposition, stresses are computed only for linear shell formulation 18.

*CONTROL

*CONTROL_IMPLICIT_EIGENVALUE

Purpose: Activate implicit eigenvalue analysis and define associated input parameters (see also *CONTROL_IMPLICIT_GENERAL).

Card Format

	1	2	3	4	5	6	7	8
Variable	NEIG	CENTER	LFLAG	LFTEND	RFLAG	RHTEND	EIGMTH	SHFSCL
Type	I	F	I	F	I	F	I	F
Default	0	0.0	0	-infinity	0	+infinity	2	0.0

VARIABLE

DESCRIPTION

NEIG	Number of eigenvalues to extract. This must be specified. The other parameters below are optional. LT.0: curve ID = (-NEIG) used for intermittent eigenvalue analysis
CENTER	Center frequency. This option finds the nearest NEIG eigenvalues located about this value.
LFLAG	Left end point finite flag. EQ.0: left end point is -infinity EQ.1: left end point is LFTEND.
LFTEND	Left end point of interval. Only used when LFLAG = 1.
RFLAG	Right end point finite flag: EQ.0: right end point is +infinity EQ.1: right end point is RHTEND.
RHTEND	Right end point of interval. Only used when RFLAG = 1.
EIGMTH	Eigenvalue extraction method: EQ.2: Block Shift and Invert Lanczos (default). EQ.3: Lanczos with [M] = [I] (for debug only).
SHFSCL	Shift scale. Generally not used, but see explanation below.

Remarks:

To perform an eigenvalue analysis, activate the implicit method by selecting IMFLAG=1 on *CONTROL_IMPLICIT_GENERAL, and indicate a nonzero value for NEIG above. By default, the lowest NEIG eigenvalues will be found. If a nonzero center frequency is specified, the NEIG eigenvalues nearest to CENTER will be found.

When $NEIG > 0$, eigenvalues will be computed at time=0 and LS-DYNA will terminate.

When $NEIG < 0$, an intermittent eigenvalue analysis will be performed. This is a transient simulation during which loads are applied, with eigenvalues computed periodically during the simulation. Changes in geometry, stress, material, and contact conditions will affect the eigenvalues. The transient simulation can be either implicit or explicit according to $IMFLAG=1$ or $IMFLAG=6$, respectively, on `*CONTROL_IMPLICIT_GENERAL`. The $curveID = -NEIG$ indicates when to extract eigenvalues, and how many to extract. Define one curve point at each desired extraction time, with a function value equal to the number of eigenvalues desired at that time. A `d3plot` database will be produced for the transient solution results. Consecutively numbered `d3eigv` and `eigout` databases will be produced for each intermittent extraction. The extraction time is indicated in each database's analysis title.

The Block Shift and Invert Lanczos code is from BCSLIB-EXT, Boeing's Extreme Mathematical Library.

When using Block Shift and Invert Lanczos, the user can specify a semifinite or finite interval region in which to compute eigenvalues. Setting $LFLAG = 1$ changes the left end point from -infinity to the value specified by $LFTEND$. Setting $RFLAG = 1$ changes the right end point from +infinity to the values given by $RHTEND$. If the interval includes `CENTER` (default value of 0.0) then the problem is to compute the $NEIG$ eigenvalues nearest to `CENTER`. If the interval does not include `CENTER`, the problem is to compute the smallest in magnitude $NEIG$ eigenvalues.

If all of the eigenvalues are desired in an interval where both end points are finite just input a large number for $NEIG$. The software will automatically compute the number of eigenvalues in the interval and lower $NEIG$ to that value. The most general problem specification is to compute $NEIG$ eigenvalues nearest `CENTER` in the interval $[LFTEND,RHTEND]$. Computing the lowest $NEIG$ eigenvalues is equivalent to computing the $NEIG$ eigenvalues nearest 0.0.

For some problems it is useful to override the internal heuristic for picking a starting point for Lanczos shift strategy, that is the initial shift. In these rare cases, the user may specify the initial shift via the parameter `SHFSCL`. `SHFSCL` should be in the range of first few nonzero frequencies.

Eigenvectors are written to an auxiliary binary plot database named "d3eigv", which is automatically created. These can be viewed using a postprocessor in the same way as a standard "d3plot" database. The time value associated with each eigenvector plot is the corresponding circular frequency. A summary table of eigenvalue results is printed to the "eigout" file.

The print control parameter, `LPRINT`, and ordering method parameter, `ORDER`, from the `*CONTROL_IMPLICIT_SOLVER` keyword card also apply to the Block Shift and Invert Eigensolver.

*CONTROL

*CONTROL_IMPLICIT_GENERAL

Purpose: Activate implicit analysis and define associated control parameters. This keyword is required for all implicit analyses.

Card Format

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

<u>VARIABLE</u>	<u>DESCRIPTION</u>
IMFLAG	Implicit/Explicit analysis type flag EQ.0: explicit analysis EQ.1: implicit analysis EQ.2: explicit followed by implicit (activates “seamless” <i>springback</i>) EQ.4: implicit with automatic implicit-explicit switching EQ.5: implicit with automatic switching and mandatory implicit finish EQ.6: explicit with intermittent eigenvalue extraction EQ.-n: curve ID=n gives IMFLAG as a function of time.
DT0	Initial time step size for implicit analysis
IMFORM	Element formulation flag for “seamless” springback (IMFLAG=2 only) EQ.1: switch to fully integrated shell formulation for springback EQ.2: retain original element formulation (default)
NSBS	Number of implicit steps in “seamless” springback (IMFLAG=2 only)
IGS	Geometric (initial stress) stiffness flag EQ.1: include EQ.2: ignore
CNSTN	Indicator for consistent tangent stiffness (solid materials 3 & 115 only): EQ.0: do not use (default) EQ.1: use.
FORM	Fully integrated element formulation (IMFLAG=2 and IMFORM=1 only) EQ.0: type 16 EQ.1: type 6.

Remarks:

IMFLAG The default value 0 indicates a standard explicit analysis will be performed. Using value 1 causes an entirely implicit analysis to be performed. Value 2 is automatically activated when the keyword ***INTERFACE_SPRINGBACK_SEAMLESS** is present, causing the analysis type to switch from explicit to implicit 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. Contact interfaces are automatically disabled during the implicit phase of “seamless” springback analysis.

When the automatic implicit-explicit switching option is activated ($IMFLAG=4$ or 5), the solution method will begin as implicit. If convergence of the equilibrium iterations fails, the solution will automatically switch to explicit for a time interval of $DTEXP$ (see ***CONTROL_IMPLICIT_AUTO**). After this time interval, the solution method will switch back to implicit and attempt to proceed. The implicit simulation may be either static or dynamic. When this feature is used in a static implicit job, simulation time is no longer arbitrary, and must be chosen along with $DTEXP$ in a realistic way to allow efficient execution of any explicit phases. Mass scaling may also be activated (see ***CONTROL_TIMESTEP**), and will apply only during the explicit phases of the calculation. In cases where much switching occurs, users must exercise caution to ensure that negligible dynamic effects are introduced by the explicit phases.

When $IMFLAG=5$, the final step of the simulation must be implicit. The termination time will be extended automatically as necessary, until a successfully converged implicit step can be obtained. This is useful for example in difficult metal forming springback simulations.

When $IMFLAG=6$, an explicit simulation will be performed. Eigenvalues will be extracted intermittently according to a curve indicated by $NEIG=(-curveID)$ on ***CONTROL_IMPLICIT_EIGENVALUE**. Beware that dynamic stress oscillations which may occur in the explicit simulation will influence the geometric (initial stress) stiffness terms used in the eigen solution, potentially producing misleading results and/or spurious modes. As an alternative, eigenvalues can also be extracted intermittently during an implicit analysis, using $IMFLAG=1$ and $NEIG=(-curveID)$.

When $IMFLAG < 0$, a curve ID is indicated which gives the solution method as a function of time. Define a curve value of zero during explicit phases, and a value of one during implicit phases. Use steeply sloping sections between phases. An arbitrary number of formulation switches may be activated with this method.

DT0 This parameter selects the initial time step size for the implicit phase of a simulation. The step size may change during a multiple step simulation if the automatic time step size control feature is active (see ***CONTROL_IMPLICIT_AUTO**.)

IMFORM Adaptive mesh must be activated when using element formulation switching. For best springback accuracy, use of shell type 16 is recommended during the entire stamping and springback analysis, in spite of the increased cost of using this element during the explicit stamping phase.

NSBS The **NSBS** option allows a “seamless” springback analysis to use multiple unloading steps (***CONTROL_IMPLICIT_STABILIZATION** is also required in this case).

IGS The geometric stiffness adds the effect of initial stress to the global stiffness matrix. This effect is seen in a piano string whose natural frequency changes with tension. Geometric stiffness does not always improve nonlinear convergence, especially when compressive stresses are present, so its inclusion is optional.

*CONTROL

*CONTROL_IMPLICIT_MODES

Purpose: Request calculation of constraint and/or attachment modes for later use in modal analysis using *PART_MODES (see also *CONTROL_IMPLICIT_GENERAL).

Card Format

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

<u>VARIABLE</u>	<u>DESCRIPTION</u>
NSIDC	Node set ID for constraint modes EQ.0: no constraint modes will be generated
NSIDA	Node set ID for attachment modes EQ.0: no attachment modes will be generated

Remarks:

To use this feature, an implicit analysis must be requested using IMFLAG=1 on *CONTROL_IMPLICIT_GENERAL, and a non-zero termination time must be specified on *CONTROL_TERMINATION. A double precision version of LS-DYNA should be used for best accuracy. Care must be taken to apply a sufficient number of constraints to the model to eliminate static rigid body motion. Computed modes are written to binary output file d3mode, which can be viewed using LS-PREPOST.

Constraint and attachment modes are generated by applying unit displacements and unit forces, respectively, to each specified degree of freedom. By default, modes are computed for all degrees of freedom for each node in sets NSIDC and NSIDA. The first and second node set attribute parameters can be optionally used to restrict the translational and rotational degrees of freedom for which modes are requested, respectively, according to the following syntax:

Node set attribute parameters DA1 and A1: translational degree of freedom codes

Node set attribute parameters DA2 and A2: rotational degree of freedom codes

<u>code</u>	<u>modes computed</u>
0	X, Y, Z degrees of freedom (DEFAULT)
1	X degree of freedom only
2	Y degree of freedom only
3	Z degree of freedom only
4	X, Y degrees of freedom only
5	Y, Z degrees of freedom only
6	X, Z degrees of freedom only
7	X, Y, Z degrees of freedom

*CONTROL

*CONTROL_IMPLICIT_SOLUTION

Purpose: These optional cards apply to implicit calculations. Use these cards to specify whether a linear or nonlinear solution is desired. Parameters are also available to control the implicit nonlinear and arc length solution methods (see also *CONTROL_IMPLICIT_GENERAL).

Card 1 Format

1 2 3 4 5 6 7 8

Variable	NSOLVR	ILIMIT	MAXREF	DCTOL	ECTOL	RCTOL	LSTOL	
Type	I	I	I	F	F	F	F	
Default	2	11	15	0.001	0.01	1.0e+10	0.90	

Optional Card 2

1 2 3 4 5 6 7 8

Variable	DNORM	DIVERG	ISTIF	NLPRINT	NLNORM			
Type	I	I	I	I	I			
Default	2	1	1	0	2			

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

1 2 3 4 5 6 7 8

Variable	ARCCTL	ARCDIR	ARCLEN	ARCMTH	ARCDMP			
Type	I	I	F	I	I			
Default	0	none	0	1	2			

(see remarks below)

VARIABLE	DESCRIPTION								
NSOLVR	Solution method for implicit analysis: EQ.1: Linear EQ.2: Nonlinear with BFGS updates (default) EQ.3: Nonlinear with Broyden updates EQ.4: Nonlinear with DFP updates EQ.5: Nonlinear with Davidon updates EQ.6: Nonlinear with BFGS updates + arclength EQ.7: Nonlinear with Broyden updates + arclength EQ.8: Nonlinear with DFP updates + arclength EQ.9: Nonlinear with Davidon updates + arclength								
ILIMIT	Iteration limit between automatic stiffness reformations								
MAXREF	Stiffness reformation limit per time step								
DCTOL	Displacement convergence tolerance								
ECTOL	Energy convergence tolerance								
RCTOL	Residual (force) convergence tolerance (DEFAULT=inactive)								
LSTOL	Line search convergence tolerance								
DNORM	Displacement norm for convergence test EQ.1: Increment vs. displacement over current step EQ.2: Increment vs. total displacement (default)								
DIVERG	Divergence flag (force imbalance increase during equilibrium iterations) EQ.1: reform stiffness if divergence detected (default) EQ.2: ignore divergence								
ISTIF	Initial stiffness formation flag EQ.1: reform stiffness at start of each step (default) EQ.n: reform stiffness at start of every "n"th step								
NLPRINT	Nonlinear solver print flag EQ.0: no nonlinear iteration information printed (new v970 default) EQ.1: print iteration information to screen, messag, d3hsp files EQ.2: print extra norm information (NLNORM=1 or 3 only) NOTE: during execution, interactive commands can be used: <table><thead><tr><th><u>interactive command</u></th><th><u>response</u></th></tr></thead><tbody><tr><td><ctrl-c> nlprint</td><td>toggle NLPRINT between 0 and 1</td></tr><tr><td><ctrl-c> diagnostic</td><td>toggle NLPRINT between 0 and 2</td></tr><tr><td><ctrl-c> information</td><td>set NLPRINT=2 for one iteration</td></tr></tbody></table>	<u>interactive command</u>	<u>response</u>	<ctrl-c> nlprint	toggle NLPRINT between 0 and 1	<ctrl-c> diagnostic	toggle NLPRINT between 0 and 2	<ctrl-c> information	set NLPRINT=2 for one iteration
<u>interactive command</u>	<u>response</u>								
<ctrl-c> nlprint	toggle NLPRINT between 0 and 1								
<ctrl-c> diagnostic	toggle NLPRINT between 0 and 2								
<ctrl-c> information	set NLPRINT=2 for one iteration								
NLNORM	Nonlinear convergence norm type EQ.1: consider translational and rotational degrees of freedom EQ.2: consider translational degrees of freedom only (default) EQ.3: same as 2, but compute extra information for NLPRINT								

*CONTROL

<u>VARIABLE</u>	<u>DESCRIPTION</u>
	<i>The following parameters are for use with arc length methods only ($6 \leq \text{NSOLVR} \leq 9$):</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
ARCLLEN	Relative arc length size. See remarks below. LE.0.0: use automatic size size, GT.0.0: use ARCLLEN*automatic step size.
ARCMTH	Arc length method EQ.1: Crisfield (default) EQ.2: Ramm
ARCDMP	Arc length damping option EQ.2: off (default) EQ.1: on, oscillations in static solution are suppressed

Remarks:

NSOLVR	<p>If a linear analysis is selected, equilibrium checking and iterations are not performed.</p> <p>The Full Newton nonlinear solution method can be invoked by using the default BFGS solver, and selecting ILIMIT=1 to form a new stiffness matrix every iteration.</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 method is frequently used to solve snap through buckling problems. <u>When applying the arc-length method, the curves that define the loading should contain only two points, and the first point should be at the origin (0,0).</u> LS-DYNA will extrapolate, if necessary, to determine the load. In this way, time and load magnitude are related by a constant. It is possible that time can become negative in case of load reversal. The arc length method cannot be used in a dynamic analysis.</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 norm ratio is reduced below DCTOL, this condition is satisfied. Smaller numbers lead to more accurate determination of equilibrium and, on the negative side, result in more iterations and higher costs. Use NLPRINT to display norm data each iteration.
ECTOL	When the energy norm ratio is reduced below ECTOL, this condition is satisfied. Smaller numbers lead to more strict determination of equilibrium and, on the negative side, result in more iterations and higher costs. Use NLPRINT to display norm data each iteration.
RCTOL	When the residual norm ratio is reduced below RCTOL, this condition is satisfied. Smaller numbers lead to more strict determination of equilibrium and, on the negative side, result in more iterations and higher costs. By default this convergence criterion is effectively disabled using RCTOL=1.e10. Use NLPRINT to display norm data each iteration.
LSTOL	A line search is performed on stiffening systems to guard against divergence of Newton-based nonlinear solvers. With the Full Newton method, it is sometimes helpful to define a large value (LSTOL=9999.0) to effectively disable line search.
DNORM	When computing the displacement ratio, the norm of the incremental displacement vector is divided by the norm of “total” displacement. This “total” displacement may be either the total over the current step, or the total over the entire simulation. The latter tends to be more lax, and can be poor at the end of simulations where large motions develop. For these problems, an effective combination is DNORM=1, and DCTOL=0.01 or larger.
DIVERG	By default, a new stiffness matrix is formed whenever divergence (growing out-of-balance force) is detected. This flag can be used to suppress this stiffness reformation.
ISTIF	By default, a new stiffness matrix is formed at the start of every time step. Suppressing this stiffness reformation can decrease the cost of simulations which have many tiny steps that are mostly linear, such as transient dynamics.
NLPRINT	This flag controls printing of displacement and energy convergence measures during the nonlinear equilibrium search. If convergence difficulty occurs, this information is helpful in determining the problem.
NLNORM	By default, only translational degrees of freedom are used in evaluating convergence norms. Use this flag to include rotational degrees of freedom, or to make additional data available for diagnosing convergence problems. This additional data includes the worst offending node and degree of freedom contributing to each norm.
ARCCTL	The arc length method can be controlled based on the displacement of a single node in the model. For example, in dome reversal problems the node at the center of the dome can be used. By default, the generalized arc length method is used, where the norm of the global displacement vector controls the solution. This includes all nodes.
ARCLLEN	In many cases the arc length method has difficulty tracking the load displacement curve through critical regions. Using $0 < \text{ARCLLEN} < 1$ will reduce the step size to assist tracking the load-displacement curve with more accuracy. Use of $\text{ARCLLEN} < 1$ will cause more steps to be taken. Suggested values are 1.0 (the default), 0.5, 0.25, and 0.10.
ARCDMP	Some static problems exhibit oscillatory response near instability points. This option numerically suppresses these oscillations, and may improve the convergence behavior of the post-buckling solution.

*CONTROL

*CONTROL_IMPLICIT_SOLVER

Purpose: These optional cards apply to implicit calculations. The linear equation solver performs the CPU-intensive stiffness matrix inversion (see also *CONTROL_IMPLICIT_GENERAL).

Card Format

	1	2	3	4	5	6	7	8
Variable	LSOLVR	LPRINT	NEGEV	ORDER	DRCM	DRCPRM	AUTOSPC	AUTOTOL
Type	I	I	I	I	I	F	I	F
Default	4	0	2	0	1	see below	1	see below

Optional Card 2

1

Variable	LCPACK							
Type	I							
Default	2							

(see remarks below)

VARIABLE

DESCRIPTION

LSOLVR

Linear equation solver method

EQ.4: SMP parallel multi-frontal sparse solver (default).

EQ.5: SMP parallel multi-frontal sparse solver, double precision

EQ.6: BCSLIB-EXT, direct, sparse, double precision

EQ.10: iterative, best of currently available iterative methods

EQ.11: iterative, Conjugate Gradient method

EQ.12: iterative, CG with Jacobi preconditioner

EQ.13: iterative, CG with Incomplete Choleski preconditioner

EQ.14: iterative, Lanczos method

EQ.15: iterative, Lanczos with Jacobi preconditioner

EQ.16: iterative, Lanczos with Incomplete Choleski preconditioner

VARIABLE	DESCRIPTION
LPRINT	Linear solver print flag controls screen and message file output EQ.0: no printing EQ.1: output summary statistics on memory, cpu requirements EQ.2: more statistics EQ.3: even more statistics and debug checking (NOTE: during execution, use the interactive command "<ctrl-c> lprint" to toggle this print flag between 0 and 1.)
NEGEV	Negative eigenvalue flag. Selects procedure when negative eigenvalues are detected during stiffness matrix inversion. EQ.1: stop, or retry step if auto step control is active EQ.2: print warning message, try to continue (default)
ORDER	Ordering option EQ.0: method set automatically by LS-DYNA EQ.1: MMD, Multiple Minimum Degree. EQ.2: Metis
DRCM	Drilling rotation constraint method: EQ.1: add stiffness (default) NE.1: add no stiffness
DRCPRM	Drilling rotation constraint parameter, DRCPRM. If adding stiffness, DRCM=1, then, for linear problems, DRCPRM=1.0; for nonlinear problems, DRCPRM=100.0; and for eigenvalue problems either 1.E-12 or 1.E-8 is used depending on the shell element type. In the latter case, the input value for DRCPRM is ignored.
AUTOSPC	Automatic Constraint Scan flag EQ.1: scan the assembled stiffness matrix looking for unconstrained, unattached degrees of freedom. Generate additional constraints as necessary to avoid negative eigenvalues. EQ.2: do not add constraints.
AUTOTOL	AUTOSPC tolerance. The test for singularity is the ratio of the smallest singular value and the largest singular value. If this ratio is less than AUTOTOL, then the triple of columns are declared singular and a constraint is generated. Default value in single precision is 1.E-4 and in double precision, 1.E-8.
LCPACK	Matrix assembly package. The old assembly package offers severely limited functionality, and supports only a few simple constraint features. This option is primarily used for debugging purposes. EQ.1: Use old assembly package from v960 EQ.2: Use v970's LCPACK (default)

*CONTROL

Remarks:

- LSOLVR The linear solver is used to compute the inverse of the global stiffness matrix, which is a costly procedure both in memory and cpu time. Direct solvers apply Gaussian elimination, while iterative solvers successively improve “guesses” at the correct solution. Iterative solvers require far less memory than direct solvers, but may suffer from convergence problems. Generally, iterative solvers are poor for automotive applications, but can be superior for large brick element soil models in civil engineering.
- Solvers 5 and 6 promote the global matrix to double precision before factoring to reduce numerical truncation error. Solvers 4 and 5 are equivalent if a double precision executable is used.
- Solver 6 is the direct linear equation solver from BCSLIB-EXT, Boeing's Extreme Mathematical Library. This option should be used whenever the factorization is too large to fit into memory. It has extensive capabilities for out-of-core solution and can solve larger problems than any of the other direct factorization methods. Solver 6 also includes a sophisticated pivoting strategy which can be superior for nearly singular matrices.
- LPRINT Select printing of the timing and storage information (LPRINT = 1) if you are comparing performance of linear equation solvers, or if you are running out of memory for large models. Minimum memory requirements for in-core and out-of-core solution are printed. This flag can also be toggled using sense switch "<ctrl-c> lprint". *For best performance, increase available memory using “memory=” on the command line until an IN-CORE solution is indicated.*
- When using solver option 6, LPRINT = 2 and 3 will cause increased printed output of statistics and performance information.
- NEGEV Negative eigenvalues result from underconstrained models (rigid body modes), severely deformed elements, or non-physical material properties. This flag allows control to be passed directly to the automatic time step controller when negative eigenvalues are detected. Otherwise, significant numerical roundoff error is likely to occur during factorization, and equilibrium iterations may fail (see *CONTROL_IMPLICIT_AUTO).
- ORDER The system of linear equations is reordered to optimize the sparsity of the factorization when using direct methods. Metis is a ordering method from University of Minnesota which is very effective for larger problems and for 3D solid problems, but also very expensive. MMD is inexpensive, but may not produce an optimum reordering, leading to higher cost during numeric factorization. MMD is usually best for smaller problems (less than 100,000 degrees of freedom).
- Reordering cost is included in the symbolic factorization phase of the linear solver (LPRINT.ge.1). For large models, if this cost exceeds 20% of the numeric factorization cost, it may be more efficient to select the MMD method.
- Note that the values of LPRINT and ORDER also affect the eigensolution software. That is LPRINT and ORDER from this keyword card is applicable to eigensolution.

***CONTROL_IMPLICIT_STABILIZATION**

Purpose: This optional card applies to implicit calculations. Artificial stabilization is required for multi-step unloading in implicit springback analysis (see also *CONTROL_IMPLICIT_GENERAL).

Card Format

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

<u>VARIABLE</u>	<u>DESCRIPTION</u>
IAS	Artificial Stabilization flag EQ.1: active EQ.2: inactive (default)
SCALE	Scale factor for artificial stabilization. For flexible parts with large springback, like outer body panels, a value of 0.001 may be required. EQ.-n: curveID = n gives SCALE as a function of time
TSTART	Start time. (Default: immediately upon entering implicit mode)
TEND	End time. (Default: termination time)

Remarks:

Artificial stabilization allows springback to occur over several steps. This is often necessary to obtain convergence during equilibrium iterations on problems with large springback deformation. Stabilization is introduced at the start time TSTART, and slowly removed as the end time TEND is approached. Intermediate results are not accurate representations of the fully unloaded state. The end time TEND must be reached exactly for total springback to be predicted accurately.

IAS	The default for IAS depends on the analysis type in *CONTROL_IMPLICIT_GENERAL. For “seamless” springback analysis, automatic time step control and artificial stabilization are activated by default. Otherwise, IAS is inactive by default.
SCALE	This is a penalty scale factor similar to that used in contact interfaces. If modified, it should be changed in order-of-magnitude increments at first. Large values suppress springback deformation until very near the termination time, making convergence during the first few steps easy. Small values may not stabilize the solution enough to allow equilibrium iterations to converge.

***CONTROL**

***CONTROL_MPP_DECOMPOSITION_AUTOMATIC**

Purpose: Instructs the program to apply a simple heuristic to try to determine the proper decomposition for the simulation.

There are no input parameters. The existence of this keyword triggers the automated decomposition. This option should not be used if there is more than one occurrence of any of the following options in the model:

- *INITIAL_VELOCITY
- *CHANGE_VELOCITY
- *BOUNDARY_PRESCRIBED_MOTION
- *BOUNDARY_PRESCRIBED_CFD

And the following control card must not be used:

- *CONTROL_MPP_DECOMPOSITION_TRANSFORMATION

For the general case, it is recommended that you specify the proper decomposition using the command *CONTROL_MPP_DECOMPOSITION_TRANSFORMATION instead.

*CONTROL_MPP_DECOMPOSITION_CHECK_SPEED

Purpose: Modifies the decomposition depending on the relative speed of the processors involved.

There are no input parameters. The existence of this keyword causes a short floating point timing routine to be executed on each processor. The information gathered is used during the decomposition, with faster processors being given a relatively larger portion of the problem. This option is not recommended on homogeneous systems.

*CONTROL

*CONTROL_MPP_DECOMPOSITION_CONTACT_DISTRIBUTE

Purpose: Ensures that the indicated contact interfaces are distributed across all processors, which can lead to better load balance for large contact interfaces.

Card Format

	1	2	3	4	5	6	7	8
Variable	ID1	ID2	ID3	ID4	ID5			
Type	I	I	I	I	I			
Default	None	None	None	None	None			

<u>VARIABLE</u>	<u>DESCRIPTION</u>
ID1	First contact interface ID to distribute. If no contact ID's are specified, the number given here corresponds to the order of the interfaces as they appear in the input, with the first being 1.
ID2, ID3, ID4, ID5	Remaining interfaces ID's to distribute.

Remarks:

Up to 5 contact interface ID's can be specified. The decomposition is modified as follows: First, all the elements involved in the first contact interface are decomposed across all the processors. Then all the elements involved in the second contact interface (excluding any already assigned to processors) are distributed, and so on. After all the contact interfaces given are processed, the rest of the input is decomposed in the normal manner. This will result in each processor having possibly several disjoint portions of the input assigned to it, which will increase communications somewhat. However, this can be offset by improved load balance in the contact. It is generally recommended that at most one or two interfaces be specified, and then only if they are of substantial size relative to the whole problem.

***CONTROL_MPP_DECOMPOSITION_CONTACT_ISOLATE**

Purpose: Ensures that the indicated contact interfaces are isolated on a single processor, which can lead to decreased communication.

Card Format

	1	2	3	4	5	6	7	8
Variable	ID1	ID2	ID3	ID4	ID5			
Type	I	I	I	I	I			
Default	None	None	None	None	None			

VARIABLE

DESCRIPTION

ID1	First contact interface ID to isolate. If no contact ID's are specified, the number given here corresponds to the order of the interfaces as they appear in the input, with the first being 1.
ID2, ID3, ID4, ID5	Remaining interfaces ID's to isolate.

Remarks:

Up to 5 contact interfaces can be specified. The decomposition is modified as follows: First, all the elements involved in the first contact interface ID are assigned to the first processor. Then all the elements involved in the second contact interface ID (excluding any already assigned to processors) are assigned to the next processor, and so on. After all the contact interfaces given are processed, the rest of the input is decomposed in the normal manner. This will result in each of the interfaces being processed on a single processor. For small contact interfaces this can result in better parallelism and decreased communication.

*CONTROL

*CONTROL_MPP_DECOMPOSITION_FILE

Purpose: Allow for pre-decomposition and a subsequent run or runs without having to do the decomposition.

Card Format

1 2 3 4 5 6 7 8

Variable	Name							
Type	A80							
Default	None							

VARIABLE

DESCRIPTION

NAME

Name of a file containing (or to contain) a decomposition record.

Remarks:

If the indicated file does not exist, it is created with a copy of the decomposition information from this run. If the file exists, it is read and the decomposition steps can be skipped. The original run that created the file must be for a number of processors that is a multiple of the number of processors currently being used. Thus, a problem can be decomposed once for, say, 48 processors. Subsequent runs are then possible on any number that divides 48: 1, 2, 3, 4, 6, etc. Since the decomposition phase generally requires more memory than execution, this allows large models to be decomposed on one system and run on another (provided the systems have compatible binary formats). The file extension “.pre” is added automatically.

***CONTROL_MPP_DECOMPOSITION_METHOD**

Purpose: Specify the decomposition method to use.

Card Format

	1	2	3	4	5	6	7	8
Variable	Name							
Type	A80							
Default	RCB							

VARIABLE**DESCRIPTION**

NAME

Name of the decomposition method to use. There are currently two options:

RCB = recursive coordinate bisection

GREEDY = a simple heuristic method

In almost all cases the RCB method is superior and should be used.

*CONTROL

*CONTROL_MPP_DECOMPOSITION_NUMPROC

Purpose: Specify the number of processors for decomposition.

Card Format

1 2 3 4 5 6 7 8

Variable	N							
Type	I							
Default	None							

VARIABLE

DESCRIPTION

N

Number of processors for decomposition.

Remarks:

This is used in conjunction with the CONTROL_MPP_DECOMPOSITION_FILE command to allow for later runs on different numbers of processors. By default, the decomposition is performed for the number of processors currently being used. However, a different value can be specified here. If $N > 1$ and only one processor is currently being used, the decomposition is done and then the program terminates. Similarly, if N is NOT a multiple of the current number of processors, the execution terminates after decomposition. Otherwise, the decomposition is performed for N processors, and the execution continues.

***CONTROL_MPP_DECOMPOSITION_RCBLOG**

Purpose: Eliminate decomposition based variations between runs of similar models..

Card Format

	1	2	3	4	5	6	7	8
Variable	Name							
Type	A80							
Default	None							

<u>VARIABLE</u>	<u>DESCRIPTION</u>
NAME	Name of a file containing (or to contain) a decomposition record.

Remarks:

If the indicated file does not exist, it is created with a copy of the decomposition information from this run. If the file exists, it is read and its contents drive the decomposition process. The resulting decomposition will be spatially identical to the one that generated the file. The intention here is that if, say, one small portion of a model is modified or remeshed for a subsequent run, the subsequent run will have a decomposition identical to the original wherever possible. Without this option, such changes to the input can result in every processor having a slightly different set of elements. This option is incompatible with the CONTROL_MPP_DECOMPOSITION_CONTACT_DISTRIBUTE and CONTROL_MPP_DECOMPOSITION_CONTACT_ISLOATE options and should only be used if the decomposition method RCB is used, which is the default (see CONTROL_MPP_DECOMPOSITION_METHOD)

***CONTROL**

***CONTROL_MPP_DECOMPOSITION_SHOW**

Purpose: Allows display of the final decomposition. There are no input parameters. The existence of this keyword causes the d3plot file to be modified so that all elements belonging to the first processor have material type 1, those on the second processor type 2, and so on. Execution terminates immediately after the decomposition phase, and no simulation is performed. This can be used in conjunction with the CONTROL_MPP_DECOMPOSITION_NUMPROC command to run on 1 processor and produce a d3plot file to visualize the resulting decomposition.

***CONTROL_MPP_DECOMPOSITION_TRANSFORMATION**

Purpose: Specifies transformations to apply to modify the decomposition.

There are 10 different transformations that can be applied. The input is described here. For a detailed description of each decomposition transformation, see the description in the Appendix for the "pfile".

Any number of transformations can appear with no need for further *CONTROL cards – all non-comment cards up to the next control card are expected to be decomposition transformations.

The first 6 transformations each take one parameter:

Card Format

	1	2	3	4	5	6	7	8
Variable	TYPE	VAL						
Type	A10	F						
Default	None	0.0						

VARIABLE**DESCRIPTION**

TYPE	Which transformation to apply. The possible values are: RX, RY, RZ, SX, SY, SZ
VAL	The amount of scaling/rotation to apply.

The remaining 4 transformations each take 9 parameters:

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	TYPE	V1	V2	V3	V4	V5	V6	
Type	A10	F	F	F	F	F	F	
Default	None	0.0	0.0	0.0	0.0	0.0	0.0	

*CONTROL

Card 2 1 2 3 4 5 6 7 8

Variable	V7	V8	V9					
Type	F	F	F					
Default	0.0	0.0	0.0					

VARIABLE

DESCRIPTION

TYPE Which transformation to apply. The possible values are: VEC3, C2R, S2R, MAT

V1-V9 Parameters to the transformation.

***CONTROL_MPP_IO_NOD3DUMP**

Purpose: Suppresses the output of all dump files.

There are no input parameters. The existence of this keyword causes the d3dump and runrsf file output routines to be skipped.

***CONTROL**

***CONTROL_MPP_IO_NODUMP**

Purpose: Suppresses the output of all dump files and full deck restart files.

There are no input parameters. The existence of this keyword causes the d3dump and runrsf file output routines to be skipped. It also suppresses output of the full deck restart file d3full.

***CONTROL_MPP_IO_NOFULL**

Purpose: Suppresses the output of the full deck restart files.

There are no input parameters. The existence of this keyword suppresses the output of the full deck restart file "d3full"

***CONTROL**

***CONTROL_MPP_IO_SWAPBYTES**

Purpose: Swap bytes on some of the output files.

There are no input parameters. The existence of this keyword causes the d3plot file and the “interface component analysis” file to be output with bytes swapped. This is to allow further processing of data on a different machine that has big endian vs. little endian incompatibilities compared to the system on which the analysis is running.

***CONTROL_NONLOCAL**

Purpose: Allocate additional memory for *MAT_NONLOCAL option.

Card Format

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

VARIABLE

DESCRIPTION

MEM

Percentage increase of memory allocated for MAT_NONLOCAL option over that required initially. This is for additional storage that may be required due to geometry changes as the calculation proceeds. Generally, a value of 10 should be sufficient.

*CONTROL

*CONTROL_OUTPUT

Purpose: Set miscellaneous output parameters. This keyword does not control the information, such as the stress and strain tensors, which is written into the binary databases. For the latter, see the keyword *DATABASE_EXTENT_BINARY.

Card Format

1 2 3 4 5 6 7 8

Variable	NPOPT	NEECHO	NREFUP	IACCOP	OPIFS	IPNINT	IKEDIT	IFLUSH
Type	I	I	I	I	F	I	I	I
Default	0	0	0	0	0.	0	100	5000

Optional Card Format

1 2 3 4 5 6 7 8

Variable	IPRTF							
Type	I							
Default	0							

VARIABLE

DESCRIPTION

NPOPT

Print suppression during input phase flag for the printed output file:
 EQ.0: no suppression,
 EQ.1: nodal coordinates, element connectivities, rigid wall definitions
 and initial velocities are not printed.

NEECHO

Print suppression during input phase flag for echo file:
 EQ.0: all data printed,
 EQ.1: nodal printing is suppressed,
 EQ.2: element printing is suppressed,
 EQ.3: both node and element printing is suppressed.

VARIABLE	DESCRIPTION
NREFUP	Flag to update reference node coordinates for beam elements. This option requires that each reference node is unique to the beam: EQ.0: no update, EQ.1: update.
IACCP	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, EQ.2: built-in, user-defined filtering With this option the keyword parameter, DT2MS, on *CONTROL_TIMESTEP must be defined. All data points between output intervals are stored and used to obtain the filtered output values. The user defined filter must be provided and linked. The procedure for handling is not yet defined.
OPIFS	Output interval for interface file (Δt), see INTRODUCTION, Execution syntax.
IPNINT	Print initial time step sizes for all elements on the first cycle: EQ.0: 100 elements with the smallest time step sizes are printed. EQ.1: the governing time step sizes for each element are printed.
IKEDIT	Problem status report interval steps to the D3HSP (printed output) file. This flag is ignored if the GLSTAT file is written, see *DATABASE_GLSTAT.
IFLUSH	Number of time steps interval for flushing I/O buffers. The default value is 5000. If the I/O buffers are not emptied and an abnormal termination occurs, the output files can be incomplete. The I/O buffers for restart files are emptied automatically whenever a restart file is written so these files are not affected by this option.
IPRTF	Default print flag for RBDOUT and MATSUM files. This flag defines the default value for the print flag which can be defined in the part definition section, see *PART. This option is meant to reduce the file sizes by eliminating data which is not of interest. EQ.0: write part data into both MATSUM and RBDOUT EQ.1: write data into RBDOUT file only EQ.2: write data into MATSUM file only EQ.3: do not write data into RBDOUT and MATSUM

*CONTROL

*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				

<u>VARIABLE</u>	<u>DESCRIPTION</u>
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.
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

*CONTROL_REMESHING

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

Card Format

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

<u>VARIABLE</u>	<u>DESCRIPTION</u>
RMIN	Minimum edge length for the surface mesh surrounding the parts which should be remeshed.
RMAX	Maximum edge length for the surface mesh surrounding the parts which should be remeshed.

Remarks:

1. The value of RMIN and RMAX should be of the same order. The value of RMAX can be set to 2-5 times greater than RMIN.

***CONTROL_RIGID**

Purpose: Special control options related to rigid bodies and the rigid-flexible bodies, see *PART_MODES.

Card Format

	1	2	3	4	5	6	7	8
Variable	LMF	JNTF	ORTHMD	PARTM	SPARSE	METALF		
Type	I	I	I	I	I	I		
Default	0	0	0	0	0	0		

VARIABLE

DESCRIPTION

LMF	Switch the explicit rigid body joint treatment to an implicit formulation which uses Lagrange multipliers to impose prescribed kinematic boundary conditions and joint constraints. This is a new option which is under development in version 970. There is a slight cost overhead due to the assembly of sparse matrix equations which are solved using standard procedures for nonlinear problems in rigid multi-body dynamics. Lagrange multiplier flag: EQ.0: explicit penalty formulation, EQ.1: implicit formulation with Lagrange multipliers.
JNTF	Generalized joint stiffness formulation; see remark 1 below: EQ.0: incremental update, EQ.1: total formulation (exact).
ORTHMD	Orthogonalize modes with respect to each other: EQ.0: true. EQ.1: false, the modes are already orthogonalized.
PARTM	Use global mass matrix to determine part mass distribution. This mass matrix may contain mass from other parts that share nodes. See remark 2 below. EQ.0: true, EQ.1: false.
SPARSE	Use sparse matrix multiply subroutines for the modal stiffness and damping matrices. See remark 3. EQ.0: false, do full matrix multiplies (frequently faster), EQ.1: true.

*CONTROL

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MATELF	Metalforming option, which should not be used for crash and other applications involving rigid bodies. Use fast update of rigid body nodes. If this option is active the rotational motion of all rigid bodies should be suppressed. EQ.0: full treatment is used EQ.1: fast update for metalforming applications

Remarks:

1. As the default, the calculation of the relative angles between two coordinate systems is done incrementally. This is an approximation, in contrast to the total formulation where the angular offsets are computed exactly. The disadvantage of the latter approach is that a singularity exists when an offset angle equals 180 degrees. For most applications, the stop angles prevents this occurrence and JNTF=1 should not cause a problem.
2. If the determination of the normal modes included the mass from both connected bodies and discrete masses, or if there are no connected bodies, then the default is preferred. When the mass of a given part ID is computed, the resulting mass vector includes the mass of all rigid bodies that are merged to the given part ID, but does not included discrete masses. See the keyword: *CONSTRAINED_RIGID_BODIES. A lumped mass matrix is always assumed.
3. Sparse matrix multiplies save a substantial number of operations if the matrix is truly sparse. However, the overhead will slow the multiplies for densely populated matrices.

***CONTROL_SHELL**

Purpose: Provide controls for computing shell response.

Card Format

	1	2	3	4	5	6	7	8
Variable	WRPANG	ESORT	IRNXX	ISTUPD	THEORY	BWC	MITER	PROJ
Type	F	I	I	I	I	I	I	I
Default	20.	0	-1	0	2	2	1	0

Optional Card

	1	2	3	4	5	6	7	8
Variable	ROTASCL	INTGRD	LAMSHT	CSTYP6	TSHELL	NFAIL1	NFAIL4	
Type	F	I	I	I	I	I	I	
Default	1..	0	0	1	0	inactive	inactive	

VARIABLE

DESCRIPTION

WRPANG	Shell element warpage angle in degrees. If a warpage greater than this angle is found, a warning message is printed. Default is 20 degrees.
ESORT	Automatic sorting of triangular shell elements to treat degenerate quadrilateral shell elements as C0 triangular shells, see option THEORY below: EQ.0: no sorting required (default). EQ.1: full sorting,
IRNXX	Shell normal update option. This option affects the Hughes-Liu, Belytschko-Wong-Chiang, and the Belytschko-Tsay shell 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 element types 1, 6, and 7. 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,

*CONTROL

<u>VARIABLE</u>	<u>DESCRIPTION</u>
	EQ.0: default set to -1, EQ.1: compute on restarts, EQ.n: compute every n cycles (Hughes-Liu shells only).
ISTUPD	Shell thickness change option. This option affects all shell element formulations: EQ.0: no change. EQ.1: membrane straining causes thickness change. This option is very important in sheet metal forming or whenever membrane stretching is important.
THEORY	Default shell theory: EQ.1: Hughes-Liu, EQ.2: Belytschko-Tsay (default), EQ.3: BCIZ triangular shell (not recommended), EQ.4: C ⁰ triangular shell, EQ.5: Belytschko-Tsay membrane, EQ.6: S/R Hughes Liu, EQ.7: S/R co-rotational Hughes Liu, EQ.8: Belytschko-Leviathan shell, EQ.9: fully integrated Belytschko-Tsay membrane, EQ.10: Belytschko-Wong-Chiang, EQ.11: Fast (co-rotational) Hughes-Liu. EQ.12: Plane stress (x-y plane), EQ.13: Plane strain (x-y plane), EQ.14: Axisymmetric solid (y-axis of symmetry) - area weighted, EQ.15: Axisymmetric solid (y-axis of symmetry) - volume weighted EQ.16: Fully integrated shell element (very fast) EQ.17: Discrete Kirchhoff triangular shell (DKT) EQ.18: Discrete Kirchhoff linear shell either quadrilateral or triangular EQ.20: C ⁰ linear shell element with drilling stiffness. For the 2D axisymmetric solid elements, high explosive applications work best with the area weighted approach and structural applications work best with the volume weighted approach. The volume weighted approach can lead to problems along the axis of symmetry under very large deformations. Often the symmetry condition is not obeyed, and the elements will kink along the axis. The volume weighted approach must be used if 2D shell elements are used in the mesh. Type 14 and 15 elements cannot be mixed in the same calculation.
BWC	Warping stiffness for Belytschko-Tsay shells: EQ.1: Belytschko-Wong-Chiang warping stiffness added. EQ.2: Belytschko-Tsay (default).
MITER	Plane stress plasticity option (applies to materials 3, 18, 19, and 24): EQ.1: iterative plasticity with 3 secant iterations (default), EQ.2: full iterative plasticity, EQ.3: radial return noniterative plasticity. May lead to false results and has to be used with great care.

VARIABLE	DESCRIPTION
PROJ	Projection method for the warping stiffness in the Belytschko-Tsay shell (the BWC option above) and the Belytschko-Wong-Chiang elements (see remarks below). This parameter applies to explicit calculations since the full projection method is always used if the solution is implicit and this input parameter is ignored. EQ.0: drill projection, EQ.1: full projection.
ROTASCL	Define a scale factor for the rotary shell mass. This option is not for general use. The rotary inertia for shells is automatically scaled to permit a larger time step size. A scale factor other than the default, i.e., unity, is not recommended.
INTGRD	Default shell through thickness numerical integration rule: EQ.0: Gauss integration. If 1-10 integration points are specified, the default rule is Gauss integration. EQ.1: Lobatto integration. If 3-10 integration points are specified, the default rule is Lobatto. For 2 point integration, the Lobatto rule is very inaccurate, so Gauss integration is used instead. Lobatto integration has an advantage in that the inner and outer integration points are on the shell surfaces.
LAMSHT	For composite shells with material types *MAT_COMPOSITE_DAMAGE and *MAT_ENHANCED_COMPOSITE_DAMAGE. If this flag is set laminated shell theory is used. Lamination theory is applied to correct for the assumption of a uniform constant shear strain through the thickness of the shell. Unless this correction is applied, the stiffness of the shell can be grossly incorrect if there are drastic differences in the elastic constants from ply to ply, especially for sandwich type shells. Generally, without this correction the results are too stiff. For the discrete Kirchhoff shell elements, which do not consider transverse shear, this option is ignored. EQ.0: do not update shear corrections, EQ.1: activate laminated shell theory.
CSTYP6	Coordinate system for the type 6 shell element. The default system computes a unique local system at each in plane point. The uniform local system computes just one system used throughout the shell element. This involves fewer calculations and is therefore more efficient. The change of systems has a slight effect on results; therefore, the older, less efficient method is the default. EQ.1: variable local coordinate system (default), EQ.2: uniform local system.
TSHELL	Thermal shell option. Four node shells are treated internally as twelve node brick elements to allow heat conduction through the thickness of the shell.

*CONTROL

<u>VARIABLE</u>	<u>DESCRIPTION</u>
NFAIL1	<p>Flag to check for highly distorted under-integrated shell elements, print a message, and delete the element or terminate. Generally, this flag is not needed for one point elements that do not use the warping stiffness. A distorted element is one where a negative jacobian exist within the domain of the shell, not just at integratiion points. The checks are made away from the CPU requirements for one point elements. If nonzero, NFAIL1 can be changed in a restart.</p> <p>EQ.1: print message and delete element. EQ.2: print message, write D3DUMP file, and terminate GT.2: print message and delete element. When NFAIL1 elements are deleted then write D3DUMP file and terminate. These NFAIL1 failed elements also include all shell elements that failed for other reasons than distortion. Before the D3DUMP file is writen, NFAIL1 is doubled, so the run can immediately be continued if desired.</p>
NFAIL4	<p>Flag to check for highly distorted fully-integrated shell elements, print a message and delete the element or terminate. Generally, this flag is recommended. A distorted element is one where a negative jacobian exist within the domain of the shell, not just at integratiion points. The checks are made away from the integration points to enable the bad elements to be deleted before an instability leading to an error termination occurs. If nonzero, NFAIL1 can be changed in a restart.</p> <p>EQ.1: print message and delete element. EQ.2: print message, write D3DUMP file, and terminate GT.2: print message and delete element. When NFAIL4 elements are deleted then write D3DUMP file amd terminate. These NFAIL4 failed elements also include all shell elements that failed for other reasons than distortion. Before the D3DUMP file is writen, NFAIL4 is doubled, so the run can immediately be continued if desired.</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, but on the other hand the element becomes too stiff due to loss of the rigid body rotation invariance under the same drill projection".

They later went on to add in the conclusions:

"The projection of only the drill rotations is very efficient and hardly increases the computation time, so it is recommended for most cases. However, it should be noted that the drill projection can result in a loss of invariance to rigid body motion when the elements are highly warped. For moderately warped configurations the drill projection appears quite accurate".

In crashworthiness and impact analysis, elements that have little or no warpage in the reference configuration can become highly warped in the deformed configuration and may affect rigid body rotations if the drill projection is used, i.e., **DO NOT USE THE DRILL PROJECTION**. Of course it is difficult to define what is meant by "moderately warped". The full projection circumvents these problems but at a significant cost. The cost increase of the drill projection versus no projection as reported by Belytschko and Leviathan is 12 percent and by timings in LS-DYNA, 7 percent, but for the full projection they report a 110 percent increase and in LS-DYNA an increase closer to 50 percent is observed.

In Version 940.xx of LS-DYNA the drill projection was used exclusively, but in one problem the lack of invariance was observed and reported; consequently, the drill projection was replaced in the Belytschko-Leviathan shell with the full projection and the full projection is now optional for the warping stiffness in the Belytschko-Tsay and Belytschko-Wong-Chiang elements. Until this problem occurred, the drill projection seemed okay. In version 950.xx and later versions of LS-DYNA the Belytschko-Leviathan shell is somewhat slower than previously. In general in light of these problems, the drill projection cannot be recommended. For implicit problems, the full projection method is used in the development of the stiffness matrix.

*CONTROL

*CONTROL_SOLID

Purpose: Provide controls for solid element response.

Card Format

	1	2	3	4	5	6	7	8
Variable	ESORT	FMATRIX	NIPTETS					
Type	I	I	I					
Default	0	0	4					

Optional Card Format (10I8)

Card 1	1	2	3	4	5	6	7	8	9	10
Variable	PM1	PM2	PM3	PM4	PM5	PM6	PM7	PM8	PM9	PM10
Type	I	I	I	I	I	I	I	I	I	I
Default	none									

<u>VARIABLE</u>	<u>DESCRIPTION</u>
ESORT	Automatic sorting of tetrahedron and pentahedron elements to treat degenerate tetrahedron and pentahedron elements as tetrahedron (formulation 10) and pentahedron (formulation 13) solids, respective. See *SECTION_SOLID. EQ.0: no sorting required (default). EQ.1: full sorting,
FMATRIX	Default method used in the calculation of the deformation gradient matrix. EQ.1: Update incrementally in time. This is the default for explicit. EQ.2: Directly compute F. This is the default for implicit and implicit/explicit switching.
NIPTETS	Number of integration points used in the quadratic tetrahedron elements. Either 4 or 5 can be specified. This option applies to the type 4 and type 16 tetrahedron elements.

<u>VARIABLE</u>	<u>DESCRIPTION</u>
PM1-PM10	Components of a permutation vector for nodes that define the 10-node tetrahedron. The nodal numbering of 10-node tetrahedron elements is somewhat arbitrary. The permutation vector allows other numbering schemes to be used. Unless defined, this permutation vector is not used. PM1-PM10 are unique number between 1 to 10 inclusive that reorders the input node ID's for a 10-node tetrahedron into the order used by LS-DYNA.

*CONTROL

*CONTROL_SOLUTION

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

Card Format

1 2 3 4 5 6 7 8

Variable	SOLN							
Type	I							
Default	0							

VARIABLE

DESCRIPTION

SOLN

Analysis solution procedure:

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

***CONTROL_SPH**

Purpose: Provide controls for computing SPH particles

Card Format

	1	2	3	4	5	6	7	8
Variable	NCBS	BOXID	DT	IDIM	MEMORY	FORM	START	MAXV
Type	I	I	F	I	I	I	F	F
Default	1	0	1.e20	none	150	0	0.0	1.e15

Card Format

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

VARIABLE**DESCRIPTION**

NCBS	Number of cycles between particle sorting
BOXID	SPH approximations are computed inside a specified BOX. When a particle has gone outside the BOX, it is deactivated. This will save computational time by eliminating particles that no longer interact with the structure.
DT	Death time. Determines when the SPH calculations are stopped.
IDIM	Space dimension for SPH particles: 3 for 3D problems 2 for 2D plane strain problems -2 for 2D axisymmetric problems When a value is not specified LS-DYNA determines the space dimension automatically by checking the use of 3D, 2D or 2D axisymmetric elements.

*CONTROL

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MEMORY	Defines the initial number of neighbors per particle. This variable is just for memory allocation of arrays during the initialization phase. During the calculation, some particles can request more neighbors and LS-DYNA will automatically adapt the size of that variable. Default value should apply for most applications.
FORM	Particle approximation theory: EQ. 0: default formulation, EQ. 1: remormalization approximation
START	Start time for particle approximation. Particle approximations will be computed when time of the analysis has reached the value defined in START.
MAXV	Maximum value for velocity for the SPH particles. Particles with a velocity greater than MAXV are deactivated.
CONT	Defines the computation of the particle approximation between two different SPH parts: EQ. 0: Particle approximation is defined (default) EQ. 1: Particle approximation is not computed. Two different SPH materials will not interact with each others and penetration is allowed.
DERIV	Time integration type for the smoothing length: EQ. 0: $\frac{d}{dt}(h(t)) = \frac{1}{d} h(t) \text{div}(v)$ (default), EQ. 1: $\frac{d}{dt}(h(t)) = \frac{1}{d} h(t) (\text{div}(v))^{1/3}$

*CONTROL_STRUCTURED_{*OPTION*}

Options include:

<BLANK>

TERM

Purpose: Write out a LS-DYNA structured input deck for Version 970. The name of this structured file is "dyna.str". This input deck will not support all capabilities that are available in Version 970. As a result some data such as load curve numbers will be output in an internal numbering system. If the TERM option is activated termination will occur after the structured input file is written. This option is useful in debugging especially if problems occur in reading the input file.

***CONTROL**

***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-DYNA terminates with a restart dump.
ENDENG	Percent change in energy ratio for termination of calculation. If undefined, this option is inactive.
ENDMAS	Percent change in the total mass for termination of calculation. This option is relevant if and only if mass scaling is used to limit the minimum time step size, see *CONTROL_TIMESTEP variable name "DT2MS".

Remarks:

1. Termination by displacement may be defined in the *TERMINATION section.
2. If the erosion flag on *CONTROL_TIMESTEP is set (ERODE=1), then the shell elements and solid elements with time steps falling below TSMIN will be eroded.

*CONTROL

*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					

<u>VARIABLE</u>	<u>DESCRIPTION</u>
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 $0.3 \leq DCP \leq 1.0$ default 1.0 transient problems $0.0 < DCP \leq 1.0$ 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	SBC
Type	I	I	I	F	I	F	F	F
Default	0	0	3	1.0e-04	8	1.	1.	0.

<u>VARIABLE</u>	<u>DESCRIPTION</u>
ATYPE	Thermal analysis type: EQ.0: Steady state analysis, EQ.1: transient analysis.
PTYPE	Thermal problem type: (see *CONTROL_THERMAL_NONLINEAR if non-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.)
SBC	Stefan Boltzmann constant. Value is used with enclosure radiation surfaces, see *BOUNDARY_RADIATION_....

*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	DT2MSLC						
Type	F	I						
Default	not used	not used						

<u>VARIABLE</u>	<u>DESCRIPTION</u>
DTINIT	Initial time step size: EQ.0.0: LS-DYNA determines initial step size.
TSSFAC	Scale factor for computed time step (old name SCFT). See Remark 1 below. (Default = .90; if high explosives are used, the default is lowered to .67).
ISDO	Basis of time size calculation for 4-node shell elements. 3-node shells use the shortest altitude for options 0,1 and the shortest side for option 2. This option has no relevance to solid elements, which use a length based on the element volume divided by the largest surface area. EQ.0: characteristic length=area/(minimum of the longest side or the longest diagonal). EQ.1: characteristic length=area/(longest diagonal).

<u>VARIABLE</u>	<u>DESCRIPTION</u>
	<p>EQ.2: based on bar wave speed and MAX [shortest side, area/(minimum of the longest side or the longest diagonal)]. THIS LAST OPTION CAN GIVE A MUCH LARGER TIME STEP SIZE THAT CAN LEAD TO INSTABILITIES IN SOME APPLICATIONS, ESPECIALLY WHEN TRIANGULAR ELEMENTS ARE USED.</p> <p>EQ.3: timestep size is based on the maximum eigenvalue. This option is okay for structural applications where the material sound speed changes slowly. The calculational cost to determine the maximum eigenvalue is significant, but the increase in the time step size often allows for significantly shorter run times without using mass scaling.</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. This option is applicable only to shell elements using material models: *MAT_PLASTIC_KINEMATIC, *MAT_POWER_LAW_PLASTICITY, *MAT_STRAIN_RATE_DEPENDENT_PLASTICITY, *MAT_PIECEWISE_LINEAR_PLASTICITY. This so-called stiffness scaling option is NOT recommended. The DT2MS option below applies to all materials and element classes and is preferred. If both TSLIMIT and DT2MS below are active and if TSLIMIT is input as a positive number, then TSLIMIT is set to 1.E-18, which makes it inactive. If TSLIMIT is negative and less than DT2MS , then TSLIMIT is applied prior to the mass being scaled. If DT2MS exceeds the magnitude of TSLIMIT, then TSLIMIT is set to 1.E-18.</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". WARNING: Superelements, *ELEMENT_DIRECT_MATRIX_INPUT, are not mass scaled; consequently, DT2MS does not affect their time step size. In this case an error termination will occur, and DT2MS will need to be reset to a smaller value.</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 t-shell elements when TSMIN (see *CONTROL_TERMINATION) is reached. If this flag is not set the calculation will terminate:</p> <p>EQ.0: no, EQ.1: yes.</p>

*CONTROL

VARIABLE	DESCRIPTION
MS1ST	Limit mass scaling to the first step and fix the mass vector according to the time steps once. The time step will not be fixed but may drop during the calculation from the specified minimum: EQ.0: no, EQ.1: yes.
DT2MSF	Reduction (or scale) factor for initial time step size to determine the minimum time step size permitted. Mass scaling is done if it is necessary to meet the Courant time step size criterion. If this option is used DT2MS=-DT2MSF multiplied by the initial time step size, Δt , before Δt is scaled by TSSFAC. This option is active if and only if DT2MS=0 above.
DT2MSLC	Load curve specifying DT2MS as a function of time during the explicit solutions phase. The load curve can only be used for increasing the magnitude of DT2MS. Consequently, the magnitude of DT2MS is taken as the maximum of the current value and the value from the load curve.

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 approximately 5mm per microsecond; therefore, if a steel structure is modeled with element sizes of 5mm, the computed time step size would be 1 microsecond. Elements made from materials with lower sound speeds, such as foams, will give larger time step sizes. Avoid excessively small elements and be aware of the effect of rotational inertia on the time step size in the Belytschko beam element. Sound speeds differ for each material, for example, consider:

AIR	331 m/s
WATER	1478
STEEL	5240
TITANIUM	5220
PLEXIGLAS	2598

- Model stiff components with rigid bodies, not by scaling Young's modulus which can substantially reduce the time step size.
- The altitude of the triangular element should be used to compute the time step size. Using the shortest side is okay only if the calculation is closely examined for possible instabilities. This is controlled by parameter ISDO.

***DAMPING**

The Keyword options in this section in alphabetical order are:

***DAMPING_FREQUENCY_RANGE**

***DAMPING_GLOBAL**

***DAMPING_PART_MASS**

***DAMPING_PART_STIFFNESS**

***DAMPING_RELATIVE**

***DAMPING_FREQUENCY_RANGE**

Purpose: This feature provides approximately constant damping (i.e. frequency-independent) over a range of frequencies.

Card Format

	1	2	3	4	5	6	7	8
Variable	CDAMP	FLOW	FHIGH	PSID				
Type	F	F	F	I				
Default	0.0	0.0	0.0	0				

VARIABLE

DESCRIPTION

CDAMP	Damping in fraction of critical.
FLOW	Lowest frequency in range of interest (cycles per unit time, e.g. Hz if time unit is seconds)
FHIGH	Highest frequency in range of interest (cycles per unit time, e.g. Hz if time unit is seconds)
PSID	Part set ID. The requested damping is applied only to the parts in the set. If PSID = 0, the damping is applied to all parts except those referred to by other *DAMPING_FREQUENCY_RANGE cards.

*DAMPING

This feature provides approximately constant damping (i.e. frequency-independent) over a range of frequencies $F_{low} < F < F_{high}$. It is intended for small damping ratios (e.g. < 0.05) and frequency ranges such that F_{high}/F_{low} is in the range 10-300. The drawback to this method of damping is that it reduces the dynamic stiffness of the model, especially at low frequencies. This effect is predictable: the natural frequencies of modes close to F_{low} are reduced by 3% for a damping ratio of 0.01 and F_{high}/F_{low} in the range 10-30. Near F_{high} the error is between zero and one third of the error at F_{low} . Estimated frequency errors are shown in the table below.

Frequency	F_{high}/F_{low}			
		3 to 30	30 to 300	300 to 3000
Error at F_{low}				
Damping Ratio	0.01	3%	4.5%	6%
	0.02	6%	9%	12%
	0.04	12%	18%	24%

It is recommended that the elastic stiffnesses in the model be increased slightly to account for this, e.g. for 0.01 damping across a frequency range of 30 to 600Hz, the average error across the frequency range is about 2%. Increase the stiffness by $(1.02)^2$, i.e. by 4%.

***DAMPING_GLOBAL**

Purpose: Define mass weighted nodal damping that applies globally to the nodes of deformable bodies and to the mass center of the rigid bodies.

Card 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 the system damping constant: EQ.0: a constant damping factor as defined by VALDMP is used, EQ.n: system damping is given by load curve n. The damping force applied to each node is $f=-d(t)mv$, where $d(t)$ is defined by load curve n.
VALDMP	System damping constant, d (this option is bypassed if the load curve number defined above is non zero).
STX	Scale factor on global x translational damping forces.
STY	Scale factor on global y translational damping forces.
STZ	Scale factor on global z translational damping forces.
SRX	Scale factor on global x rotational damping moments.
SRY	Scale factor on global y rotational damping moments.
SRZ	Scale factor on global z rotational damping moments.

*DAMPING

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 some value approaching the critical damping factor for the lowest frequency mode of interest.

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

The natural frequency (given in radians per unit time) is generally taken as the fundamental frequency of the structure. This frequency can be determined from an eigenvalue analysis or from an undamped transient analysis. Note that this damping applies to both translational and rotational degrees of freedom. Also note that mass proportional damping will damp rigid body motion as well as vibration.

Energy dissipated by through mass weighted damping is reported as system damping energy in the ASCII file GLSTAT. This energy is computed whenever system damping is active.

***DAMPING_PART_MASS**

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.

Energy dissipated by through mass weighted damping is reported as system damping energy in the ASCII file GLSTAT. This energy is computed whenever system damping is active.

***DAMPING_PART_STIFFNESS**

Purpose: Assign Rayleigh stiffness damping coefficient by part ID.

Card Format

	1	2	3	4	5	6	7	8
Variable	PID	COEF						
Type	I	F						
Default	none	0.0						

VARIABLE

DESCRIPTION

PID	Part ID, see *PART.
COEF	Rayleigh damping coefficient. Two methods are now available: LT.0.0: Rayleigh damping coefficient is set based on a given frequency and applied uniformly to each element in the part ID. This approach is used in versions of LS-DYNA prior to version 960. See notes below. EQ.0.0: Inactive. GT.0.0: Rayleigh damping coefficient for stiffness weighted damping. Values between 0.01 and 0.25 are recommended. Higher values are strongly discouraged, and values less than 0.01 may have little effect. The damping coefficient is uniquely defined for each element of the part ID.

Remarks:

The damping matrix in Rayleigh damping is defined as:

$$C = \alpha M + \beta K$$

where C, M, and K are the damping, mass, and stiffness matrices, respectively. The constants α and β are the mass and stiffness proportional damping constants. The mass proportional damping can be treated by system damping, see keywords: *DAMPING_GLOBAL and DAMPING_PART_MASS. Transforming C with the *i*th eigenvector ϕ_i gives:

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

where ω_i is the *i*th frequency (radians/unit time) and ξ_i is the corresponding modal damping parameter.

*DAMPING

Generally, the stiffness proportional damping is effective for high frequencies and is orthogonal to rigid body motion. Mass proportional damping is more effective for low frequencies and will damp rigid body motion. If a large value of the stiffness based damping coefficient is used, it may be necessary to lower the time step size significantly. This must be done manually by reducing the time step scale factor on the *CONTROL_TIMESTEP control card. Since a good value of β is not easily identified, the coefficient, COEF, is defined such that a value of .10 roughly corresponds to 10% damping in the high frequency domain.

In versions prior to 960, one damping coefficient is defined that applies to all elements of the entire part. With this older approach if 10% of critical damping is sought in the i th mode then set:

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

and input β as a negative number. Typically, β is some fraction of the time step size.

Energy dissipated by Rayleigh damping is computed if and only if the flag, RYLEN, on the control card, *CONTROL_ENERGY is set to 2. This energy is accumulated as element internal energy and is included in the energy balance. In the GLSTAT file this energy will be lumped in with the internal energy.

***DAMPING_RELATIVE**

Purpose: Apply damping relative to the motion of a rigid body.

Card Format

	1	2	3	4	5	6	7	8
Variable	CDAMP	FREQ	PIDRB	PSID				
Type	F	F	F	I				
Default	0	0	0	0				

VARIABLE**DESCRIPTION**

CDAMP	Fraction of critical damping.
FREQ	Frequency at which CDAMP is to apply (cycles per unit time, e.g. Hz if time unit is seconds).
PIDRB	Part ID of rigid body, see *PART. Motion relative to this rigid body will be damped.
PSID	Part set ID. The requested damping is applied only to the parts in the set.

Remarks:

1. This feature provides damping of vibrations for objects that are moving through space. The vibrations are damped, but not the rigid body motion. This is achieved by calculating the velocity of each node relative to that of a rigid body, and applying a damping force proportional to that velocity. The forces are reacted onto the rigid body such that overall momentum is conserved. It is intended that the rigid body is embedded within the moving object.
2. Vibrations at frequencies below FREQ are damped by more than CDAMP, while those at frequencies above FREQ are damped by less than CDAMP. It is recommended that FREQ be set to the frequency of the lowest mode of vibration.

*DAMPING

***DATABASE**

The database definitions are optional, but are necessary to obtain output files containing results information. In this section the database keywords are defined in alphabetical order:

***DATABASE_OPTION**

***DATABASE_ADAMS**

***DATABASE_BINARY_OPTION**

***DATABASE_CROSS_SECTION_OPTION1_{OPTION2}**

***DATABASE_EXTENT_OPTION**

***DATABASE_FORMAT**

***DATABASE_FSI**

***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):

ABSTAT	Airbag statistics.
AVSFLT	AVS database. See *DATABASE_EXTENT_OPTION.
BNDOUT	Boundary condition forces and energy
DEFGEO	Deformed geometry file. (Note that to output this file in Chrysler format insert the following line in your <i>.cshrc</i> file: “setenv LSTC_DEFGEO chrysler”) The NASBDF file (NASTRAN Bulk Data) is created whenever the DEFGEO file is requested.
DEFORC	Discrete elements.
ELOUT	Element data. See *DATABASE_HISTORY_OPTION.
GCEOUT	Geometric contact entities.
GLSTAT	Global data. Always obtained if SSSTAT file is activated.
H3OUT	HybridIII rigid body dummies.
JNTFORC	Joint force file
MATSUM	Material energies. See Remarks 1 and 2 below.
MOVIE	MOVIE. See *DATABASE_EXTENT_OPTION.
MPGS	MPGS. See *DATABASE_EXTENT_OPTION.
NCFORC	Nodal interface forces. See *CONTACT - Card 1 (SPR and MPR)
NODFOR	Nodal force groups. See *DATABASE_NODAL_FORCE_GROUP.
NODOUT	Nodal point data. See *DATABASE_HISTORY_OPTION.
RBDOUT	Rigid body data. See Remark 2 below.
RCFORC	Resultant interface forces.
RWFORC	Wall forces.
SBTOUT	Seat belt output file
SECFORC	Cross section forces. See *DATABASE_CROSS_SECTION_OPTION.
SLEOUT	Sliding interface energy. See *CONTROL_ENERGY
SPCFORC	SPC reaction forces.
SPHOUT	SPH data. See *DATABASE_HISTORY_OPTION.
SSSTAT	Subsystem data. See *DATABASE_EXTENT_SSSTAT.
SWFORC	Nodal constraint reaction forces (spotwelds and rivets).
TPRINT	Thermal output from a coupled structural/thermal or thermal only analysis.
TRHIST	Tracer particle history information. See *DATABASE_TRACER.

Card Format

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

VARIABLE

DESCRIPTION

DT	Time interval between outputs. If DT is zero, no output is printed.
BINARY	Flag for binary file EQ.1: ASCII file is written. This is the default on serial and shared memory computers. EQ.2: Data written to a binary database, which contains data that would otherwise be output to the ASCII file. The ASCII file in this case is not created. This is the default on distributed memory computers. EQ.3: ASCII file is written and the data is also written to the binary database.

The file names and corresponding unit numbers are:

	<u>I/O UNIT #</u>	<u>FILE NAME</u>
Airbag statistics	i/o unit #43	ABSTAT
ASCII database	i/o unit #44	AVSFLT
Boundary conditions	i/o unit #46	BNDOUT (nodal forces and energies)
Smug animator database	i/o unit#40	DEFGEO
Discrete elements	i/o unit#36	DEFORC
Element data	i/o unit#34	ELOUT
Contact entities	i/o unit #48	GCEOUT
Global data	i/o unit#35	GLSTAT
Joint forces	i/o unit #53	JNTFORC
Material energies	i/o unit#37	MATSUM
MOVIE file family	i/o unit #50	MOVIE _{nnn} .xxx where.nnn=001-999
MPGS file family	i/o unit #50	MPGS _{nnn} .xxx where nnn=001-999
Nastran/BDF file	i/o unit#49	NASBDF (see comment below)
Nodal interface forces	i/o unit#38	NCFORC

*DATABASE

	<u>I/O UNIT #</u>	<u>FILE NAME</u>
Nodal force group	i/o unit #45	NODFOR
Nodal point data	i/o unit#33	NODOUT
Rigid body data	i/o unit #47	RBDOUT
Resultant interface forces	i/o unit#39	RCFORC
Rigidwall forces	i/o unit#32	RWFORC
Seat belts	i/o unit #52	SBTOUT
Cross-section forces	i/o unit#31	SECFORC
Interface energies	i/o unit #51	SLEOUT
SPC reaction forces	i/o unit#41	SPCFORC
SPH element data	i/o unit#68	SPHOUT
Subsystems statistics	i/o unit#58	SSSTAT
Nodal constraint resultants	i/o unit #42	SWFORC (spotwelds/rivets)
Thermal output	i/o unit #73	TPRINT
Tracer particles	i/o unit #70	TRHIST

Output Components for ASCII Files

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

ELOUT

Beam	Stress Shell	Brick	Strain Shell
axial force resultant	xx,yy,zz stress	xx,yy,zz stress	xx,yy,zz strain
s shear resultant	xy,yz,zx stress	xy,yz,zx stress	xy,yz,zx strain
t shear resultant	plastic strain	effective stress	lower surface strain
s moment resultant		yield function	upper surface strain
t moment resultant			
torsional resultant			

GCEOUT

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

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

NCFORC	NODOUT	NODFOR
x force	x,y,z displacement	x,y,z force
y force	x,y,z velocity	
z force	x,y,z acceleration	
	x,y,z rotation	
	x,y,z rotational velocity	
	x,y,z rotational acceleration	

RBDOUT	RCFORC	RWFORC
x,y,z displacement	x,y,z force	normal
x,y,z velocity		x,y,z force
x,y,z acceleration		

SECFORC	SLEOUT	SPCFORC	SWFORC
x,y,z force	slave energy	x,y,z force	axial force
x,y,z moment	master energy	x,y,z moment	shear force
x,y,z center			
area			
resultant force			

*DATABASE

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-PREPOST. Arbitrary cross plotting of results between ASCII files is easily handled.
5. Resultant contact forces reported in RCFORC are averaged over the preceding output interval.
6. “Spring and damper energy” reported in GLSTAT is a subset of “Internal energy”. The “Spring and damper energy” includes internal energy of discrete elements, seatbelt elements, and that associated with joint stiffness (see *CONSTRAINED_JOINT_STIFFNESS_...).

***DATABASE_ADAMS**

Purpose: Request output of an MDI Modal Neutral File for later use in the ADAMS software.

Card Format

	1	2	3	4	5	6	7	8
Variable	IFLAG	M_UNITS	L_UNITS	T_UNITS				
Type	I	F	F	F				
Default	0	none	none	none				

VARIABLE**DESCRIPTION**

IFLAG	Flag controlling write of modal neutral file after eigenvalue analysis EQ.0: do not write (default), EQ.1: write to file "d3mnf"
M_UNITS	Mass units of measure used in this model. EQ.-1: kilogram EQ.-2: gram EQ.-3: megagram (metric ton) EQ.-4: lbf*sec**2/in (psi-compatible) EQ.-5: slug EQ.-6: pound-mass
L_UNITS	Length units of measure used in this model. EQ.-1: meter EQ.-2: centimeter EQ.-3: millimeter EQ.-4: inch EQ.-5: foot
T_UNITS	Time units of measure used in this model. EQ.-1: second EQ.-2: millisecond EQ.-3: minute EQ.-4: hour

Remarks:

1. This option is not available for every platform. Check LS-DYNA Banner upon execution of the program to see if this feature is enabled.
2. Models must be created using a combination of the above units.

*DATABASE

*DATABASE_BINARY_OPTION

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

D3DRLF	Dynamic relaxation database.
D3DUMP	Binary output restart files. Define output frequency in cycles.
D3MEAN	Averaging interval and statistics level for mean value database.
D3PART	Dt for partial output states See also *DATABASE_EXTENT_BINARY.
D3PLOT	Dt for complete output states. See also *DATABASE_EXTENT_BINARY.
D3THDT	Dt for time history data of element subsets. See *DATABASE_HISTORY.
RUNRSF	Binary output restart file. Define output frequency in cycles.
INTFOR	Dt for output of contact interface data (file name must be given on the execution line using "S="). Also see *CONTACT variables mpr and spr.
XTFILE	Flag to specify output of extra time history data to XTFILE at same time as D3THDT file. The following card is left blank for this option.
D3CRACKD	Dt for output of crack data file for the Winfrith concrete model (file name must be given on the execution line using "q="). This file can be used with the D3PLOT file to show crack formation of the deformed concrete materials.

The D3DUMP and the RUNRSF options create complete databases which are necessary for restarts, see *RESTART. When RUNRSF is specified, the same file is overwritten after each interval, an option allows a series of files to be overwritten in a cyclic order. When D3DUMP is specified, a new restart file is created after each interval. When D3DUMP is specified, a new restart file is created after each interval, thus a "family" of files is created numbered sequentially D3DUMP01, D3DUMP02, etc. The default file names are RUNRSF and D3DUMP unless other names are specified on the execution line, see the INTRODUCTION, EXECUTION SYNTAX. Since all data held in memory is written into the restart files, these files can be quite large and care should be taken with the D3DUMP files not to create too many. If *DATABASE_BINARY_D3PLOT is not specified in the keyword deck then a complete output state will be written ever timestep.

The D3PLOT, D3PART, D3DRLF, and the INTFOR files contain plotting information to plot data over the three dimensional geometry of the model. These databases can be plotted with LS-PREPOST. The D3THDT file contains time history data for element subsets as well as global information, see *DATABASE_HISTORY. This data can be plotted with LS-PREPOST. The default names for the D3PLOT, D3PART, D3DRLF, and the D3THDT files are D3PLOT, D3PART, D3DRLF, and D3THDT. For INTFOR a unique name must be specified on the execution line with S=iff, (iff=file name), see the INTRODUCTION, EXECUTION SYNTAX. The file structure is such that each file contains the full geometry at the beginning, followed by the analysis generated output data at the specified time intervals. For the contents of the D3PLOT, D3PART and D3THDT files see also the *DATABASE_EXTENT_BINARY definition. It is possible to severely restrict the information that is dumped and consequently reduce the size of the databases. The contents of the D3THDT file are also specified with the *DATABASE_HISTORY definition. It should also be noted in particular that the databases can be considerably reduced for models with rigid bodies containing many elements.

Card Format

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

Optional Card that only applies to the D3PLOT database

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

VARIABLE

DESCRIPTION

DT	Time interval between outputs.
CYCL	Output interval in time steps (a time step is a cycle). For the D3DRFL file a positive number 'n' will cause plot dumps to be written at every n'th convergence check interval specified on the *CONTROL_DYNAMIC_RELAXATION card.
NR	Number of Running restart files written in a cyclical fashion. The default number is one, i.e. the same file is overwritten each time.
LCDT	Optional load curve ID specifying time interval between dumps. This option is only available for the D3PLOT, D3PART, D3THDT and INTFOR files.

*DATABASE

<u>VARIABLE</u>	<u>DESCRIPTION</u>
BEAM	<p>Option flag for *DATABASE_BINARY_D3PLOT or D3PART.</p> <p>EQ.0: Discrete spring and damper elements are added to the D3PLOT or D3PART database where they are display as beam elements. The element global X, global Y, global Z and resultant forces are written to the database,</p> <p>EQ.1 No discrete spring and damper elements are added to the D3PLOT or D3PART database. This option is useful when translating old LS-DYNA input decks to KEYWORD input. In older input decks there is no requirement that beam and spring elements have unique ID's, and beam elements may be created for the spring and dampers with identical ID's to existing beam elements causing a fatal error,</p> <p>EQ.2. Discrete spring and damper elements are added to the D3PLOT or D3PART database where they are displayed as beam elements (similar to option 0). In this option the element resultant force is written to its first database position allowing beam axial forces and spring resultant forces to be plotted at the same time. This can be useful during some post-processing applications.</p>
NPLTC	<p>DT=ENDTIME/NPLTC applies to D3PLOT and D3PART only. This overrides the DT specified in the first field.</p>
PSETID	<p>SET_PART ID for D3PART only.</p>
ISTATS	<p>Set the level of statistics to collect. This applies to D3MEAN only, and is also restricted to the incompressible CFD solver variables.</p> <p>EQ.0: don't collect any statistics (default),</p> <p>EQ.1: generate mean quantities,</p> <p>EQ.2: generate second moments in addition to the mean quantities,</p> <p>EQ.3: generate higher-order moments in addition to all other moments.</p>
TSTART	<p>Set the simulation time at which collection of the time-averaged statistics will begin (D3MEAN only). TSTART=0.0 is the default.</p>
I AVG	<p>Set the interval to write out the time-averaged statistics (D3MEAN only). The time-averaged statistics are re-initialized and collection of new statistics starts after the time-averaged data is written to the database.</p> <p>EQ.0: I AVG=100 (default).</p>
IOOPT	<p>This option applies to the D3PLOT file only. Flag to govern behavior of plot frequency load curve:</p> <p>EQ.1: At the time each plot is generated, the load curve value is added to the current time to determine the next plot time.(this is the default behavior)</p> <p>EQ 2: At the time each plot is generated, the next plot time T is computed so that T = the current time plus the load curve value at time T.</p> <p>EQ 3: A plot is generated for each ordinate point in the load curve definition. The actual value of the load curve is ignored.</p>

Remarks:

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

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

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

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

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

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

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

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

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

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

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

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

Format (1 of 2) for the PLANE option

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

Format (2 of 2) for the PLANE option

	1	2	3	4	5	6	7	8
Variable	XHEV	YHEV	ZHEV	LENL	LENM	ID	ITYPE	
Type	F	F	F	F	F	I	I	
Default	0.	0.	0.	infinity	infinity	global	0	

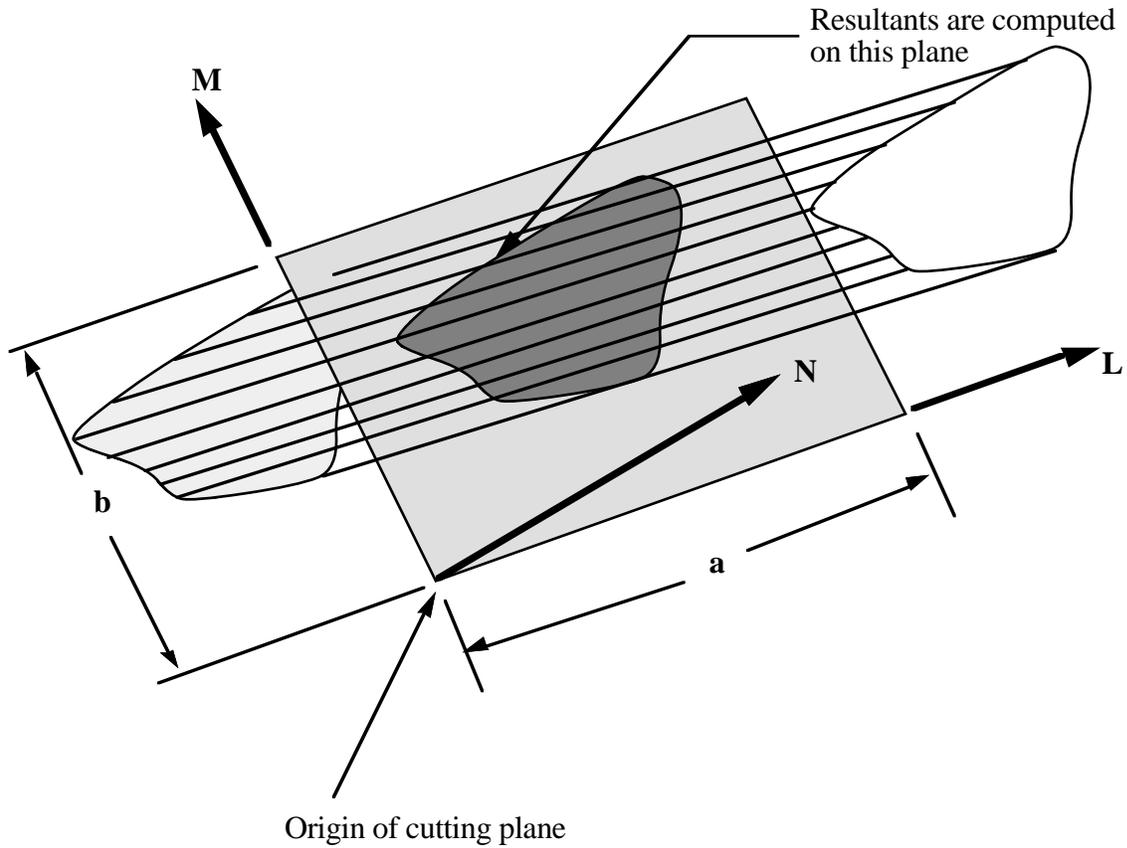


Figure 9.1. Definition of cutting plane for automatic definition of interface for cross-sectional forces. The automatic definition does not check for springs and dampers in the section. For best results the cutting plane should cleanly pass through the middle of the elements, distributing them equally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

The set option requires that the equivalent of the automatically generated input via the cutting plane be identified manually and defined in sets. All nodes in the cross-section and their related elements that contribute to the cross-sectional force resultants should be defined.

Format (1 of 1) for the SET option

	1	2	3	4	5	6	7	8
Variable	NSID	HSID	BSID	SSID	TSID	DSID	ID	ITYPE
Type	I	I	I	I	I	I	I	I
Default	required	0	0	0	0	0	global	0

<u>VARIABLE</u>	<u>DESCRIPTION</u>
CSID	Optional ID for cross section. If not specified cross section ID is taken to be the cross section order in the input deck.
PSID	Part set ID. If zero all parts are included.
XCT	x-coordinate of tail of any outward drawn normal vector, N , originating on wall (tail) and terminating in space (head), see Figure 9.1.
YCT	y-coordinate of tail of normal vector, N .
ZCT	z-coordinate of tail of normal vector, N .
XCH	x-coordinate of head of normal vector, N .
YCH	y-coordinate of head of normal vector, N .
ZCH	z-coordinate of head of normal vector, N .
XHEV	x-coordinate of head of edge vector, L .
YHEV	y-coordinate of head of edge vector, L .
ZHEV	z-coordinate of head of edge vector, L .
LENL	Length of edge a, in L direction.
LENM	Length of edge b, in M direction.
NSID	Nodal set ID, see <i>*SET_NODE_OPTION</i> .
HSID	Solid element set ID, see <i>*SET_SOLID</i> .
BSID	Beam element set ID, see <i>*SET_BEAM</i> .

*DATABASE

<u>VARIABLE</u>	<u>DESCRIPTION</u>
SSID	Shell element set ID, see *SET_SHELL_OPTION.
TSID	Thick shell element set ID, see *SET_TSHELL.
DSID	Discrete element set ID, see *SET_DISCRETE.
ID	Rigid body (see *MAT_RIGID, type 20) or accelerometer ID (see *ELEMENT_SEATBELT_ACCELEROMETER). The force resultants are output in the <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
BINARY
MOVIE
MPGS
SSSTAT

Purpose: Specify output database to be written. Binary applies to the data written to the D3PLOT, D3PART, and D3THDT files. See *DATABASE_BINARY_OPTION.

For the AVS, MPGS, and MOVIE options the following cards apply:

Define as many cards as necessary. The created MPGS and MOVIE databases consist of a geometry file and one file for each output database.

Card Format

	1	2	3	4	5	6	7	8
Variable	VTYPE	COMP						
Type	I	I						

VARIABLE

DESCRIPTION

VTYPE

Variable type:
EQ.0: node,
EQ.1: brick,
EQ.2: beam,
EQ.3: shell,
EQ.4: thick shell.

COMP

Component ID. For the corresponding VTYPE, integer components from the following tables can be chosen:
VTYPE.EQ.0: Table 9.1,
VTYPE.EQ.1: Table 9.2,
VTYPE.EQ.2: not supported,
VTYPE.EQ.3: Table 9.3,
VTYPE.EQ.4: not supported.

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

Table 9.2. Brick Element Quantities

Component ID	Quantity
1	x-stress
2	y-stress
3	z-stress
4	xy-stress
5	yz-stress
6	zx-stress
7	effective plastic strain

Table 9.3. Shell and Thick Shell Element Quantities

Component ID	Quantity
1	midsurface x-stress
2	midsurface y-stress
3	midsurface z-stress
4	midsurface xy-stress
5	midsurface yz-stress
6	midsurface xz-stress
7	midsurface effective plastic strain
8	inner surface x-stress
9	inner surface y-stress
10	inner surface z-stress
11	inner surface xy-stress
12	inner surface yz-stress
13	inner surface zx-stress
14	inner surface effective plastic strain
15	outer surface x-stress
16	outer surface y-stress
17	outer surface z-stress
18	outer surface xy-stress
19	outer surface yz-stress
20	outer surface zx-stress

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

Component ID	Quantity
21	outer surface effective plastic strain
22	bending moment-mxx (4-node shell)
23	bending moment-myy (4-node shell)
24	bending moment-mxy (4-node shell)
25	shear resultant-qxx (4-node shell)
26	shear resultant-qyy (4-node shell)
27	normal resultant-nxx (4-node shell)
28	normal resultant-nyy (4-node shell)
29	normal resultant-nxy (4-node shell)
30	thickness (4-node shell)
31	element dependent variable
32	element dependent variable
33	inner surface x-strain
34	inner surface y-strain
35	inner surface z-strain
36	inner surface xy-strain
37	inner surface yz-strain
38	inner surface zx-strain
39	outer surface x-strain
40	outer surface y-strain
41	outer surface z-strain
42	outer surface xy-strain
43	outer surface yz-strain
44	outer surface zx-strain
45	internal energy
46	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 ID	Quantity
1	x-force resultant
2	y-force resultant
3	z-force resultant
4	x-moment resultant
5	y-moment resultant
6	z-moment resultant

*DATABASE

For the **BINARY** option the following cards apply (Card 3 is optional):

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					

Card 3 1 2 3 4 5 6 7 8

Variable	NINTSLD							
Type	I							
Default	1							
Remarks								

<u>VARIABLE</u>	<u>DESCRIPTION</u>
NEIPH	Number of additional integration point history variables written to the binary database for solid elements. The integration point data is written in the same order that it is stored in memory-each material model has its own history variables that are stored. For user defined materials it is important to store the history data that is needed for plotting before the data which is not of interest.
NEIPS	Number of additional integration point history variables written to the binary database for both shell and thick shell elements for each integration point, see NEIPH above.
MAXINT	Number of shell integration points written to the binary database, see also *INTEGRATION_SHELL. If the default value of 3 is used then results are output for the outermost (top) and innermost (bottom) integration points together with results for the neutral axis. If MAXINT is set to 3 and the element has 1 integration point then all three results will be the same. If a value other than 3 is used then results for the first MAXINT integration points in the element will be output. Note: If the element has an even number of integration points and MAXINT is not set to 3 then you will not get mid-surface results. See Remarks below.
STRFLG	Set to 1 to dump strain tensors for solid, shell and thick shell elements for plotting by LS-PREPOST and ASCII file ELOUT. For shell and thick shell elements two tensors are written, one at the innermost and one at the outermost integration point. For solid elements a single strain tensor is written.
SIGFLG	Flag for including stress tensor in the shell LS-DYNA database: EQ.1: include (default), EQ.2: exclude.
EPSFLG	Flag for including the effective plastic strains in the shell LS-DYNA database: EQ.1: include (default), EQ.2: exclude.
RLTFLG	Flag for including stress resultants in the shell LS-DYNA database: EQ.1: include (default), EQ.2: exclude.
ENGFLG	Flag for including shell internal energy density and thickness in the LS-DYNA database: EQ.1: include (default), EQ.2: exclude.
CMPFLG	Orthotropic and anisotropic material stress and strain output in local material coordinate system for solids, shells and thick shells. EQ.0: global, EQ.1: local.

*DATABASE

<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 1000 states: EQ.0: more than one state can be on each plotfile, EQ.1: one state only on each plotfile.
BEAMIP	Number of beam integration points for output. This option does not apply to beams that use a resultant formulation.
DCOMP	Data compression to eliminate rigid body data: EQ.1: off (default), no rigid body data compression, EQ.2: on, rigid body data compression active, EQ.3: off , no rigid body data compression, but nodal velocities and accelerations are eliminated from the database. EQ.4: on, rigid body dataa compression active and nodal velocities and accelerations are eliminated from the database.
SHGE	Output shell hourglass energy density: EQ.1: off (default), no hourglass energy written, EQ.2: on.
STSSZ	Output shell element time step, mass, or added mass: EQ.1: off (default), EQ.2: output time step size, EQ.3: output mass, added mass, or time step size. See remark 3 below.
N3THDT	Material energy write option for D3THDT database EQ.1: off, energy is NOT written to D3THDT database, EQ.2: on (default), energy is written to D3THDT database.
NINTSLD	Number of solid element integration points written to the LS-DYNA database. The default value is 1. For solids with multiple integration points NINTSLD may be set to 8. Currently, no other values for NINTSLD are allowed. For solids with multiple integration points, an average value is output if NINTSLD is set to 1.

Remarks:

1. If MAXINT is set to 3 then mid-surface, inner-surface and outer-surface stresses are output at the center of the element to the LS-DYNA database. For an even number of integration points, the points closest to the center are averaged to obtain the midsurface values. If multiple integration points are used in the shell plane, the stresses at the center of the element are found by computing the average of these points. For MAXINT equal to 3 LS-DYNA assumes that the data for the user defined integration rules are ordered from bottom to top even if this is not the case. If MAXINT is not equal to 3, then the stresses at the center of the element are output in the order that they are stored for the selected integration rule. If multiple points are used in plane the stresses are first averaged.

2. Beam stresses are output to the LS-DYNA database if and only if BEAMIP is greater than zero. In this latter case the data that is output is written in the same order that the integration points are defined. The data at each integration point consists of the following five values for elastic-plastic Hughes-Liu beams: the normal stress, σ_{rr} ; the transverse shear stresses, σ_{rs} and σ_{tr} ; the effective plastic strain, and the axial strain which is logarithmic. For beams that are not elastic-plastic, the first history variable, if any, is output instead of the plastic strain. For the beam elements of Belytschko and his co-workers, the transverse shear stress components are not used in the formulation. No data is output for the Belytschko-Schwer resultant beam.
3. If mass scaling is active, the output of the time step size reveals little information about the calculation. If global mass scaling is used for a constant time step, the total element mass is output; however, if the mass is increased so that a minimum time step size is maintained (DT2MS is negative), the added mass is output. Also, see the control card *CONTROL_TIMESTEP.

*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_FSI

Purpose: When present, this card activates the output of an ASCII file called “dbfsi” containing some coupling information (force, pressure, accumulated mass over some surfaces, etc.). This card is used only when the *CONSTRAINED_LAGRANGE_IN_SOLID card is used.

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	DT								
Type	F								

Define one surface per card

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

Variable	DBFSI_ID	SID	SIDTYPE						
Type	I	I	I						

VARIABLE

DESCRIPTION

DT

Output interval

DBFSI_ID

Surface ID (for reference purposes only) or a DATABASE_FSI entity ID. It consists of a geometric entity defined by the set ID below.

SID

Set ID defining the geometrical surface(s) through/upon which some data is to be tracked and output to an ASCII file called “dbfsi”. This set ID can be a (1) PID or (2) PSID or (3) SGSID.

SIDTYPE

Set type:
 EQ.0: Part set,
 EQ.1: Part,
 EQ.2: Segment set.

Remarks:

1. When a Lagrangian mesh overlaps with an Eulerian or ALE mesh, the fluid-structure (or ALE-Lagrangian) interaction may be modeled. This command allows for the tracking of certain coupling information related to the flow across, and the load on some selected Lagrangian surfaces.

2. The output parameters in the dbfsi ASCII file are:

- p = Averaged pressure on the surface being tracked (Pa)
- fx,fy,fz = Total force components (N) over the entity(ies) defined (acting at centroid of each surface)
- pleak = Accumulated porous leakage mass (Kg). LCIDPOR must be defined in the *CONSTRAINED_LAGRANGIAN_IN_SOLID card.
- mflux = Accumulated mass flowing through the surface being tracked (Kg).

Example:

Consider a model with a Lagrangian mesh overlaps with an Eulerian or ALE mesh. On the Lagrangian mesh, there are 4 Lagrangian surface sets over which some data is to be written out.

```

$...|...1...|...2...|...3...|...4...|...5...|...6...|...7...|...8
*DATABASE_FSI
$ DTOUT
  0.0001
$ DBFSI_ID      SID      SIDTYPE
   11           311        2
   12            3         1
   13            8         1
   14           12         0
$...|...1...|...2...|...3...|...4...|...5...|...6...|...7...|...8
$ This reads:
$ The 1st DBFSI_ID = ENTITY_ID = 11 => the surface entity ID is 11.
$ This DBFSI entity ID 11 is defined by a SETID = 311.
$ This SETID 311 is a SGSID = as specified by SIDTYPE=2
$...|...1...|...2...|...3...|...4...|...5...|...6...|...7...|...8
$ The output dbfsi looks like the following:
$...|...1...|...2...|...3...|...4...|...5...|...6...|...7...|...8

```

Fluid-structure interaction output
Number of surfaces: 4

id	p	fx	fy	fz	pleak	mflux
time= 0.0000E+00						
11	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
12	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
13	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
14	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
time= 0.10035E-03						
11	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.1594E-04
12	0.7193E+05	0.4228E-01	-0.5402E+01	0.1022E-03	0.1039E-11	-0.1159E-06
13	0.1277E+06	0.6381E+02	-0.4088E+03	-0.6202E+01	0.9993E-07	0.2497E-04
14	0.7193E+05	0.4228E-01	-0.5402E+01	0.1022E-03	0.1039E-11	-0.1159E-06
time= 0.20014E-03						
11	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.3198E-04
12	0.6512E+06	0.5008E+01	-0.5100E+02	-0.2486E-01	0.4432E-07	0.5135E-06
13	0.1335E+06	-0.6592E+01	-0.3123E+03	-0.2926E+01	0.1151E-05	0.1457E-03
14	0.6512E+06	0.5008E+01	-0.5100E+02	-0.2486E-01	0.4432E-07	0.5135E-06

```

$-----1-----2-----3-----4-----5-----6-----7-----
$ p = averaged pressure on the surface being tracked (Pa)
$ fx,fy,fz = Total force components (N) over the entity(ies) defined (acting at
$ centroid of each surface)
$ pleak = accumulated porous leakage mass (Kg). LCIDPOR must be defined
$ in the *CONSTRAINED_LAGRANGIAN_IN_SOLID card.
$ mflux = accumulated mass fluxing through the surface being tracked (Kg)
$-----1-----2-----3-----4-----5-----6-----7-----

```

*DATABASE

*DATABASE_HISTORY_OPTION

Options include:

BEAM
BEAM_SET
NODE
NODE
NODE_ID
NODE_LOCAL
NODE_LOCAL_ID
NODE_SET
NODE_SET_LOCAL
SHELL
SHELL_ID
SHELL_SET
SOLID
SOLID_ID
SOLID_SET
SPH
SPH_SET
TSHELL
TSHELL_ID
TSHELL_SET

Purpose: Control which nodes or elements are output into the binary history file, D3THDT, the ASCII file NODOUT, the ASCII file ELOUT and the ASCII file SPHOUT. Define as many cards as necessary. The next "*" card terminates the input. See also *DATABASE_BINARY_OPTION and *DATABASE_OPTION.

Card Format for options NODE, NODE_SET, SHELL, SHELL_SET, SOLID, SOLID_SET, SPH, SPH_SET, TSHELL, and TSHELL_SET

Cards 1,2,... 1 2 3 4 5 6 7 8

Variable	ID1	ID2	ID3	ID4	ID5	ID6	ID7	ID8
Type	I	I	I	I	I	I	I	I

*DATABASE

Read the second card for NODE_LOCAL_ID, otherwise, skip.

Card 1,2,...

1-7

8

Variable	HEADING	
Type	A70	

<u>VARIABLE</u>	<u>DESCRIPTION</u>
ID	NODE/NODE_SET set ID. The contents of the files are given in Table 9.1 for nodes. See the remark below concerning accelerometer nodes.
CID	Coordinate system ID for nodal output. See DEFINE_COORDINATE options.
REF	Output reference: EQ.0: Output is in the local system fixed for all time from the beginning of the calculation. EQ.1: Output is in the local system which is defined by the DEFINE_COORDINATE_NODES. The local system can change orientation depending on the movement of the three defining nodes. The defining nodes can belong to either deformable or rigid parts. EQ.2: Output is <i>relative</i> to the local system which is defined by the DEFINE_COORDINATE_NODES option. The local system can change orientation depending on the movement of the three defining nodes. If dynamic relaxation is used, the reference location is reset when convergence is achieved.
HEADING	A description of the nodal point. It is suggested that unique description be used. This description is written into the D3HSP file and into the ASCII database NODOUT.

Remarks:

1. If a node belongs to an accelerometer, see *ELEMENT_SEATBELT_ACCELEROMETER, and if it also appears as an active node in the NODE_LOCAL or NODE_SET_LOCAL keyword, the coordinate system, CID, transformations will be skipped and the LOCAL option will have no effect.

*DATABASE_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_COARSEN**

***DEFINE_BOX_DRAWBEAD**

***DEFINE_BOX_SPH**

***DEFINE_COORDINATE_NODES**

***DEFINE_COORDINATE_SYSTEM**

***DEFINE_COORDINATE_VECTOR**

***DEFINE_CURVE**

***DEFINE_CURVE_FEEDBACK**

***DEFINE_CURVE_SMOOTH**

***DEFINE_CURVE_TRIM_{*OPTION*}**

***DEFINE_SD_ORIENTATION**

***DEFINE_TABLE**

***DEFINE_TRANSFORMATION**

***DEFINE_VECTOR**

An additional option **_TITLE** may be appended to all the ***DEFINE** keywords. If this option is used then an addition line is read for each section in 80a format which can be used to describe the defined curve, table etc.. At present LS-DYNA does make use of the title. Inclusion of titles gives greater clarity to input decks.

Examples for the ***DEFINE** keyword can be found at the end of this section.

*DEFINE

*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	XMN	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.
XMN	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_COARSEN**

Purpose: Define a specific box-shaped volume indicating elements which are protected from mesh coarsening. See also *CONTROL_COARSEN.

Card Format

	1	2	3	4				
Variable	BOXID	XMN	XXM	YMN	YMX	ZMN	ZMX	IFLAG
Type	I	F	F	F	F	F	F	I
Default	none	0.0	0.0	0.0	0.0	0.0	0.0	0

VARIABLE**DESCRIPTION**

BOXID	Box ID. Define unique numbers.
XMN	Minimum x-coordinate.
XXM	Maximum x-coordinate.
YMN	Minimum y-coordinate.
YMX	Maximum y-coordinate.
ZMN	Minimum z-coordinate.
ZMX	Maximum z-coordinate.
IFLAG	Flag for protecting elements inside or outside of box. EQ.0: elements inside the box cannot be coarsened EQ.1: elements outside the box cannot be coarsened

Remarks:

1. Many boxes may be defined. If an element is protected by any box then it may not be coarsened.

*DEFINE

*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_BOX_SPH**

Purpose: Define a box-shaped volume. Two diagonally opposite corner points of a box are specified in global coordinates. Particle approximations of SPH elements are computed when particles are located inside the box. The load curve describes the motion of the maximum and minimum coordinates of the box.

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	BOXID	XMN	XXM	YMN	YMX	ZMN	ZMX	VID
Type	I	F	F	F	F	F	F	I
Default	none	0.0	0.0	0.0	0.0	0.0	0.0	0

Card 2 1 2 3 4 5 6 7

Variable	LCID	VD						
Type	I	I						
Default	0	0						

VARIABLE**DESCRIPTION**

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

<u>VARIABLE</u>	<u>DESCRIPTION</u>
VID	Vector ID for DOF, see *DEFINE_VECTOR.
LCID	Load curve ID to describe motion value versus time, see *DEFINE_CURVE
VD	Velocity/Displacement flag:: EQ.0: velocity, EQ.1: displacement

***DEFINE_COORDINATE_NODES**

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

Card Format

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

<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.
FLAG	Set to unity, 1, if the local system is to be updated each time step for the BOUNDARY_SPC nodal constraints and ELEMENT_BEAM type 6, the discrete beam element. Generally, this option when used with nodal SPC's is <i>not recommended</i> since it can cause excursions in the energy balance because the constraint forces at the node may go through a displacement if the node is partially constrained

Remark:

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

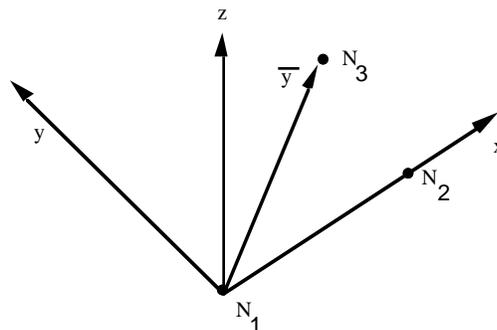


Figure 10.1. Definition of local coordinate system using three nodes.

*DEFINE

*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								

<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

*DEFINE_COORDINATE_VECTOR

Purpose: Define a local coordinate system with two vectors, see Figure 10.2. The vector cross product, $z = x \times xy$, 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	

<u>VARIABLE</u>	<u>DESCRIPTION</u>
CID	Coordinate system ID. A unique number has to be defined.
XX	X-coordinate on local x-axis. Origin lies at (0,0,0).
YX	Y-coordinate on local x-axis
ZX	Z-coordinate on local x-axis
XV	X-coordinate of local x-y vector
YV	Y-coordinate of local x-y vector
ZV	Z-coordinate of local x-y vector

Remark:

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

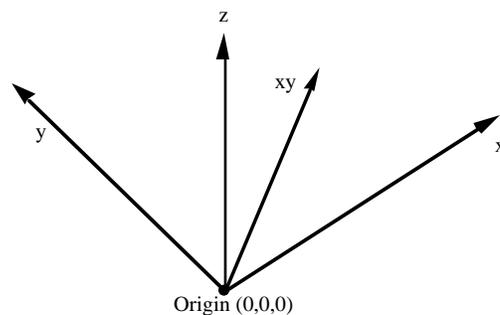


Figure 10.2. Definition of the coordinate system with two vectors.

***DEFINE_CURVE**

Purpose: Define a curve [for example, load (ordinate value) versus time (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-DYNA allows load curve ID's and table ID's to be used interchangeably. A unique number has to be defined. Note: <u>The magnitude of LCID is restricted to 5 significant digits. This limitation will be removed in a future release of LS-DYNA.</u>
SIDR	Stress initialization by dynamic relaxation: EQ.0: load curve used in transient analysis only or for other applications, EQ.1: load curve used in stress initialization but not transient analysis, EQ.2: load curve applies to both initialization and transient analysis.

*DEFINE

<u>VARIABLE</u>	<u>DESCRIPTION</u>
SFA	Scale factor for abscissa value. This is useful for simple modifications. EQ.0.0: default set to 1.0.
SFO	Scale factor for ordinate value (function). This is useful for simple modifications. EQ.0.0: default set to 1.0.
OFFA	Offset for abscissa values, see explanation below.
OFFO	Offset for ordinate values (function), see explanation below.
DATTYP	Data type. Usually 0, set to 1 <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_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			

VARIABLE**DESCRIPTION**

LCID	ID number for load curve to be scaled.
PID	Active part ID for load curve control
BOXID	Box ID. Elements of specified part ID contained in box are checked. If the box ID is set to zero the all elements of the active part are checked.
FLDID	Load curve ID which defines the flow limit diagram as shown in Figure 10.3. If the product of <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.

*DEFINE

<u>VARIABLE</u>	<u>DESCRIPTION</u>
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.

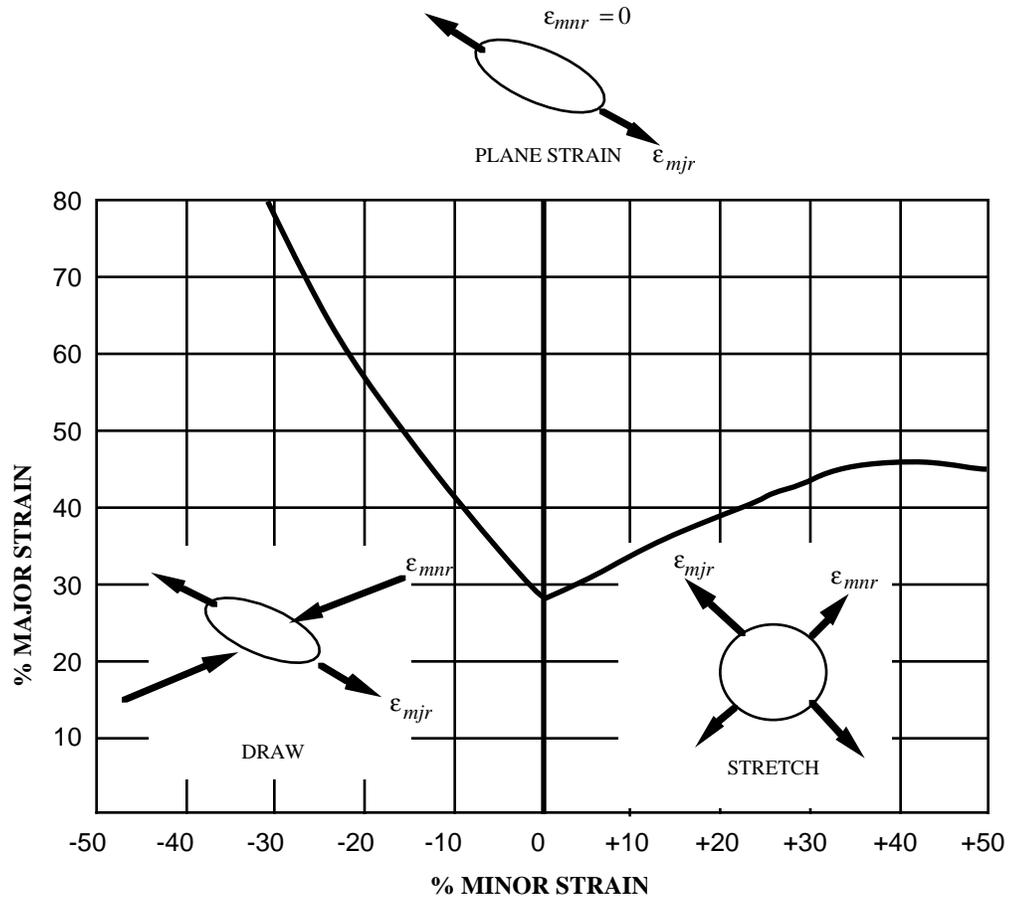


Figure 10.3. Flow limit diagram.

*DEFINE

*DEFINE_CURVE_SMOOTH

Purpose: Define a smoothly varying curve using few parameters. This shape is useful for velocity control of tools in metal forming applications.

Card Format

	1	2	3	4	5	6	7	8
Variable	LCID	SIDR	DIST	TSTART	TEND	TRISE	V0	
Type	I	I	F	F	F	F	F	
Default	none	none	none	none	none	none	none	

VARIABLE

DESCRIPTION

LCID	Load curve ID, must be unique.
SIDR	Stress initialization by dynamic relaxation: EQ.0: load curve used in transient analysis only or for other applications, EQ.1: load curve used in stress initialization but not transient analysis, EQ.2: load curve applies to both initialization and transient analysis.
DIST	Total distance tool will travel (area under curve).
TSTART	Time curve starts to rise
TEND	Time curve returns to zero. If TEND is nonzero, VMAX will be computed automatically to satisfy required travel distance DIST. Input either TEND or VMAX.
TRISE	Rise time
VMAX	Maximum velocity (maximum value of curve). If VMAX is nonzero, TEND will be computed automatically to satisfy required travel distance DIST. Input either TEND or VMAX.

Remarks:

See Figure 10.4.

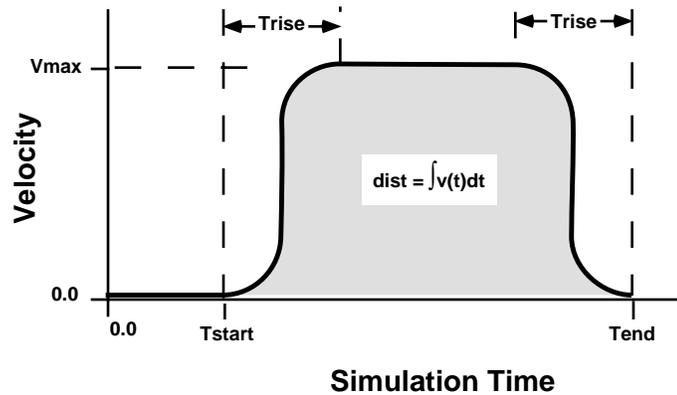


Figure 10.4. Smooth curve created automatically using *DEFINE_CURVE_SMOOTH. This shape is commonly used to control velocity of tools in metal forming applications as shown in the above graph, but can be used for other applications in place of any standard load curve.

*DEFINE

*DEFINE_CURVE_TRIM_{OPTION}

Available options include:

<BLANK>

3D

Purpose: Define a curve for trimming. Also, see *INTERFACE_SPRINGBACK. When option 3D is used, the trimming will be processed based on the element normal rather than the vector

Card Format

	1	2	3	4	5	6	7	8
Variable	TCID	TCTYPE	TFLG	TDIR	TCTOL	TOLN/IGB	NSEED	
Type	I	I	I	I	F	F	I	
Default	none	none	none	none	0.25	2.0	NONE	
Remarks	1,2,3			figure 10.5	4			

Card 2, 3, 4, etc. defined if and only if TCTYPE=1. Put one pair of points per card (2E20.0) Input is terminated when a "*" card is found.

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

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 10.5). EQ. 0: default vector (0,0,1) is used. Curve is defined in global XY plane, and projected onto mesh in global Z-direction to define trim line.
TCTOL	Tolerance limiting size of small elements created during trimming (see Figure 10.6). LT.0: "simple" trimming, producing jagged edge mesh
TOLN	The maximum gap between the trimming curve and the mesh. If the gap is bigger than this value, this section in the curve will not be used. Used only option 3D is chosen, If option 3D is not used, then IGB.EQ.0: trimming curve is defined in local coordinate system IGB.EQ.1: trimming curve is defined in global coordinate system
NSEED:	Any node in the side which will be kept after trimming. Used only when option 3D is chosen.\
C X	x-coordinate of trim curve Defined if and only if TCTYPE=1.
CY	y-coordinate of trim curve Defined if and only if TCTYPE=1.
FILENAME	Name of IGES database containing trim curve(s). Defined if and only if TCTYPE=2.

Remarks:

1. This command in combination with *ELEMENT_TRIM trims the requested parts before the job starts.
2. If the command *ELEMENT_TRIM does not exist the parts are trimmed after the job is terminated.

- 3 Pre-trimming (*ELEMENT_TRIM + *DEFINE_CURVE_TRIM) can handle adaptive mesh and post-trimming. The keyword *DEFINE_CURVE_TRIM by itself cannot deal with an adaptive mesh. See the detailed procedure outlined in the Remarks in the Section *INTERFACE_SPRINGBACK.
- 4 The trimming tolerance TCTOL limits the size of the smallest element created during trimming. A value of 0.0 places no limit on element size. A value of 0.5 restricts new elements to be at least half of the size of the parent element. A value of 1.0 allows no new elements to be generated, only repositioning of existing nodes to lie on the trim curve. A negative tolerance value activates "simple" trimming, where entire elements are removed, leaving a jagged edge.

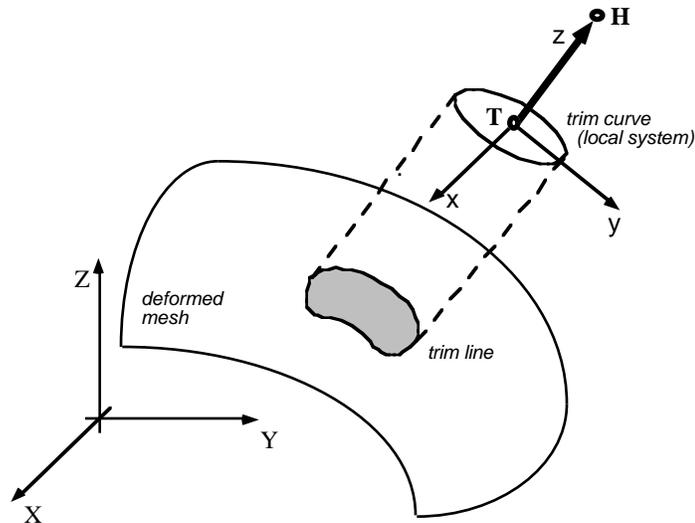


Figure 10.5. Trimming Orientation Vector. The tail (**T**) and head (**H**) points define a local coordinate system (x,y,z). The global coordinate system is named (X,Y,Z). The local x -direction is constructed in the Xz plane. If X and z nearly coincide ($|X \cdot z| > 0.95$), then the local x -direction is instead constructed in the Yz plane. Trim curve data is input in the x - y plane, and projected in the z -direction onto the deformed mesh to obtain the trim line.

*DEFINE

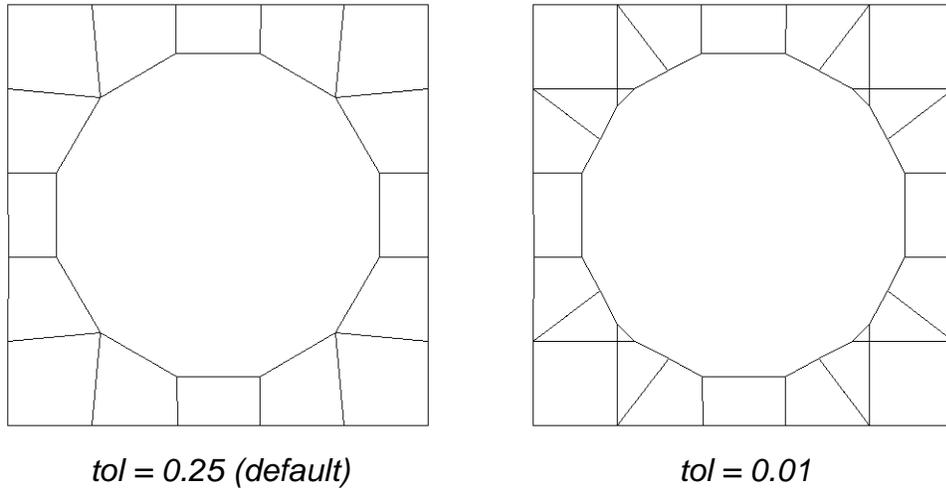


Figure 10.6 **Trimming Tolerance.** The tolerance limits the size of the small elements generated during trimming. The default tolerance (left) produces large elements. Using a tolerance of 0.01 (right) allows smaller elements, and more detail in the trim line.

***DEFINE_SD_ORIENTATION**

Purpose: Define orientation vectors for discrete springs and dampers. These orientation vectors are optional for this element class. Four alternative options are possible. With the first two options, IOP= 0 or 1, the vector is defined by coordinates and is fixed permanently in space. The third and fourth 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.

*DEFINE

<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_TABLE**

Purpose: Define a table. This input section is somewhat unique in that another keyword, ***DEFINE_CURVE**, is used as part of the input in this section. A table consists of a ***DEFINE_TABLE** card followed by n lines of input. Each of the n additional lines define a numerical value in ascending order corresponding to a ***DEFINE_CURVE** input which follows the ***DEFINE_TABLE** keyword and the related input. For example, to define strain rate dependency where it is desired to provide a stress versus strain curve for each strain rate, n strain rates would be defined following the ***DEFINE_TABLE** keyword. The curves then follow which make up the table. Each curve may have unique spacing and an arbitrary number of points in their definition. (Load curve ID's defined for the table may be referenced elsewhere in the input.) *However, the curves must not cross except at the origin and the curves must share the same origin and end point.* This rather awkward input is done for efficiency reasons related to the desire to avoid indirect addressing in the inner loops used in the constitutive model stress evaluation.

Card 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							

***DEFINE**

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-DYNA allows load curve ID's and table ID's to be used interchangeably.
VALUE	Load curve will be defined corresponding to this value, e.g., this value could be a strain rate, see purpose above.

Remark:

1. If for example, 10 stress-strain curves for 10 different strain rates are given, 10 cards with the ascending values of strain rate then follow the first card. Afterwards, 10 corresponding ***DEFINE_CURVE** specifications have to follow.

***DEFINE_TRANSFORMATION**

Purpose: Define a transformation for the INCLUDE_TRANSFORM keyword option. The *DEFINE_TRANSFORMATION command must be defined before the *INCLUDE_TRANSFORM command can be used.

Card Format Cards 1, 2, 3, 4, ... (The next “*” card terminates the input.) This set is a combination of a series of options listed in the table defined below.

Card 1 1 2 3 4 5 6 7 8

Variable	TRANID							
Type	I							
Default	none							

Card 2 1 2 3 4 5 6 7 8

Variable	OPTION	A1	A2	A3	A4	A5	A6	A7
Type	A	F	F	F	F	F	F	F

VARIABLE

DESCRIPTION

TRANID	Transform ID.
OPTION	For the available options see the table below.
A1-A7	Specified entity. Each card must have an option specified. See table below for the three available options.

*DEFINE

FORMAT (A10,7F10.0)		
OPTION	ENTITIES + ATTRIBUTES	FUNCTION
SCALE	a1, a2, a3	Scale the x, y, and z coordinates of a point by a1, a2, and a3, respectively. If zero, a default of unity is set.
ROTATE	a1, a2, a3, a4, a5, a6, a7	Rotate through an angle, a7, about a line with direction cosines a1, a2, and a3 passing through the point a4, a5, and a6.
TRANSL	a1, a2, a3	Translate the x, y, and z coordinates of a point by a1, a2, and a3, respectively.

The ordering of the SCALE, ROTATE, and TRANSL commands is important. It is generally recommend to first scale, then rotate, and finally translate the model.

The *DEFINE_TRANSFORMATION command is used 3 times to input the same dummy model and position it as follows:

1. Transformation id 1000 imports the dummy model (dummy.k) and rotates it 45 degrees about z-axis at the point (0.0,0.0,0.0)
2. Transformation id 2000 imports the same dummy model (dummy.k) and translates 1000 units in the x direction.
3. Transformation id 3000 imports the same dummy model (dummy.k) and translates 2000 units in the x direction. For each *DEFINE_TRANSFORMATION, the commands TRANSL, SCALE, and ROTATE are available. The transformations are applied in the order in which they are defined in the file, e.g., transformation id 1000 in this example would translate, scale and then rotate the model. *INCLUDE_TRANSFORM uses a transformation id defined by a *DEFINE_TRANSFORMATION command to import a model and perform the associated transformations. It also allows the user upon importing the model to apply offsets to the various entity ids and perform unit conversion of the imported model.

*KEYWORD

*DEFINE_TRANSFORMATION

1000

\$ option & dx& dy& dz&

TRANSL 0000.0 0.0 0.0

\$ option & dx& dy& dz&

SCALE 1.00 1.0 1.0

\$ option & dx& dy& dz& px& py& pz& angle&

ROTATE 0.00 0.0 1.0 0.00 0.00 0.0 45.00

*DEFINE_TRANSFORMATION

2000

\$ option & dx& dy& dz&

TRANSL 1000.0 0.0 0.0

*DEFINE_TRANSFORMATION

\$ trandid &

3000

\$ option & dx& dy& dz&

TRANSL 2000.0 0.0 0.0

*INCLUDE_TRANSFORM

dummy.k

\$idnoff & ideoff& idpoff& idmoff & idsoff & iddoftf& iddoftf &

0 0 0 0 0 0 0

\$ idroftf& ilctmf&

0 0

\$ fctmas& fcttim& fctlen& fcttem & incout&

1.0000 1.0000 1.00 1.0 1

\$ trandid &

1000

*INCLUDE_TRANSFORM

dummy.k

\$idnoff & ideoff& idpoff& idmoff & idsoff & iddoftf& iddoftf &

1000000 1000000 1000000 1000000 1000000 1000000 1000000

\$ idroftf& ilctmf&

1000000 1000000

\$ fctmas& fcttim& fctlen& fcttem & incout&

1.0000 1.0000 1.00 1.0 1

*DEFINE

```
$ traid &
    2000
*INCLUDE_TRANSFORM
dummy.k
$ idnoff &   ideoff&   idpoff& idmoff &   idsoff &   iddoft&   iddoft &
    2000000  2000000  2000000  2000000  2000000  2000000  2000000
$ idroft&   ilctmf&
    2000000  2000000
$ fctmas&   fcttim&   fctlen&   fcttem &   incout&
    1.0000   1.0000   1.00    1.0    1
$ traid &
    3000
*END
```

***DEFINE_VECTOR**

Purpose: Define a vector by defining the coordinates of two points.

Card Format

	1	2	3	4	5	6	7	8
Variable	VID	XT	YT	ZT	XH	YH	ZH	
Type	I	F	F	F	F	F	F	
Default	0	0.0	0.0	0.0	0.0	0.0	0.0	
Remarks								

VARIABLE**DESCRIPTION**

VID	Vector ID
XT	X-coordinate of tail of vector
YT	Y-coordinate of tail of vector
ZT	Z-coordinate of tail of vector
XH	X-coordinate of head of vector
YH	Y-coordinate of head of vector
ZH	Z-coordinate of head of vector

Remark:

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

***DEFINE**

***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**

VARIABLE	DESCRIPTION
SWSET	Set number for this automatic switch set. Must be unique.
CODE	Activation switch code. Defines the test to activate the automatic material switch of the part: EQ.0: switch takes place at time 1, EQ.1: switch takes place between time 1 and time 2 if rigid wall force (specified below) is zero, EQ.2: switch takes place between time 1 and time 2 if contact surface force (specified below) is zero, EQ.3: switch takes place between time 1 and time 2 if rigid wall force (specified below) is non-zero, EQ.4: switch takes place between time 1 and time 2 if contact surface force (specified below) is non-zero.
TIME 1	Switch will not take place before this time.
TIME 2	Switch will not take place after this time: EQ.0 Time 2 set to 1.0e20.
TIME 3	Delay period. After this part switch has taken place, another automatic switch will not take place for the duration of the delay period. If set to zero a part switch may take place immediately after this switch.
ENTNO	Rigid wall/contact surface number for switch codes 1, 2, 3, 4.
RELSW	Related switch set. The related switch set is another automatic switch set that must be activated before this part switch can take place: EQ.0: no related switch set.
PAIRED	Define a pair of related switches. EQ. 0: not paired EQ. 1: paired with switch set RELSW and is the Master switch. EQ.-1: paired with switch set RELSW and is the Slave switch .
NRBF	Flag to delete or activate nodal rigid bodies. If nodal rigid bodies or generalized, weld definitions are active in the deformable bodies that are switched to rigid, then the definitions should be deleted to avoid instabilities: EQ.0: no change, EQ.1: delete, EQ.2: activate.
NCSF	Flag to delete or activate nodal constraint set. If nodal constraint/spotweld definitions are active in the deformable bodies that are switched to rigid, then the definitions should be deleted to avoid instabilities: EQ.0: no change, EQ.1: delete, EQ.2: activate.

*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 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         4         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						

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

<u>VARIABLE</u>	<u>DESCRIPTION</u>
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**

VARIABLE	DESCRIPTION
PID	Part ID, see *PART.
XC	x-coordinate of center of mass
YC	y-coordinate of center of mass
ZC	z-coordinate of center of mass
TM	Translational mass
IXX	I_{xx} , xx component of inertia tensor
IXY	I_{xy}
IXZ	I_{xz}
IYY	I_{yy}
IYZ	I_{yz}
IZZ	I_{zz}

***ELEMENT**

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

***ELEMENT_BEAM_{OPTION}_{OPTION}**

***ELEMENT_DIRECT_MATRIX_INPUT**

***ELEMENT_DISCRETE**

***ELEMENT_INERTIA_{OPTION}**

***ELEMENT_MASS**

***ELEMENT_PLOTEL**

***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}_{OPTION}

Available options include:

<BLANK>

THICKNESS or SCALAR

PID

OFFSET

ORIENTATION

Purpose: Define two node elements including 3D beams, trusses, 2D axisymmetric shells, and 2D plane strain beam elements. The type of the element and its formulation is specified through the part ID (see *PART) and the section ID (see *SECTION_BEAM).

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

The PID option is used by the type 9 spot weld element only and is ignored for all other beam types. When the PID option is active an additional card is read that gives two part ID's that are tied by the spot weld element. If the PID option is inactive for the type 9 element the nodal points of the spot weld are located to the two nearest segments. The surface of each segment should project to the other and in the most typical case the node defining the weld, assuming only one node is used, should lie in the middle; however, this is not a requirement. Note that with the spotweld elements only one node is needed to define the weld, and two nodes are optional.

Card Format (10I8)

1 2 3 4 5 6 7 8 9 10

Variable	EID	PID	N ₁	N ₂	N ₃	RT1	RR1	RT2	RR2	LOCAL
Type	I	I	I	I	I	I	I	I	I	I
Default	none	none	none	none	none	0	0	0	0	2
Remarks					1	2,3	2,3	2,3	2,3	2,3

*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 SCALAR is specified after the keyword)

1 2 3 4 5 6 7 8 9 10

Variable	VOL	INER	CID	DOFN1	DOFN2
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										

*ELEMENT

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

1 2 3 4 5 6 7 8

Variable	WX1	WY1	WZ1	WX2	WY2	WZ2		
Type	F	F	F	F	F	F		
Default	0.0	0.0	0.0	0.0	0.0	0.0		
Remarks								

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

1 2 3 4 5 6 7 8

Variable	VX	VY	VZ					
Type	F	F	F					
Default	0.0	0.0	0.0					
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. For the zero length discrete beam elements where one end is attached to ground, set N2=-N1. In this case, a fully constrained nodal point will be created with a unique ID for node N2.

VARIABLE	DESCRIPTION
N3	Nodal point 3 for orientation. The third node, N3, is optional for beam types 3, 6, 7, 8, and 9 if the latter, type 9, has a circular cross section. The third node is used for the discrete beam, type 6, if and only if SCOOR is set to 2.0 in the *SECTION_BEAM input, but even in this case it is optional. An orientation vector can be defined directly by using the option, ORIENTATION. In this case N3 can be defined as zero.
RT1, RT2	Release conditions for translations at nodes N1 and N2, respectively: EQ.0: no translational degrees-of-freedom are released EQ.1: x-translational degree-of-freedom EQ.2: y-translational degree-of-freedom EQ.3: z-translational degree-of-freedom EQ.4: x and y-translational degrees-of-freedom EQ.5: y and z-translational degrees-of-freedom EQ.6: z and x-translational degrees-of-freedom EQ.7: x, y, and z-translational degrees-of-freedom (3DOF) This option does not apply to the spot weld, beam type 9.
RR1, RR2	Release conditions for rotations at nodes N1 and N2, respectively: EQ.0: no rotational degrees-of-freedom are released EQ.1: x-rotational degree-of-freedom EQ.2: y-rotational degree-of-freedom EQ.3: z-rotational degree-of-freedom EQ.4: x and y-rotational degrees-of-freedom EQ.5: y and z-rotational degrees-of-freedom EQ.6: z and x-rotational degrees-of-freedom EQ.7: x, y, and z-rotational degrees-of-freedom (3DOF) This option does not apply to the spot weld, beam type 9.
LOCAL	Coordinate system option: EQ.1: global coordinate system EQ.2: local coordinate system (default)
PARM1	Based on beam type: Type.EQ.1: beam thickness, s direction at node 1 Type.EQ.2: area Type.EQ.3: area Type.EQ.4: beam thickness, s direction at node 1 Type.EQ.5: beam thickness, s direction at node 1 Type.EQ.6: volume, see discription for VOL below. Type.EQ.7: beam thickness, s direction at node 1 Type.EQ.8: beam thickness, s direction at node 1 Type.EQ.9: beam thickness, s direction at node 1
PARM2	Based on beam type: Type.EQ.1: beam thickness, s direction at node 2 Type.EQ.2: I_{ss} Type.EQ.3: 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

*ELEMENT

VARIABLE

DESCRIPTION

	Type.EQ.6: Inertia, see discription for INER below. Type.EQ.7: beam thickness, s direction at node 2 Type.EQ.8: beam thickness, s direction at node 2 Type.EQ.9: beam thickness, s direction at node 2
PARM3	Based on beam type: Type.EQ.1: beam thickness, t direction at node 1 Type.EQ.2: I_{tt} Type.EQ.3: 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: print flag to SWFORC file. The default is taken from the SECTION_BEAM input. To override set PARM5 to 1. to surpress printing and to 2.0 to print.
PID1	Optional part ID for spot weld element type 9.
PID2	Optional part ID for spot weld element type 9.
VOL	Volume of discrete beam and scalar beam. If the mass density of the material model for the discrete beam is set to unity, the magnitude of the lumped mass can be defined here instead. This lumped mass is partitioned to the two nodes of the beam element. The translational time step size for the type 6 beam is dependent on the volume, mass density, and the translational stiffness values, so it is important to define this parameter.

<u>VARIABLE</u>	<u>DESCRIPTION</u>
	Defining the volume is also essential for mass scaling if the type 6 beam controls the time step size.
INER	Mass moment of inertia for the six degree of freedom discrete beam and scalar beam. This lumped inertia is partitioned to the two nodes of the beam element. The rotational time step size for the type 6 beam is dependent on the lumped inertia and the rotational stiffness values, so it is important to define this parameter if the rotational springs are active. Defining the rotational inertia is also essential for mass scaling if the type 6 beam rotational stiffness controls the time step size.
CID	Coordinate system ID for orientation, materials type ID (66-69, 93, 95, 97, 121, 146), see *DEFINE_COORDINATE_SYSTEM. If CID=0, a default coordinate system is defined in the global system or on the third node of the beam, which is used for orientation. This option is not defined for material types than act between two nodal points, such as cable elements. The coordinate system rotates with the discrete beam, see SCOR above.
DOFN1	Active degree-of-freedom at node 1, a number between 1 to 6 where 1 in x-translation and 4 is x-rotation.
DOFN2	Active degree-of-freedom at node 2, a number between 1 to 6.
WX1-WZ1	Offset vector at nodal point N1.
WX2-WZ2	Offset vector at nodal point N2.
VX,VY, VZ	Orientation vector at node N1. In this case the orientation nodal point N3, is defined as zero.

Remarks:

1. A plane through N₁, N₂, and N₃ 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, or local relative to the local beam coordinate system, see Figure 12.1. A local coordinate system is stored for each node of the beam element and the orientation of the local coordinate systems rotates with the node. To properly track the response, the nodal points with a released resultant are automatically replaced with new nodes to accommodate the added degrees-of-freedom. Then constraint equations are used to join the nodal points together with the proper release conditions imposed. **Consequently, nodal points which belong to beam elements which have release conditions applied cannot be subjected to other constraints such as applied displacement /velocity/acceleration boundary conditions, nodal rigid bodies, nodal constraint sets, or any of the constraint type contact definitions.** Force type loading conditions and penalty based contact algorithms may be used with this option.
3. Please note that this option may lead to nonphysical constraints if the translational degrees-of-freedom are released, but this should not be a problem if the displacements are infinitesimal.

***ELEMENT**

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.

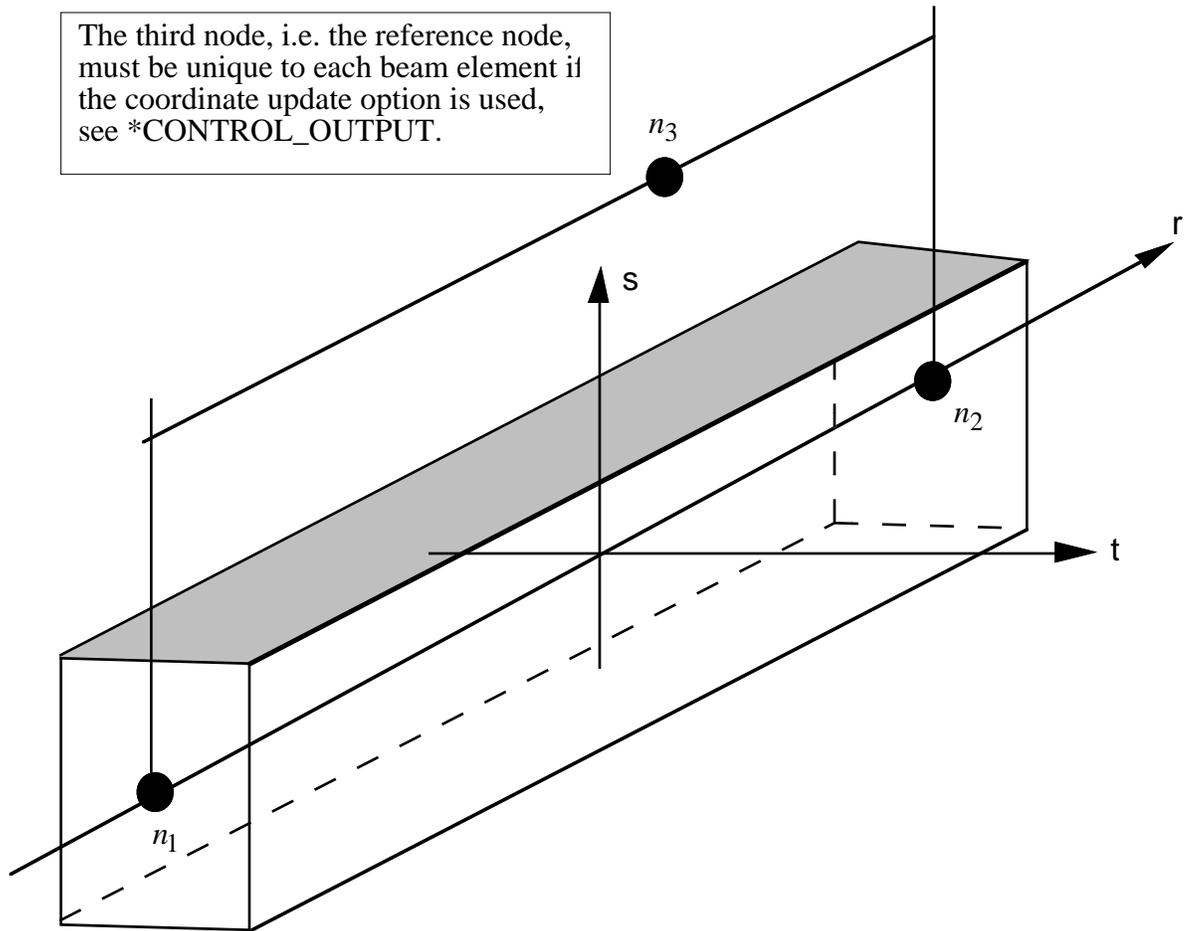


Figure 12.1. LS-DYNA beam elements. Node n_3 determines the initial orientation of the cross section.

*ELEMENT

*ELEMENT_DIRECT_MATRIX_INPUT

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

Card Format (I10)

Card 1 1 2 3 4 5 6 7 8

Variable	EID								
Type	I								

Card Format (A80)

Card 2

Variable	FILENAME
Type	C

Card Format (3A10)

Card 3 1 2 3 4 5 6 7 8

Variable	MASS	DAMP	STIF						
Type	C	C	C						

***ELEMENT**

<u>VARIABLE</u>	<u>DESCRIPTION</u>
EID	Super element ID.
FILENAME	Path and name of a file which contains the input matrices for this element.
MASS	Name of mass matrix in the file defined by FILENAME. This filename should be no more than eight characters to be compatible with NASTRAN.
DAMP	Name of damping matrix in the file defined by FILENAME. This filename should be no more than eight characters to be compatible with NASTRAN.
STIF	Name of stiffness matrix in the file defined by FILENAME. This filename should be no more than eight characters to be compatible with NASTRAN.

*ELEMENT

*ELEMENT_DISCRETE

Purpose: Define a discrete (spring or damper) element between two nodes or a node and ground. It is recommended that beam type 6, see *ELEMENT_BEAM and SECTION_BEAM, be used whenever possible, especially if orientation is specified. The latter option tends to be more accurate and cost effective. The *ELEMENT_DISCRETE option is no longer being developed and extended.

Note: These elements enter into the time step calculations. Care must be taken to ensure that the nodal masses connected by the springs and dampers are defined and unrealistically high stiffness and damping values must be avoided. **All rotations are in radians.**

Card Format (5I8,E16.0,I8,E16.0)

1 2 3 4 5 6 7 8 9 10

Variable	EID	PID	N1	N2	VID	S	PF	OFFSET
Type	I	I	I	I	I	F	I	F
Default	none	none	none	none	0	1.	0	0

VARIABLE

DESCRIPTION

EID	Element ID. A unique number has to be used.
PID	Part ID, see *PART.
N1	Nodal point 1.
N2	Nodal point 2. If zero, the spring/damper connects node N1 to ground.
VID	Orientation option. The orientation option should be used cautiously since forces, which are generated as the nodal points displace, are not orthogonal to rigid body rotation unless the nodes are coincident.. The type 6, 3D beam element, is recommended when orientation is required with the absolute value of the parameter SCOOR set to 2 or 3, since this option avoids rotational constraints. EQ.0: the spring/damper acts along the axis from node N1 to N2, NE.0: the spring/damper acts along the axis defined by the orientation vector, VID defined in the *DEFINE_SD_ORIENTATION section.
S	Scale factor on forces.
PF	Print flag: EQ.0: forces are printed in DEFORC file, see *DATABASE_OPTION, EQ.1: forces are not printed DEFORC file.
OFFSET	Initial offset. The initial offset is a displacement or rotation at time zero. For example, a positive offset on a translational spring will lead to a tensile force being developed at time zero.

*ELEMENT

*ELEMENT_INERTIA_{OPTION}

One option is available:

<BLANK>

OFFSET

to allow the lumped mass and inertia tensor to be offset from the nodal point. The nodal point can belong to either a deformable or rigid node.

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

Card Format (3I8)

	1	2	3	4	5	6	7	8	9	10
Variable	EID	NID	CSID							
Type	I	I	I							
Default	none	none	none							
Remarks			1							

Card Format (7E10.0)

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

*ELEMENT

Card Format (3E10.0) Define if and only if the OFFSET option is active.

1 2 3 4 5 6 7 8

Variable	X-OFF	Y-OFF	Z-OFF					
Type	F	F	F					
Default	0.	0.	0.					
Remarks		2	2					

VARIABLE

DESCRIPTION

EID	Element ID. A unique number must be used.
NID	Node ID. Node to which the mass is assigned.
CSID	Coordinate set ID EQ.0: global inertia tensor GE.1: principal moments of inertias with orientation vectors defined by Coordinate set CSID. See *DEFINE_COORDINATE_SYSTEM and *DEFINE_COORDINATE_VECTOR.
IXX	XX component of inertia tensor.
IXY	XY component of inertia tensor.
IXZ	XZ component of inertia tensor.
IYY	YY component of inertia tensor.
IYZ	YZ component of inertia tensor.
IZZ	ZZ component of inertia tensor.
MASS	Lumped mass
X-OFF	x-offset from nodal point.
Y-OFF	y-offset from nodal point.
Z-OFF	z-offset from nodal point.

Remarks:

1. The coordinate system cannot be defined 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	PID					
Type	I	I	F	I					
Default	none	none	0.	none					
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
PID	Part ID. This input is optional.

*ELEMENT

*ELEMENT_PLOTEL

Purpose: Define a null beam element for visualization.

Card Format (3I8)

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

VARIABLE

DESCRIPTION

EID	Element ID. A unique number must be used.
N1	Nodal point (end) 1.
N2	Nodal point (end) 2.

Remarks:

1. Part ID, 10000000, is assigned to PLOTEL elements.
2. PLOTEL element ID's must be unique with respect to other beam elements.

*ELEMENT

*ELEMENT_SEATBELT

Purpose: Define a seat belt element.

Card Format (5I8,E16.0)

1 2 3 4 5 6 7 8 9 10

Variable	EID	PID	N ₁	N ₂	SBRID	SLEN			
Type	I	I	I	I	I	F			
Default	none	none	none	none	none	0.0			
Remarks									

VARIABLE

DESCRIPTION

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

Remarks:

1. The retractor ID should be defined only if the element is initially **inside** a retractor, see *ELEMENT_SEATBELT_RETRACTOR.
2. Belt elements are single degree of freedom elements connecting two nodes. When the strain in an element is positive (i.e. the current length is greater 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. Whenever computed accelerations are compared to experimental results or whenever computed accelerations are compared between different runs, accelerometers are essential. Raw nodal accelerations contain considerable numerical noise and their comparisons are generally meaningless and, therefore, misleading.

Card Format

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

VARIABLE**DESCRIPTION**

SBACID	Accelerometer ID. A unique number must be used.
NID1	Node 1 ID
NID2	Node 2 ID
NID3	Node 3 ID
IGRAV	Gravitational accelerations due to body force loads. EQ.0: included in acceleration output EQ.1: removed from acceleration output
INTOPT	Integration option. If the accelerometer undergoes rigid body translation without rotation this option has no effect; however, if rotation occurs, option 1 may provide better agreement with the output of the accelerometer. EQ.0: velocities are integrated from the global accelerations and transformed into the local system of the accelerometer EQ.1: velocities are integrated directly from the local accelerations of the accelerometer.

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:

*ELEMENT

- local \mathbf{x} from node 1 to node 2,
- local \mathbf{z} perpendicular to the plane containing nodes, 1, 2, and 3 ($\mathbf{z} = \mathbf{x} \times \mathbf{a}$), where \mathbf{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	LMTFRC				
Type	I	F	I	F				
Default	0	0.0	0	0				
Remarks								

VARIABLE

DESCRIPTION

SBPRID

Pretensioner ID. A unique number has to be used.

SBPRTY

Pretensioner type (see Remark 2 below):

EQ.1: pyrotechnic retractor with force limits,

EQ.2: pre-loaded spring becomes active,

EQ.3: lock spring removed,

EQ.4: force versus time retractor.

EQ.5: pyrotechnic retractor (old type in version 950) but with optional force limiter, LMTFRC.

*ELEMENT

<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).
LMTFRC	Optional limiting force for retractor type 5. If zero, this option is ignored.

Remarks:

1. At least one sensor should be defined.

Pretensioners allow modeling of five types of active devices which tighten the belt during the initial stages of a crash. Types 1 and 5 represent a pyrotechnic device which spins the spool of a retractor, causing the belt to be reeled in. The user defines a pull-in versus time curve which applies once the pretensioner activates. Types 2 and 3 represent preloaded springs or torsion bars which move the buckle when released. The pretensioner is associated with any type of spring element including rotational. Note that the preloaded spring, locking spring and any restraints on the motion of the associated nodes are defined in the normal way; the action of the pretensioner is merely to cancel the force in one spring until (or after) it fires. With the second type, the force in the spring element is canceled out until the pretensioner is activated. In this case the spring in question is normally a stiff, linear spring which acts as a locking mechanism, preventing motion of the seat belt buckle relative to the vehicle. A preloaded spring is defined in parallel with the locking spring. This type avoids the problem of the buckle being free to 'drift' before the pretensioner is activated. Type 4, a force type, is described below.

To activate the pretensioner, the following sequence of events must occur:

1. Any one of up to four sensors must be triggered.
 2. Then a user-defined time delay occurs.
 3. Then the pretensioner acts.
2. In the 950 version of LS-DYNA, there are three types of seatbelt pretensioners that can be simulated. Types 2 and 3 are simple triggers for activating or deactivating springs, which then pull on the buckle. No changes have been made to these, and they are not discussed here. The type 1 pretensioner is intended to simulate a pyrotechnic retractor. The user inputs a load curve describing the pull-in of the pretensioner as a function of time. This pretensioner type interacts with the retractor, forcing it to pull in the amount of belt indicated. It works well, and does

exactly what it says it will do, but it can be difficult to use in practice. The reason for this is that it has no regard for the forces being exerted on the belt. If a pull-in of 20mm is specified at a particular time, then 20mm of belt will be pulled in, even if this results in unrealistic forces in the seatbelt. Furthermore, there was no explicit way to turn this pretensioner off. Once defined, it overrode the retractor completely, and the amount of belt passing into or out of the retractor depended solely on the load curve specified.

In the 970 version of LS-DYNA, the behavior of the type 1 pretensioner was changed due to user feedback regarding these shortcomings. The behavior now is fundamentally simpler, though a bit confusing to explain. Each retractor has a loading (and optional unloading) curve that describes the force on the belt element as a function of the amount of belt that has been pulled out of the retractor since the retractor locked. The new type 1 pretensioner acts as a shift of this retractor load curve. An example will make this clear. Suppose at a particular time that 5mm of belt material has left the retractor. The retractor will respond with a force corresponding to 5mm pull-out on its loading curve. But suppose this retractor has a type 1 pretensioner defined, and at this instant of time the pretensioner specifies a pull-in of 20mm. The retractor will then respond with a force that corresponds to (5mm + 20mm) on its loading curve. This results in a much larger force. The effect can be that belt material will be pulled in, but unlike in the 950 version, there is no guarantee. The benefit of this implementation is that the force vs. pull-in load curve for the retractor is followed and no unrealistic forces are generated. Still, it may be difficult to produce realistic models using this option, so two new types of pretensioners have been added. These are available in 970 versions 1300 and later.

The type 4 pretensioner takes a force vs. time curve, See Figure 12.2. Each time step, the retractor computes the desired force without regard to the pretensioner. If the resulting force is less than that specified by the pretensioner load curve, then the pretensioner value is used instead. As time goes on, the pretensioner load curve should drop below the forces generated by the retractor, and the pretensioner is then essentially inactive. This provides for good control of the actual forces, so no unrealistic values are generated. The actual direction and amount of belt movement is unspecified, and will depend on the other forces being exerted on the belt. This is suitable when the force the pretensioner exerts over time is known.

The type 5 pretensioner is essentially the same as the old type 1 pretensioner, but with the addition of a force limiting value. The pull-in is given as a function of time, and the belt is drawn into the retractor exactly as desired. However, if at any point the forces generated in the belt exceed the pretensioner force limit, then the pretensioner is deactivated and the retractor takes over. In order to prevent a large discontinuity in the force at this point, the loading curve for the retractor is shifted (in the abscissa) by the amount required to put the current (pull-out, force) on the load curve. For example, suppose the current force is 1000, and the current pull-out is -10 (10mm of belt has been pulled IN by the pretensioner). If the retractor would normally generate a force of 1000 after 25mm of belt had been pulled OUT, then the load curve is shifted to the left by 35, and remains that way for the duration of the calculation. So that at the current pull in of 10, it will generate the force normally associated with a pull out of 25. If the belt reaches a pull out of 5, the force will be as if it were pulled out 40 (5 + the shift of 35), and so on. This option is included for those who liked the general behavior of the old type 1 pretensioner, but has the added feature of the force limit to prevent unrealistic behavior.

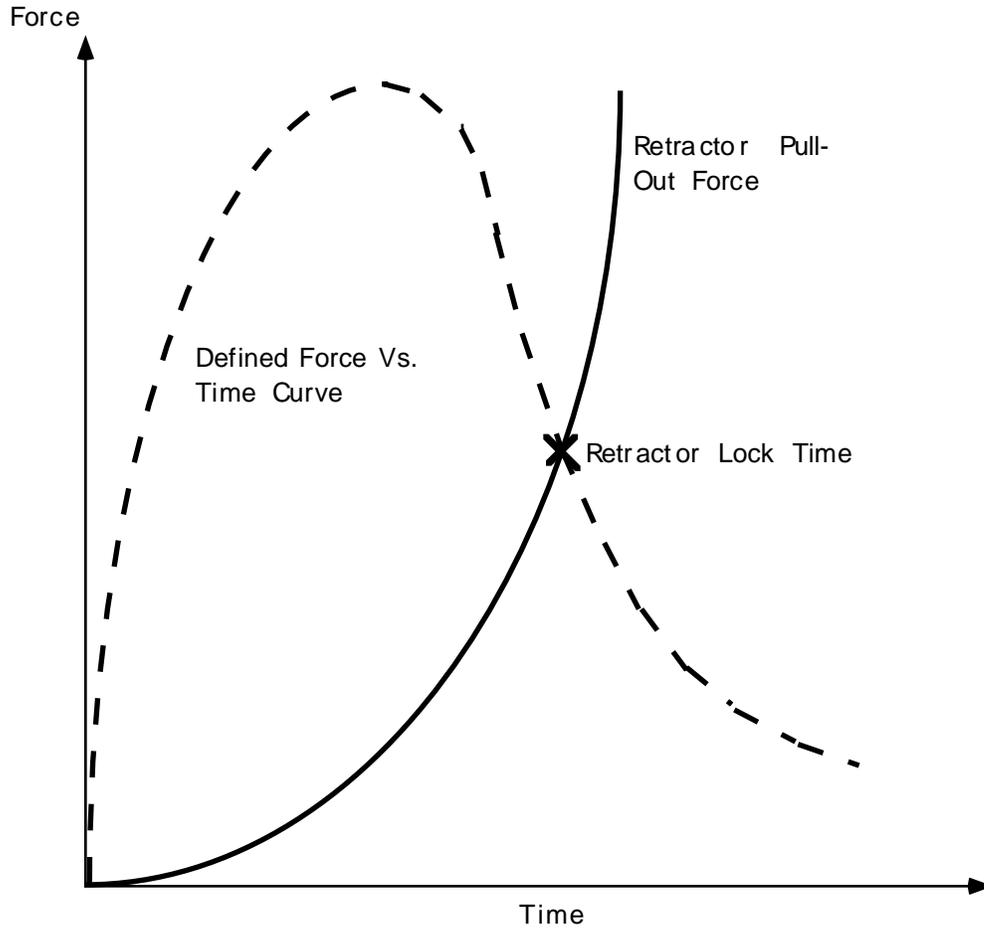


Figure 12.2. Force versus time pretensioner. At the intersection, the retractor locks.

***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

*ELEMENT

<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 12.4.
ULCID	Load curve for unloading (Pull-out, Force), see Figure 12.4.
LFED	Fed length, see explanation below.

Remarks:

1. The retractor node should not be on any belt elements. The element defined should have one node coincident with the retractor node but should not be inside the retractor.
2. At least one sensor should be defined.
3. The first point of the load curve should be $(0, T_{\min})$. T_{\min} is the minimum tension. All subsequent tension values should be greater than T_{\min} .
4. The unloading curve should start at zero tension and increase monotonically (i.e., no segments of negative or zero slope).

Retractors allow belt material to be paid out into a belt element. Retractors operate in one of two regimes: unlocked when the belt material is paid out, or reeled in under constant tension and locked when a user defined force-pullout relationship applies.

The retractor is initially unlocked, and the following sequence of events must occur for it to become locked:

1. Any one of up to four sensors must be triggered. (The sensors are described below.)
2. Then a user-defined time delay occurs.
3. Then a user-defined length of belt must be paid out (optional).
4. Then the retractor locks and once locked, it remains locked.

In the unlocked regime, the retractor attempts to apply a constant tension to the belt. This feature allows an initial tightening of the belt and takes up any slack whenever it occurs. The tension value is taken from the first point on the force-pullout load curve. The maximum rate of pull out or pull in is given by $0.01 \times \text{fed length}$ 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_{crit} into e1, where

$$L_{crit} = \text{fed length} - 1.1 \times \text{minimum length}$$

(minimum length defined on belt material input)

(fed length defined on retractor input)

element e2 emerges with an unstretched length of $1.1 \times$ 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 11.3, up to 4 sensors which can trigger unlocking, a time delay, a payout delay (optional), load and unload curve numbers, and the fed length. The

*ELEMENT

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 12.4. The final section can be very steep (but must have a finite slope).

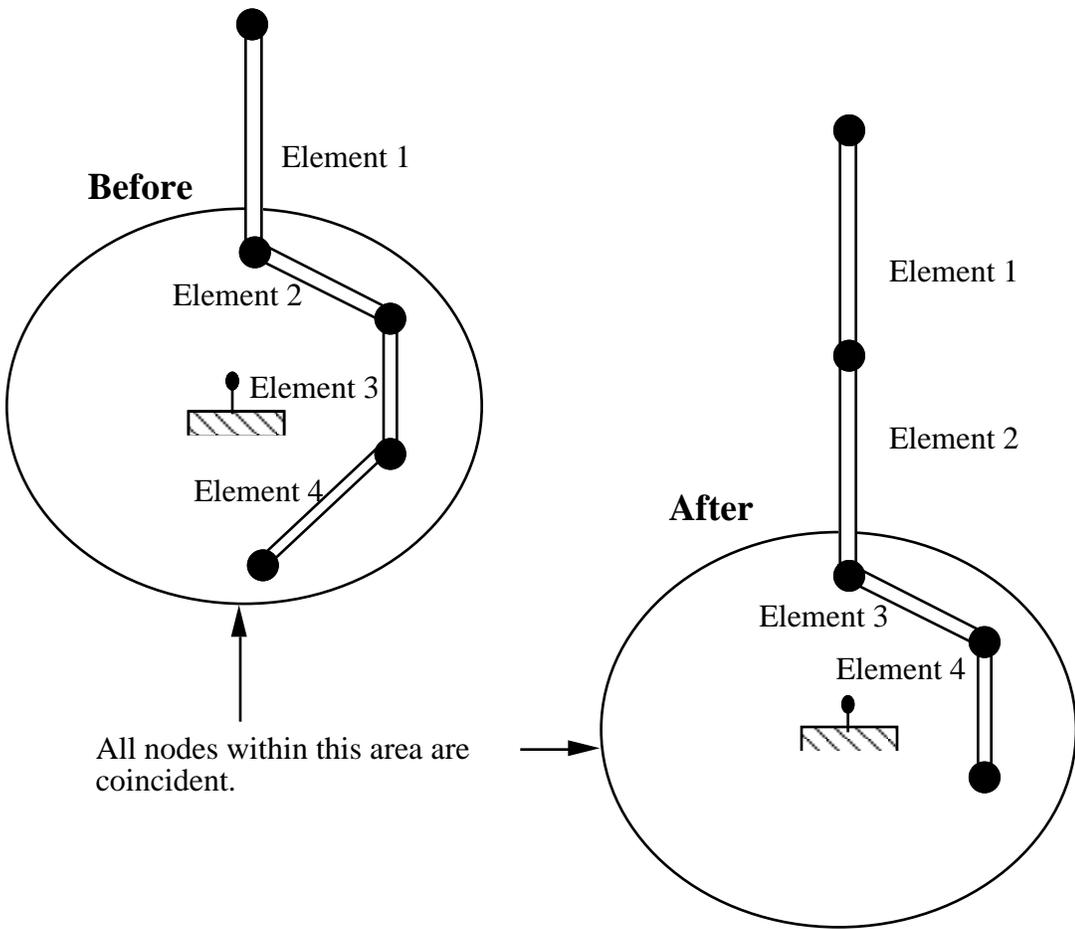


Figure 12.3. Elements in a retractor.

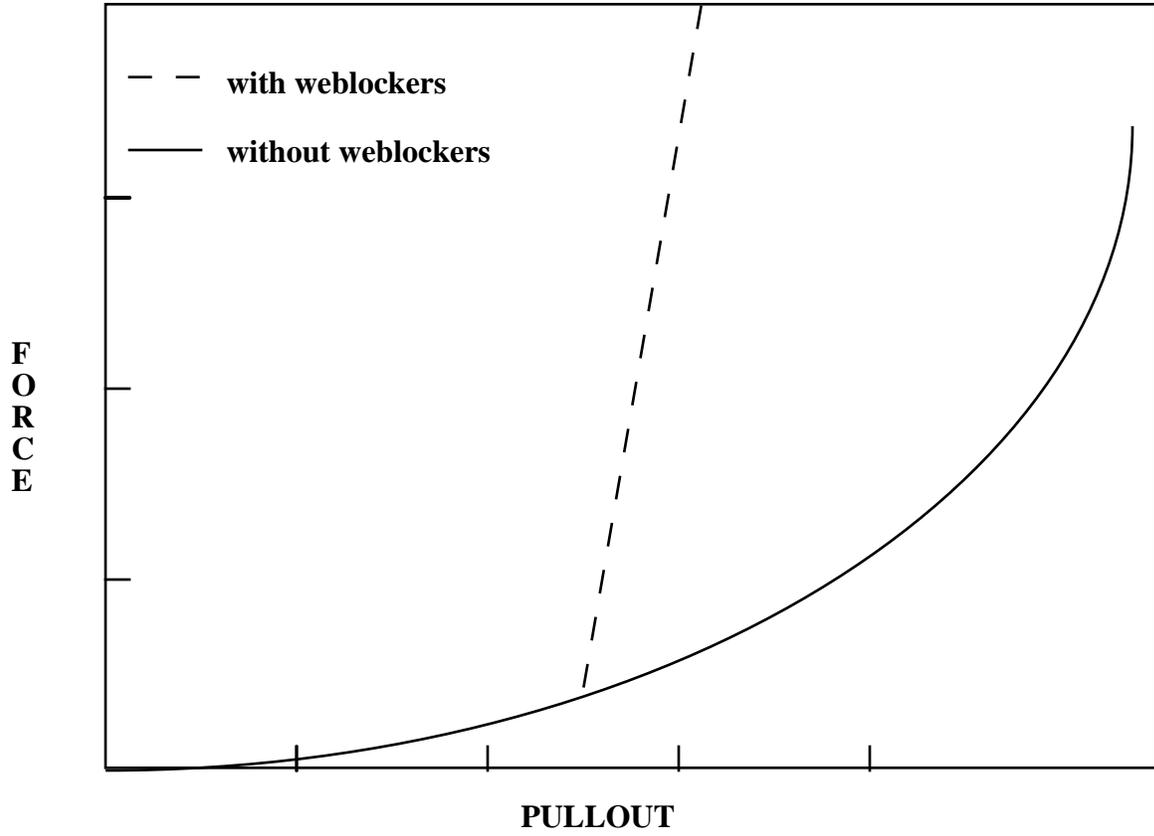


Figure 12.4. Retractor force pull characteristics.

***ELEMENT_SEATBELT_SENSOR**

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

Card Format

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

Second Card if SBSTYP=1

	1	2	3	4	5	6	7	8
Variable	NID	DOF	ACC	ATIME				
Type	I	I	F	F				
Default	0	0	0.0	0.0				
Remarks	1							

*ELEMENT

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				

<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:

*ELEMENT

Type 1 – When the magnitude of x-, y-, or z- acceleration of a given node has remained above a given level continuously for a given time, the sensor triggers. This does not work with nodes on rigid bodies.

Type 2 – When the rate of belt payout from a given retractor has remained above a given level continuously for a given time, the sensor triggers.

Type 3 – The sensor triggers at a given time.

Type 4 – The sensor triggers when the distance between two nodes exceeds a given maximum or becomes less than a given minimum. This type of sensor is intended for use with an explicit mass/spring representation of the sensor mechanism.

By default, the sensors are inactive during dynamic relaxation. This allows initial tightening of the belt and positioning of the occupant on the seat without locking the retractor or firing any pretensioners. However, a flag can be set in the sensor input to make the sensors active during the dynamic relaxation phase.

***ELEMENT_SEATBELT_SLIPRING**

Purpose: Define seat belt slip ring.

Card Format

	1	2	3	4	5	6	7	8
Variable	SBSRID	SBID1	SBID2	FC	SBRNID	LTIME	FCS	
Type	I	I	I	F	I	F	F	
Default	0	0	0	0.0	0	1.0E20	0.0	
Remarks	yes	yes	yes	yes	yes			

VARIABLE**DESCRIPTION**

SBSRID	Slipring ID. A unique number has to be used.
SBID1	Seat belt element 1 ID
SBID2	Seat belt element 2 ID
FC	Coulomb dynamic friction coefficient
SBRNID	Slip ring node, NID
LTIME	Slip ring lockup time. After this time no material is moved from one side of the slip ring to the other. This option is not active during dynamic relaxation.
FCS	Optional static Coulomb friction coefficient.

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 12.5) 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

*ELEMENT

between elements 1 and 2 and the coefficient of friction, μ . The tension in the belts are taken as T_1 and T_2 , where T_2 is on the high tension side and T_1 is the force on the low tension side. Thus, if T_2 is sufficiently close to T_1 , no slip occurs; otherwise, slip is just sufficient to reduce the ratio T_2/T_1 to $e^{\mu\Theta}$. No slip occurs if both elements are slack. The out-of-balance force at node B is reacted on the slinging node; the motion of node B follows that of slinging node.

If, due to slip through the slinging, 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 slinging and reappears on the other side (see Figure 12.5). 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 slinging.

To define a slinging, the user identifies the two belt elements which meet at the slinging, the friction coefficient, and the slinging node. The two elements must have a common node coincident with the slinging node. No attempt should be made to restrain or constrain the common node for its motion will automatically be constrained to follow the slinging node. Typically, the slinging 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.

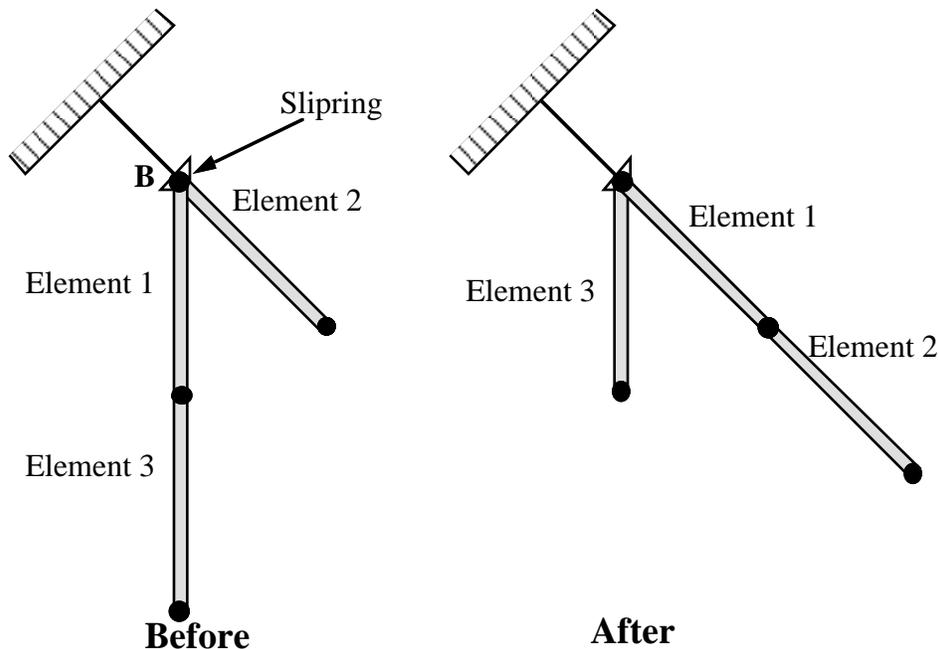


Figure 12.5. Elements passing through slinging.

***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.

3. Counterclockwise node numbering determines the top surface, see Figure 12.6.
4. Stresses and strain output in the binary databases are by default given in the global coordinate system. Stress resultants are output in the local coordinate system for the shell element.
5. Interior angles must be less than 180 degrees.
6. To allow for an arbitrary orientation of the shell elements within the finite element mesh, each ply in the composite has a unique material orientation angle which measures the offset from some reference in the element. Each integration point through the shell thickness, typically though not limited to one point per ply, requires the definition of the orientation angle at that point. The reference is determined by the angle ψ , which can be defined for each element on the element card, and is measured from the 1-2 element side. Figures 12.7 and 12.8 depict these angles.

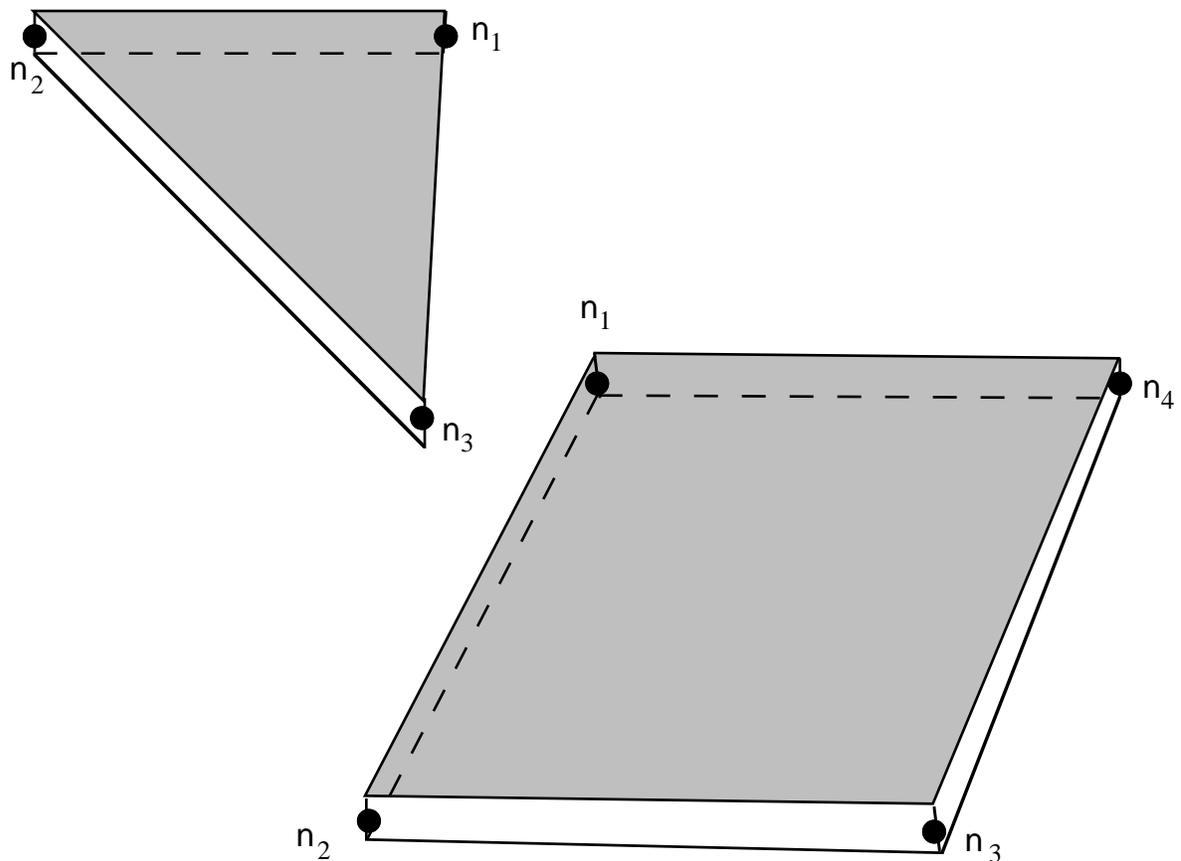


Figure 12.6. LS-DYNA shell elements. Counterclockwise node numbering determines the top surface.

*ELEMENT

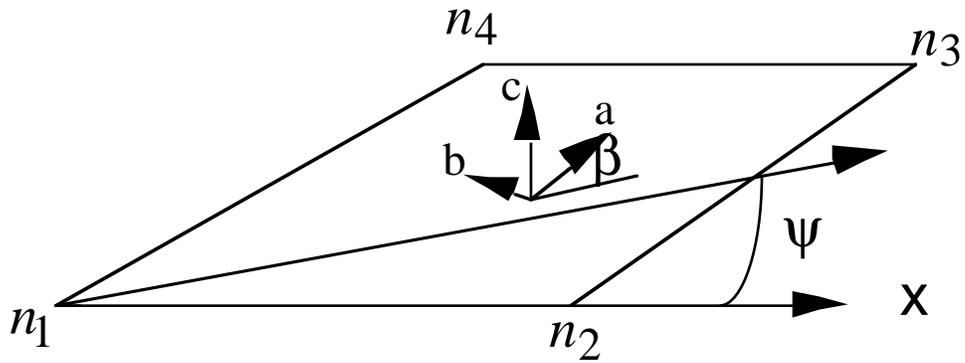


Figure 12.7 Orientation of material directions relative to the 1-2 side.

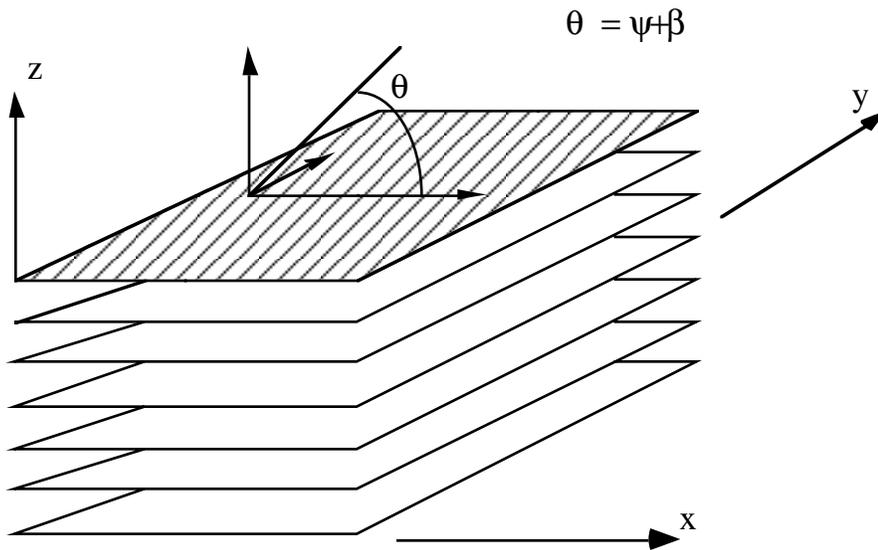


Figure 12.8. A multi-layer laminate can be defined. The angle β_i is defined for the i th lamina (integration point), see *SECTION_SHELL.

*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 (2I8)

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

Variable	EID	PID								
Type	I	I								
Default	none	none								
Remarks	1									

Card 2

Variable	N1	N2	N3	N4	N5	N6	N7	N8	N9	N10
Type	I	I	I	I	I	I	I	I	I	I
Default	none									
Remarks										

*ELEMENT

Optional Cards (Required if ORTHO is specified after the keyword)

Optional card 1 1 2 3 4 5 6 7 8 9 10

Variable	A1	A2	A3		
Type	F	F	F		
Default	0.	0.	0.		
Remarks	2				

Optional card 2

Variable	D1	D2	D3		
Type	F	F	F		
Default	0.	0.	0.		
Remarks	2				

VARIABLE

DESCRIPTION

EID	Element ID. A unique number has to be chosen.
PID	Part ID, see *PART.
N1	Nodal point 1
N2	Nodal point 2
N3	Nodal point 3
.	.
.	.
N8	Nodal point 8
A1	x-component of local material direction a, or else rotation angle in degrees (see remark 4).

<u>VARIABLE</u>	<u>DESCRIPTION</u>
A2	y-component of local material direction a.
A3	z-component of local material direction a.
D1	x-component of vector in the plane of the material vectors a and b.
D2	y-component of vector in the plane of the material vectors a and b.
D3	z-component of vector in the plane of the material vectors a and b.

Remarks:

- Four, six, and eight node elements are depicted in Figure 12.9 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. The input of nodes on the element cards for the tetrahedron and pentahedron elements is given by:

4-noded tetrahedron N1, N2, N3, N4, N4, N4, N4, 0, 0

6-noded pentahedron N1, N2, N3, N4, N5, N5, N6, N6, 0, 0

If hexahedrons are mixed with tetrahedrons and pentahedrons in the input under the same part ID, degenerate tetrahedrons and pentahedrons are used. One problem with degenerate elements is related to an uneven mass distribution (node 4 of the tetrahedron has five times the mass of nodes 1-3) which can make these elements somewhat unstable with the default time step size. By using the control flag under the keyword, *CONTROL_SOLID, automatic sorting can be invoked to treat the degenerate elements as type 10 and type 15 tetrahedrons and pentahedrons elements, respectively.

For elements with 4-8 nodes the card formats of LS_DYNA versions 940-970 are still valid. Card 2 is not defined in the older format.

Card Format (10I8)

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

Variable	EID	PID	N1	N2	N3	N4	N5	N6	N7	N8
Type	I	I	I	I	I	I	I	I	I	I

*ELEMENT

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 12.10):

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

These vectors are internally normalized within LS-DYNA.

3. Stress output for solid elements is in the global coordinate system by default.
4. If vector **d** is input as a zero length vector, then A1 is interpreted as a rotation angle in degrees which is used for AOPT=3 on various orthotropic material cards such as *MAT_OPTION TROPIC_ELASTIC.

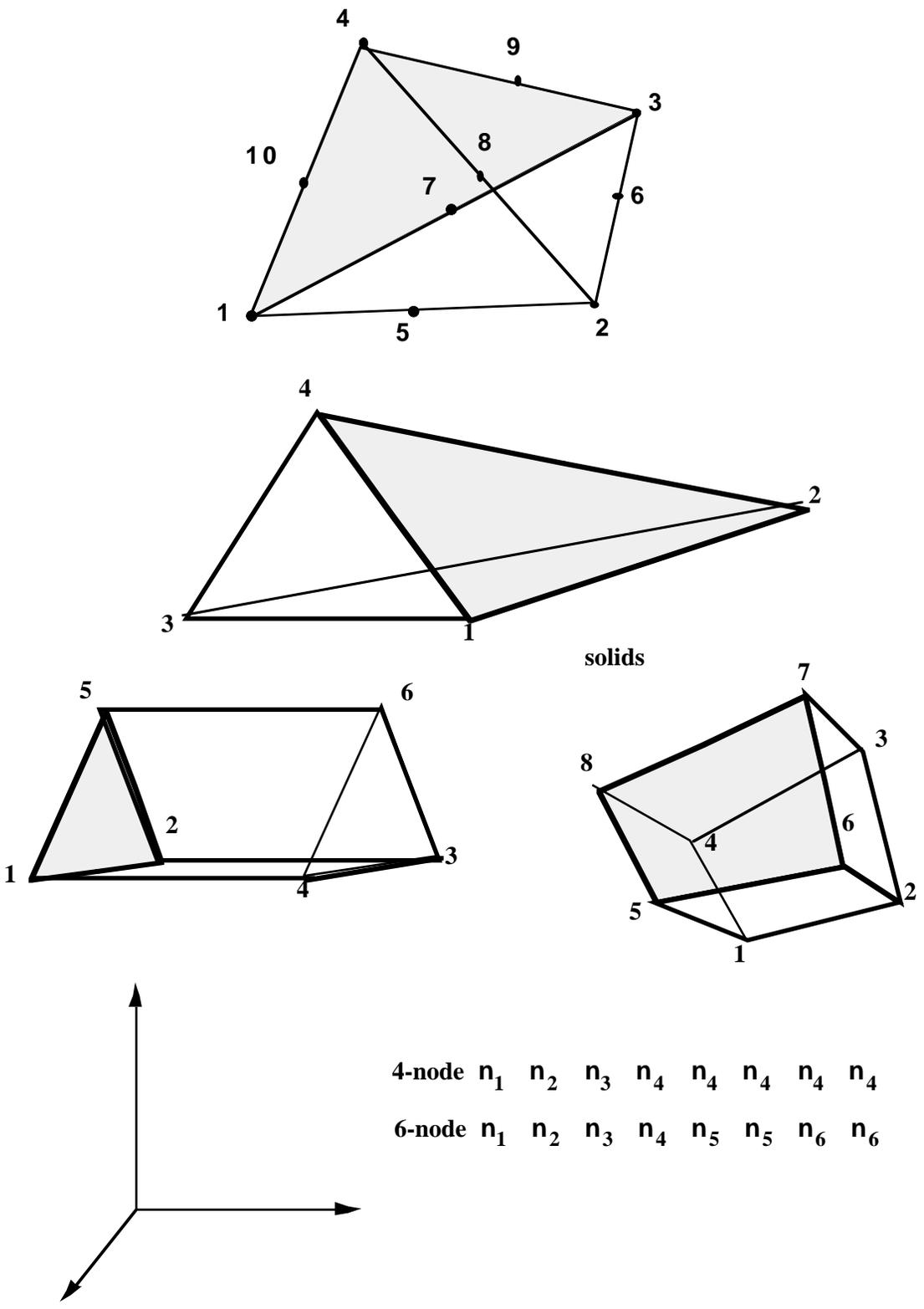


Figure 12.9. Four, six, and eight node solid elements. Nodes 1-4 are on the bottom surface.

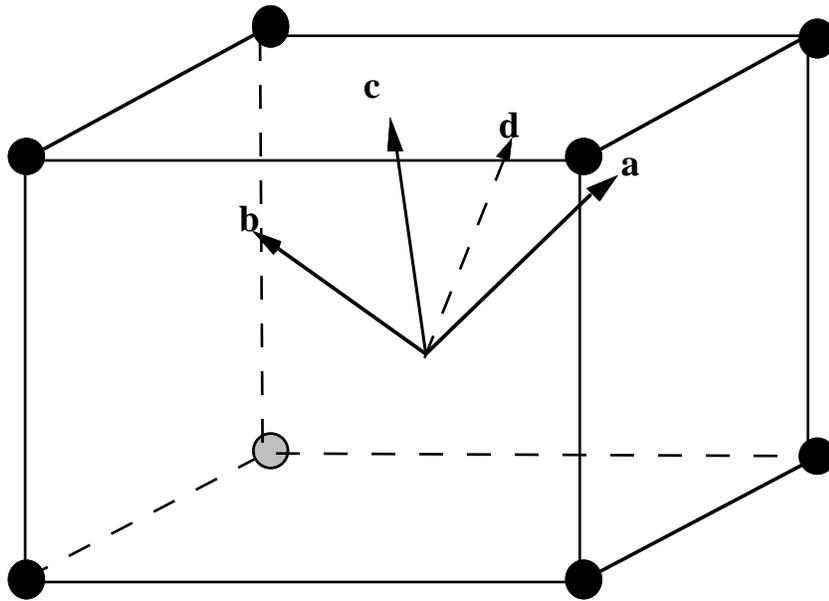


Figure 12.10 Two vectors **a** and **d** are defined and the triad is computed and stored. Vectors **b** and **d** lie in the same plane.

***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 (8I10)

Card 1 1 2 3 4 5 6 7 8

Variable	PSID							
Type	I							
Default	none							

VARIABLE

DESCRIPTION

PSID

Part set ID for trimming, see *SET_PART.

Remarks:

1. This keyword is used in combination with *DEFINE_CURVE_TRIM to trim the parts in PSID at time=0, i.e. before the simulation begins.

*ELEMENT_TSHELL

Purpose: Define an eight node thick shell element which is available with either fully reduced or selectively reduced integration rules. This plane stress element can be used as an alternative to the 4 node shell elements in cases where an 8-noded element is desired. Care must be taken in defining the element connectivity as N1 to N4 define the lower surface of the thick shell. The number of through-thickness integration points is defined by the user. The definition is completed by the *PART and *SECTION_TSHELL cards.

Card Format (10I8)

	1	2	3	4	5	6	7	8	9	10
Variable	EID	PID	N ₁	N ₂	N ₃	N ₄	N ₅	N ₆	N ₇	N ₈
Type	I	I	I	I	I	I	I	I	I	I
Default	none	none	none	none	none	none	none	none	none	none
Remarks			1							

VARIABLE

DESCRIPTION

EID	Element ID. Unique numbers have to be used.
PID	Part ID, see *PART.
N1	Nodal point 1
N2	Nodal point 2
N3	Nodal point 3
.	.
.	.
N8	Nodal point 8

*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 12.11. 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.

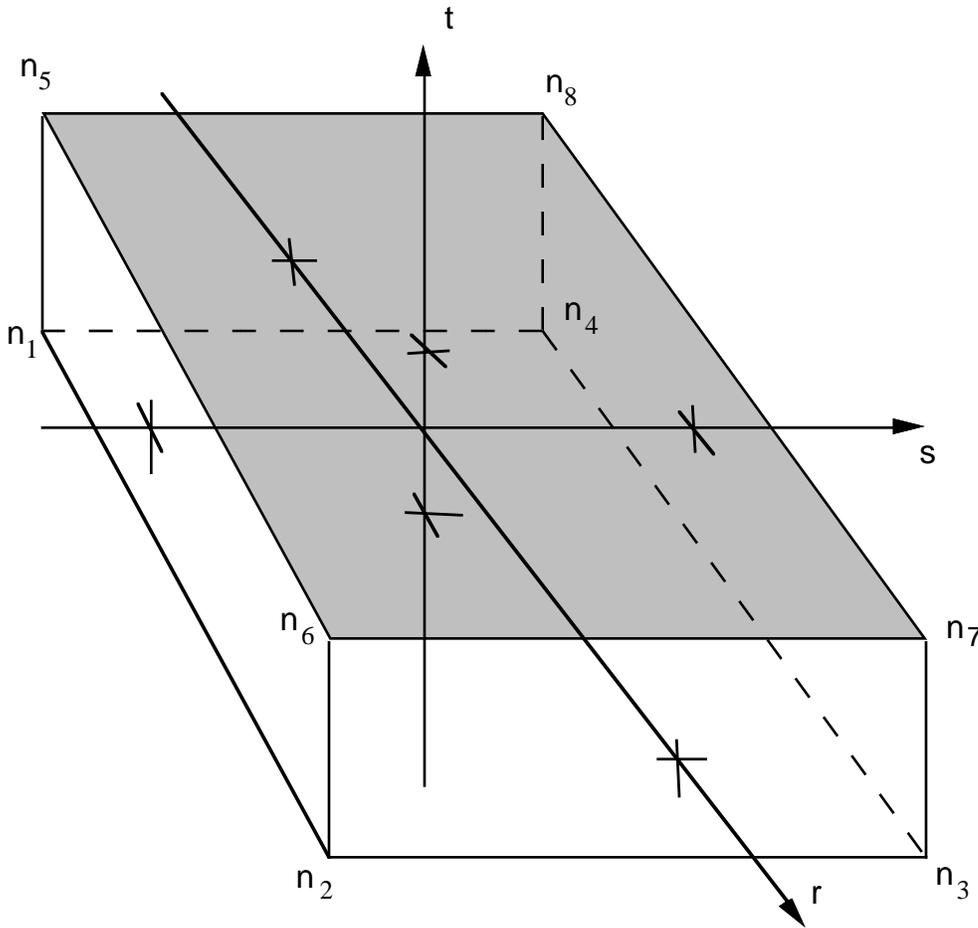


Figure 12.11. Solid 8-node Shell Element.

***EOS**

LS-DYNA has historically referenced equations of state by type identifiers. Below these identifiers are given with the corresponding keyword name in the order that they appear in the manual. The equations of state can be used with a subset of the materials that are available for solid elements.

TYPE 1:	*EOS_LINEAR_POLYNOMIAL
TYPE 2:	*EOS_JWL
TYPE 3:	*EOS_SACK_TUESDAY
TYPE 4:	*EOS_GRUNEISEN
TYPE 5:	*EOS_RATIO_OF_POLYNOMIALS
TYPE 6:	*EOS_LINEAR_POLYNOMIAL_WITH_ENERGY_LEAK
TYPE 7:	*EOS_IGNITION_AND_GROWTH_OF_REACTION_IN_HE
TYPE 8:	*EOS_TABULATED_COMPACTION
TYPE 9:	*EOS_TABULATED
TYPE 10:	*EOS_PROPELLANT_DEFLAGRATION
TYPE 11:	*EOS_TENSOR_PORE_COLLAPSE
TYPE 12:	*EOS_IDEAL_GAS
TYPE 14:	*EOS_JWLB

An additional option **_TITLE** may be appended to all the ***EOS** keywords. If this option is used then an addition line is read for each section in 80a format which can be used to describe the equation of state. At present LS-DYNA does not make use of the title. Inclusion of titles gives greater clarity to input decks.

*EOS

DEFINITIONS & NOTES ON SOME COMMONLY USED PARAMETERS:

In order to prescribe the boundary and/or initial thermodynamic condition, manual computations are often necessary. Some conventions or definitions may simplify this process. Some basic variables are defined in the following. Since many of these variables have already been denoted by different symbols, the notations used here are unique in this section only! They are presented to only clarify their usage. A corresponding SI unit set is also presented as example. First consider a few volumetric parameters since they are a measure of compression (or expansion).

$$\text{Volume} = V \left[m^3 \right]$$

$$\text{Mass} = M \left[Kg \right]$$

$$\text{Current specific volume} = v = \frac{V}{M} = \frac{1}{\rho} \left[\frac{m^3}{Kg} \right]$$

$$\text{Reference specific volume} = v_0 = \frac{V_0}{M} = \frac{1}{\rho_0} \left[\frac{m^3}{Kg} \right]$$

$$\text{Relative volume} = v_r = \frac{V}{V_0} = \frac{V/M}{V_0/M} = \frac{v}{v_0} = \frac{\rho_0}{\rho}$$

$$\text{Current normalized volume increment} = \frac{dv}{v} = \frac{v - v_0}{v} = 1 - \frac{1}{v_r} = 1 - \frac{\rho}{\rho_0}$$

$$\text{We usually deal with a volumetric parameter called } \mu = \frac{1}{v_r} - 1 = \frac{v_0 - v}{v} = -\frac{dv}{v} = \frac{\rho}{\rho_0} - 1$$

$$\text{Sometimes another volumetric parameter is used: } \eta = \frac{v_0}{v} = \frac{\rho}{\rho_0}$$

$$\text{Thus the relation between } \mu \text{ and } \eta \text{ is } \mu = \frac{v_0 - v}{v} = \eta - 1$$

The following table summarizes these volumetric parameters.

VARIABLES	COMPRESSION	NO LOAD	EXPANSION
$v_r = \frac{v}{v_0} = \frac{\rho_0}{\rho}$	< 1	1	> 1
$\eta = \frac{1}{v_r} = \frac{v_0}{v} = \frac{\rho}{\rho_0}$	> 1	1	< 1
$\mu = \frac{1}{v_r} - 1 = \eta - 1$	> 0	0	< 0

Recalling the definition of an isotropic bulk modulus is (Y. C. Fung, “Foundations of Solid Mechanics”, page 129, equation 5) $\frac{\Delta v}{v} = -\frac{P}{K}$. This may be rewritten as $P = K \left[-\frac{\Delta v}{v} \right] = K\mu$. The

bulk modulus, K , thus is equivalent to C_1 in *EOS_LINEAR_POLYNOMIAL when all other coefficients are zero. Next consider some internal energy parameters since they represent the thermal energy state (temperature dependency) of a system.

$$\text{Internal energy} = E = MC_v T \quad [Joule]$$

Note that the capital “ E ” here is the absolute internal energy. It is not the same as that used in the subsequent *EOS section.

$$\text{Specific internal energy} = e = \frac{E}{M} = C_v T \quad \left[\frac{Joule}{Kg} \right]$$

$$\text{Internal energy per current specific volume} = e_v = \frac{M}{V} C_v T = \rho C_v T = \frac{C_v T}{v} \quad \left[\frac{Joule}{m^3} = \frac{N}{m^2} \right]$$

$$\text{Internal energy per reference specific volume} = e_{v_0} = \frac{M}{V_0} C_v T = \rho_0 C_v T = \frac{C_v T}{v_0} \quad \left[\frac{Joule}{m^3} = \frac{N}{m^2} \right]$$

Note that e_{v_0} is actually the capital “ E ” or “ E_0 ” referred to in this *EOS section. The “reference” state is a unique state with respect to which the material stress tensor is computed. Since μ is frequently used as a measure of compression (or expansion), v_0 is very critical in computing the pressure level in a material. Incorrect choice of v_0 would lead to incorrect pressure computed. In general, v_0 is chosen such that at zero compression or expansion the material should be in equilibrium with its ambient surrounding. The relative volume at time zero, $v_{r0} = \frac{v_{t=0}}{v_0}$, corresponds to “ V_0 ” in this EOS section. “ V_0 ” is not the specific volume ($V_0 \neq v_0$).

A thermodynamic state of a material, not undergoing any chemical reactions or phase change, is generally defined by two state variables, for example, $P = P(\rho, T) = P(v, e) = P(v_r, e_v) = P(\mu, e_{v_0})$, etc. are possible forms relating pressure to two other state variables. In fact the last form is popularly used to compute pressure. In this section, the equation of state (EOS) is used to calculate pressure. Different forms of the EOS describe different types of volumetric compression (or expansion) behaviors of different materials. The coefficients for each EOS model come from data-fitting, phenomenological descriptions, or derivations based on classical thermodynamics, etc.

BOUNDARY OR INITIAL CONDITION SETTING:

Usually, setting the pressure alone is not enough to provide a determinate state of a material. It takes one more thermodynamic variable. Frequently, the pressure is specified (given), and the user must make an assumption on either e_{v_0} or μ . Consider two possibilities (a) e_{v_0} is defined or assumed from which e_{v_0} may be computed, or (2) v_r is defined or assumed from which μ may be obtained. With a known pressure, the users can back-calculate, via the EOS, for the needed value of e_{v_0} or v_r to fill into the input file.

*EOS

*EOS_LINEAR_POLYNOMIAL

Purpose: Define coefficients for linear polynomial EOS and initialize the initial thermodynamic state of the material. This is done by defining E0 and V0 below.

Card 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	The 0 th polynomial equation coefficient.
C1	The 1 st polynomial equation coefficient (when used by itself, this is the elastic bulk modulus, i.e. it cannot be used for deformation that is beyond the elastic regime).
...	...
C6	The 6 th polynomial equation coefficient.
E0	Initial internal energy per unit reference specific volume
V0	Initial relative volume

Remarks:

1. The linear polynomial equation of state is linear in internal energy. The pressure is given by:

$$P = C_0 + C_1\mu + C_2\mu^2 + C_3\mu^3 + (C_4 + C_5\mu + C_6\mu^2) E.$$

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

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

The units of E are the units of pressure.

2. When $C_0 \neq 0$, it does not necessarily mean that the initial pressure is zero, $P_0 \neq C_0$! The initial pressure also depends on μ_0 and E_0 , too. The pressure in a material is computed from the whole equation above, $P = P(\mu, E)$. It is always preferable to initialize the initial condition based on μ_0 and E_0 . The use of $C_0 \neq 0$ must be done with caution as it may change the form and behavior of the material. The safest way is to use the whole EOS equation to check for the pressure value. For example for an ideal gas, it may not be proper to define $C_4 = C_5 = \gamma - 1$ and $C_0 \neq 0$ at the same time.

*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 relative volume.

Remarks:

The JWL equation of state defines the pressure as

$$p = A \left(1 - \frac{\omega}{R_1 V} \right) e^{-R_1 V} + B \left(1 - \frac{\omega}{R_2 V} \right) e^{-R_2 V} + \frac{\omega E}{V},$$

and is usually used for detonation products of high explosives.

***EOS_SACK_TUESDAY**

This is Equation of state Form 3.

Card Format

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

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

VARIABLE

DESCRIPTION

EOSID Equation of state ID
C
S1
S2
S3
GAMAO
A
E0 Initial internal energy
V0 Initial relative volume

Remarks:

The Gruneisen equation of state with cubic shock velocity-particle velocity defines pressure for compressed materials as

$$p = \frac{\rho_0 C^2 \mu \left[1 + \left(1 - \frac{\gamma_0}{2} \right) \mu - \frac{a}{2} \mu^2 \right]}{\left[1 - (S_1 - 1) \mu - S_2 \frac{\mu^2}{\mu + 1} - S_3 \frac{\mu^3}{(\mu + 1)^2} \right]^2} + (\gamma_0 + a\mu) E.$$

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; γ_0 is the Gruneisen gamma; a is the first order volume correction to γ_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	α
BETA	β
E0	Initial internal energy
V0	Initial relative volume

Remarks:

The ratio of polynomials equation of state defines the pressure as

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

where

$$F_i = \sum_{j=0}^n A_{ij} \mu^j \quad n = 4 \text{ if } i < 3$$

$$\mu = \frac{\rho}{\rho_0} - 1 \quad n = 3 \text{ if } i \geq 3$$

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					

VARIABLE

DESCRIPTION

EOSID Equation of state label

C0

C1

C2

C3

C4

C5

C6

E0 Initial internal energy

V0 Initial relative volume

LCID Load curve ID defining the energy deposition rate

Remarks:

This polynomial equation of state, linear in the internal energy per initial volume, E , is given by

$$p = C_0 + C_1\mu + C_2\mu^2 + C_3\mu^3 + (C_4 + C_5\mu + C_6\mu^2)E$$

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 μ^2 to zero, i.e.,

$$C_2 = C_6 = 0$$

Internal energy, E , is increased according to an energy deposition rate versus time curve whose ID is defined in the input.

*EOS

*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		

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

EOSID Equation of state label

A

B

XP1

VARIABLE	DESCRIPTION
XP2	
FRER	
G	
R1	
R2	
R3	
R5	
R6	
FMXIG	
FREQ	
GROW1	
EM	
AR1	
ES1	
CVP	Heat capacity of reaction products
CVR	Heat capacity of unreacted HE
EETAL	
CCRIT	
ENQ	Heat of reaction
TMP0	Initial temperature (°K)
GROW2	
AR2	
ES2	
EN	
FMXGR	
FMNGR	

*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

$$\frac{\partial F}{\partial t} = \text{freq} (1-F)^{\text{frer}} (V_e^{-1} - 1 - \text{ccrit})^{\text{cetal}} \quad (\text{Ignition})$$

$$+ \text{grow} 1 (1-F)^{\text{esl}} F^{\text{arl}} p^{\text{em}} \quad (\text{Growth})$$

$$+ \text{grow} 2 (1-F)^{\text{es2}} F^{\text{ar2}} p^{\text{en}} \quad (\text{Completion})$$

The ignition rate is set equal to zero when $F \geq \text{fmixig}$, 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	γ
E0	Initial internal energy
V0	Initial relative volume

Remarks:

The tabulated compaction model is linear in internal energy. Pressure is defined by

$$p = C(\epsilon_v) + \gamma T(\epsilon_v)E$$

in the loading phase. The volumetric strain, ϵ_v is given by the natural logarithm of the relative volume. Unloading occurs along the unloading bulk modulus to the pressure cutoff. Reloading always follows the unloading path to the point where unloading began, and continues on the loading path, see Figure 13.1. Up to 10 points and as few as 2 may be used when defining the tabulated functions. LS-DYNA will extrapolate to find the pressure if necessary.

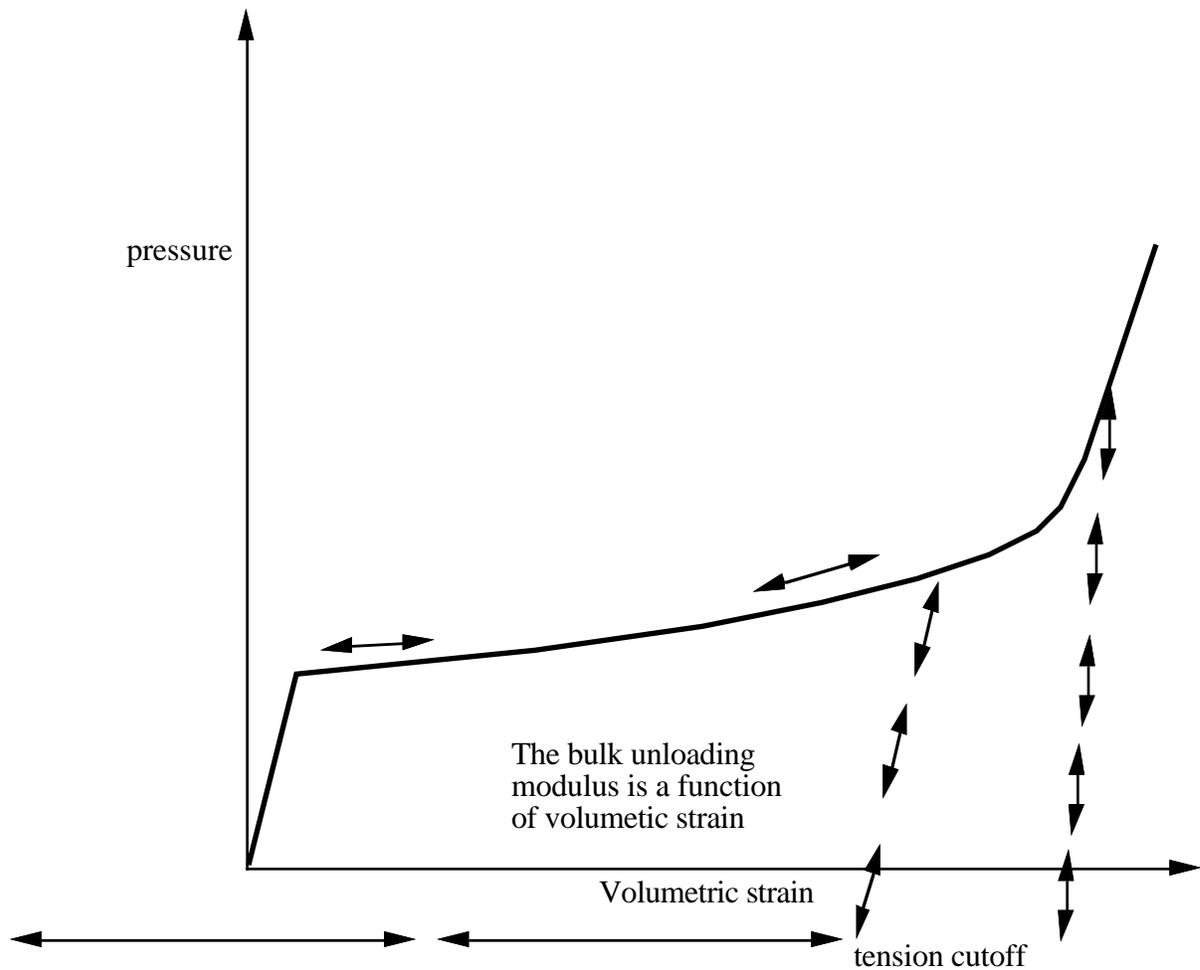


Figure 13.1. Pressure versus volumetric strain curve for Equation of state Form 8 with compaction. In the compacted states the bulk unloading modulus depends on the peak volumetric strain.

*EOS

*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	γ
E0	Initial internal energy
V0	Initial relative volume

Remarks:

The tabulated equation of state model is linear in internal energy. Pressure is defined by

$$P = C (\epsilon_V) + \gamma T(\epsilon_V) E$$

The volumetric strain, ϵ_V is given by the natural logarithm of the relative volume. Up to 10 points and as few as 2 may be used when defining the tabulated functions. LS-DYNA will extrapolate to find the pressure if necessary.

*EOS

*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		

VARIABLE	DESCRIPTION
A	Product JWL coefficient
B	Product JWL coefficient
XP1	Product JWL coefficient
XP2	Product JWL coefficient
FRER	Unreacted Co-volume
G	Product wC_V
R1	Unreacted JWL coefficient
R2	Unreacted JWL coefficient
R3	Unreacted wC_V
R5	Unreacted JWL coefficient
R6	Unreacted JWL coefficient
FMXIG	Initial Fraction Reacted F_0
FREQ	Initial Pressure P_0
GROW1	First burn rate coefficient
EM	Pressure Exponent (1 st term)
AR1	Exponent on F (1 st term)
ES1	Exponent on (1-F) (1 st term)
CVP	Heat capacity products
CVR	Heat capacity unreacted
EETAL	Extra, not presently used
CCRIT	Product co-volume
ENQ	Heat of Reaction
TMP0	Initial Temperature (298°K)
GROW2	Second burn rate coefficient

*EOS

<u>VARIABLE</u>	<u>DESCRIPTION</u>
AR2	Exponent on F (2 nd term)
ES2	Exponent on (1-F) (2 nd term)
EN	Pressure Exponent (2 nd term)
FMXGR	Maximum F for 1 st term
FMNGR	Minimum F for 2 nd 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:

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

where p is pressure (in Mbars), V is the relative specific volume (inverse of relative density), ω is the Gruneisen coefficient, C_v is heat capacity (in Mbars -cc/cc°K), T is temperature in °K, d is the co-volume, and A , B , R_1 and R_2 are constants. Setting $A=B=0$, yields the van der Waal's co-volume equation of state. The JWL equation of state is generally useful at pressures above several kilobars, while the van der Waal's is useful at pressures below that range and above the range for which the perfect gas law holds. Of course, setting $A=B=d=0$ yields the perfect gas law. If accurate values of ω and C_v plus the correct distribution between "cold" compression and internal energies are used, the calculated temperatures are very reasonable and thus can be used to check propellant performance.

The reaction rate used for the propellant deflagration process is of the form:

$$\frac{\partial F}{\partial t} = \underset{\text{for } 0 < F < F_{limit1}}{Z(1-F)^y F^x p^w} + \underset{\text{for } F_{limit2} < F < 1}{V(1-F)^u Frp^s}$$

where F is the fraction reacted ($F = 0$ implies no reaction, $F = 1$ is complete reaction), t is time, and p is pressure (in Mbars), $r, s, u, w, x, y, F_{limit1}$ and F_{limit2} are constants used to describe the pressure dependance and surface area dependence of the reaction rates. Two (or more) pressure dependant reaction rates are included in case the propellant is a mixture or exhibited a sharp change in reaction rate at some pressure or temperature. Burning surface area dependences can be approximated using the $(1-F)^y F^x$ terms. Other forms of the reaction rate law, such as Arrhenius temperature dependent $e^{-E/RT}$ type rates, can be used, but these require very accurate temperatures calculations. Although the theoretical justification of pressure dependent burn rates at kilobar type pressures is not complete, a vast amount of experimental burn rate versus pressure data does demonstrate this effect and hydrodynamic calculations using pressure dependent burn accurately simulate such experiments.

The deflagration reactive flow model is activated by any pressure or particle velocity increase on one or more zone boundaries in the reactive material. Such an increase creates pressure in those zones and the decomposition begins. If the pressure is relieved, the reaction rate decreases and can go to zero. This feature is important for short duration, partial decomposition reactions. If the pressure is maintained, the fraction reacted eventually reaches one and the material is completely converted to product molecules. The deflagration front rates of advance through the propellant calculated by this model for several propellants are quite close to the experimentally observed burn rate versus pressure curves.

To obtain good agreement with experimental deflagration data, the model requires an accurate description of the unreacted propellant equation of state, either an analytical fit to experimental compression data or an estimated fit based on previous experience with similar materials. This is also true for the reaction products equation of state. The more experimental burn rate, pressure production and energy delivery data available, the better the form and constants in the reaction rate equation can be determined.

Therefore, the equations used in the burn subroutine for the pressure in the unreacted propellant

$$P_u = R1 \cdot e^{-R5 \cdot V_u} + R2 \cdot e^{-R6 \cdot V_u} + \frac{R3 \cdot T_u}{V_u - FRER}$$

where V_u and T_u are the relative volume and temperature respectively of the unreacted propellant. The relative density is obviously the inverse of the relative volume. The pressure P_p in the reaction products is given by:

$$P_p = A \cdot e^{-XP1 \cdot V_p} + B \cdot e^{-XP2 \cdot V_p} + \frac{G \cdot T_p}{V_p - CCRIT}$$

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 FMXGR , the GROW1 term is set equal to zero, and, if F is less than FMNGR , the GROW2 term is zero. Thus, two separate (or overlapping) burn rates can be used to describe the rate at which the propellant decomposes.

This equation of state subroutine is used together with a material model to describe the propellant. In the airbag propellant case, a null material model (type #10) can be used. Material type #10 is usually used for a solid propellant or explosive when the shear modulus and yield strength are defined. The propellant material is defined by the material model and the unreacted equation of state until the reaction begins. The calculated mixture states are used until the reaction is complete and then the reaction product equation of state is used. The heat of reaction, ENQ , is assumed to be a constant and the same at all values of F but more complex energy release laws could be implemented.

***EOS_TENSOR_PORE_COLLAPSE**

This is Equation of state Form 11.

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	EOSID	NLD	NCR	MU1	MU2	IE0	EC0	
Type	I	F	F	F	F	F	F	

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

<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 (μ_1), and the excess compression point required to completely crush the material (μ_2). From this data and the maximum excess compression the material has attained (μ_{max}), the pressure for any excess compression (μ) can be determined.

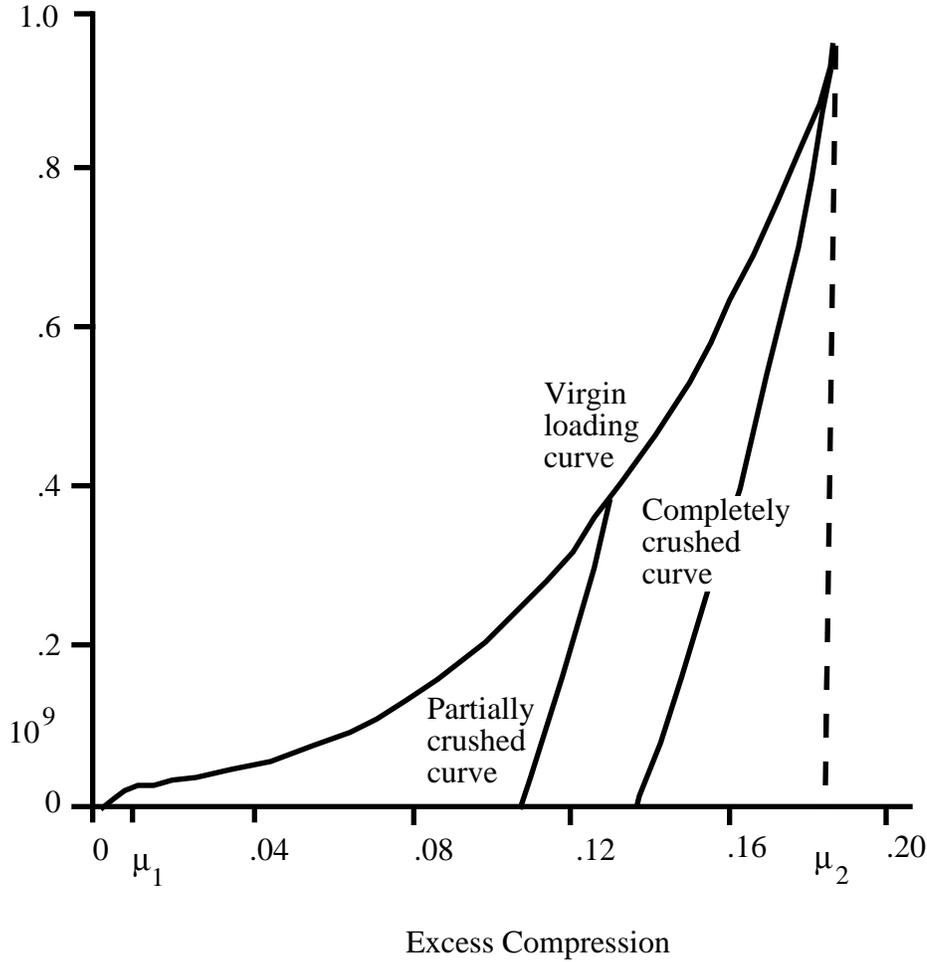


Figure 13.2. Pressure versus compaction curve.

Unloading occurs along the virgin loading curve until the excess compression surpasses μ_1 . After that, the unloading follows a path between the completely crushed curve and the virgin loading curve. Reloading will follow this curve back up to the virgin loading curve. Once the excess compression exceeds μ_2 , then all unloading will follow the completely crushed curve.

For unloading between μ_1 and μ_2 a partially crushed curve is determined by the relationship:

$$P_{pc}(\mu) = P_{cc} \left(\frac{(1 + \mu_B)(1 + \mu)}{1 + \mu_{\max}} - 1 \right).$$

where

$$\mu_B = P_{cc}^{-1}(P_{\max})$$

and the subscripts pc and cc refer to the partially crushed and completely crushed states, respectively. This is more readily understood in terms of the relative volume (V).

$$V = \frac{1}{1 + \mu}$$

$$P_{pc}(V) = P_{cc} \left(\frac{V_B}{V_{\min}} V \right)$$

This representation suggests that for a fixed $V_{\min} \left(= \frac{1}{\mu_{\max} + 1} \right)$ the partially crushed curve will separate linearly from the completely crushed curve as V increases to account for pore recovery in the material.

The bulk modulus K is determined to be the slope of the current curve times one plus the excess compression:

$$K = \frac{\partial P}{\partial \mu} (1 + \mu)$$

The slope $\frac{\partial P}{\partial \mu}$ for the partially crushed curve is obtained by differentiation as:

$$\frac{\partial P}{\partial \mu} = \frac{\partial P_{cc} \left(\frac{(1 + \mu_B)(1 + \mu)}{1 + \mu_{\max}} \right) (1 + \mu_B)}{\partial \mu (1 + \mu_{\max})}$$

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_IDEAL_GAS

This is Equation of state Form 12.

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	EOSID	CV0	CP0	CL	CQ	T0	V0	
Type	I	F	F	F	F	F	F	

<u>VARIABLE</u>	<u>DESCRIPTION</u>
EOSID	Equation of state label
CV0	Specific heat constant for definition of Cv
CP0	Specific heat constant for definition of Cp
CL	Specific heat constant for linear Cv and Cp
CQ	Specific heat constant for quadratic Cv and Cp
T0	Initial temperature
V0	Initial relative volume

Remarks:

1. The pressure in the ideal gas law is defined as

$$p = \rho(C_p - C_v)T$$
$$C_p = C_{p0} + C_L T + C_Q T^2$$
$$C_v = C_{v0} + C_L T + C_Q T^2$$

where C_p and C_v are the specific heat capacities at constant pressure and at constant volume, respectively. ρ is the density.

2. When dealing with Eulerian/ALE models, the ideal gas model is implemented to preserve the adiabatic state during advection. The adiabatic state is conserved on the expense of a perfect internal energy conservation.

4. The ideal gas model is good for low density gas only. Deviation from the ideal gas behavior may be indicated by the compressibility factor defined as

$$Z = \frac{Pv}{RT}$$

When Z deviates from 1, the gas behavior deviates from ideal.

*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				

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

<u>VARIABLE</u>	<u>DESCRIPTION</u>
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 ω are input constants defined above.

JWLB input constants for some common explosives as found in Baker and Stiel [1997] are given in the following table.

	TATB	LX-14	PETN	TNT	Octol 70/30
ρ_0 (g/cc)	1.800	1.821	1.765	1.631	1.803
E0 (Mbar)	.07040	.10205	.10910	.06656	.09590
DCJ (cm/ μ 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
ω	.27952	.38375	.32357	.30270	.39023
A λ 1	1423.9	18307.	12.257	.00000	.011929
B λ 1	14387.	1390.1	52.404	1098.0	18466.
R λ 1	19.780	19.309	43.932	15.614	20.029
A λ 2	5.0364	4.4882	8.6351	11.468	5.4192
B λ 2	-2.6332	-2.6181	-4.9176	-6.5011	-3.2394
R λ 2	1.7062	1.5076	2.1303	2.1593	1.5868

***HOURGLASS**

***HOURGLASS**

Purpose: Define hourglass and bulk viscosity properties which are referenced via HGID in the *PART command. Properties specified here, when invoked for a particular part, override those in *CONTROL_HOURGLASS and *CONTROL_BULK_VISCOSITY.

An additional option **_TITLE** may be appended to ***HOURGLASS** keywords. If this option is used then an addition line is read for each section in 80a format which can be used to describe the section. At present LS-DYNA does make use of the title. Inclusion of titles gives greater clarity to input decks.

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	HGID	IHQ	QM	IBQ	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. This ID is referenced by HGID in the *PART command.

IHQ Hourglass control type. For solid elements six options are available. For quadrilateral shell and membrane elements the hourglass control is based on the formulation of Belytschko and Tsay, i.e., options 1-3 are identical, and options 4-6 are identical:

EQ.0:default=1 regardless of IHQ in *control_hourglass,

EQ.1:standard LS-DYNA viscous form,

EQ.2:Flanagan-Belytschko viscous form,

EQ.3: Flanagan-Belytschko viscous form with exact volume integration for solid elements,

EQ.4:Flanagan-Belytschko stiffness form,

EQ.5:Flanagan-Belytschko stiffness form with exact volume integration for solid elements.

*HOURGLASS

<u>VARIABLE</u>	<u>DESCRIPTION</u>
	<p>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.</p> <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 may cause instabilities. The recommended default applies to all options except for IHQ=6. The stiffness forms, however, can stiffen the response especially if deformations are large and therefore should be used with care. For the shell and membrane elements QM is taken as the membrane hourglass coefficient, the bending as QB, and warping as QW. These coefficients can be specified independently, but generally, QM=QB=QW, is adequate. For type 6 solid element hourglass control, see note 4 below.
IBQ	Bulk viscosity type (See remark 3. below.): EQ.1: standard LS-DYNA.
Q1	Quadratic bulk viscosity coefficient.
Q2	Linear bulk viscosity coefficient. QB Hourglass coefficient for shell bending. The default: QB=QM. See remark 4. 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. Type 6 hourglass control is for 2D and 3D solid elements only. Based on elastic constants and an assumed strain field, it produces accurate coarse mesh bending results for elastic material when QM=1.0. For plasticity models with a yield stress tangent modulus that is much smaller than the elastic modulus, a smaller value of QM (0.001 to 0.1) may produce

better results. For any material, keep in mind that the stiffness is based on the elastic constants, so if the material softens, a QM value smaller than 1.0 may work better. For anisotropic materials, an average of the elastic constants is used. For fluids modeled with null material, type 6 hourglass control is viscous and is scaled to the viscosity coefficient of the material (see *MAT_NULL).

5. In part, the computational efficiency of the Belytschko-Lin-Tsay and the under integrated Hughes-Liu shell elements are derived from their use of one-point quadrature in the plane of the element. To suppress the hourglass deformation modes that accompany one-point quadrature, hourglass viscous or stiffness based stresses are added to the physical stresses at the local element level. The discussion of the hourglass control that follows pertains to all one point quadrilateral shell and membrane elements in LS-DYNA.

The hourglass shape vector τ_I is defined as

$$\tau_I = h_I - (h_j \hat{x}_{aj}) B_{aj}$$

where, \hat{x}_{aj} are the element coordinates in the local system at the Ith element node, B_{aj} 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., $\hat{x}_{aj} = (\hat{x}_{1I}, \hat{x}_{2I}) = (\hat{x}_I, \hat{y}_I)$.

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

$$\dot{Q}_\alpha^M = \frac{QM \cdot EtA}{8} B_{\beta I} B_{\beta I} \dot{q}_\alpha^M$$

$$\dot{Q}_\alpha^B = \frac{QB \cdot Et^3 A}{192} B_{\beta I} B_{\beta I} \dot{q}_\alpha^B$$

*HOURLASS

$$\dot{Q}_3^W = \frac{QW \cdot \kappa G t^3 A}{12} B_{\beta 1} B_{\beta 1} \dot{q}_3^B$$

where t is the shell thickness. The hourglass coefficients: Q_M , Q_B , and Q_W are generally assigned values between 0.05 and 0.10.

Finally, the hourglass stresses which are updated using the time step, Δt , from the stress rates in the usual way, i.e.,

$$Q^{n+1} = Q^n + \Delta t \dot{Q}$$

and the hourglass resultant forces are then

$$\hat{f}_{\alpha I}^H = \tau_I Q_\alpha^M$$

$$\hat{m}_{\alpha I}^H = \tau_I Q_\alpha^B$$

$$\hat{f}_{3I}^H = \tau_I Q_3^W$$

where the superscript H emphasizes that these are internal force contributions from the hourglass deformations.

***INCLUDE**

Purpose: The keyword ***INCLUDE** provides a means of reading independent input files containing model data. The file contents are placed directly at the location of the ***INCLUDE** line.

***INCLUDE_{OPTION}**

Options include:

<BLANK>

NASTRAN

STAMPED_PART

TRANSFORM

The **STAMPED_PART** option allows the plastic strain and thickness distribution of the stamping simulation to be mapped onto a part in the crash model. Between the stamped part and the crash part, note the following points:

1. The outer boundaries of the parts do not need to match since only the regions of the crash part which overlap the stamped part are initialized.
2. Arbitrary mesh patterns are assumed.
3. Element formulations can change
4. Three nodes on each part are used to reorient the stamped part for the mapping of the strain and thickness distributions. After reorientation, the three nodes on each part should approximately coincide.
5. The number of in plane integrations points can change.
6. The number of through thickness integration points can change. Full interpolation is used.
7. The node and element ID's between the stamped part and the crash part do not need to be unique.

The **TRANSFORM** option allows for node, element, and set ID's to be offset and for coordinates and constitutive parameters to be transformed and scaled.

Card Format. The card is required.

Card 1

1

Variable	FILENAME
Type	C

*INCLUDE

If the NASTRAN option is active then define the following input line.

Card Format for the NASTRAN option

Card 2 1 2 3 4 5 6 7 8

Variable	BEAMDF	SHELLDF	SOLIDDF					
Type	I	I	I					
Default	2	21	18					

If the STAMPED_PART option is active then define the following input.

Card Format for the STAMPED_PART option

Card 2 1 2 3 4 5 6 7 8

Variable	PID	THICK	PSTRN	STRAIN	STRESS	INCOUT		
Type	I	I	I	I	I	I		
Default	none	0	0	0	0	0		

Card 3 1 2

Variable	N1S	N2S	N3S	N1C	N2C	N3C		
Type	I	I	I	I	I	I		
Default	0	0	0	0	0	0		
Remarks	2	2	2	2	2	2		

Optional Cards

Card 4 1

Variable	ISYM							
Type	I							

Card 5 1 2 3

Variable	X01	Y01	Z01					
Type	f	f	f					

Card 6 1 2 3 4 5 6

Variable	X02	Y02	Z02	X03	Y03	Z03		
Type	f	f	f	f	f	f		

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

Card 2 1 2 3 4 5 6 7 8

Variable	IDNOFF	IDEOFF	IDPOFF	IDMOFF	IDSOFF	IDFOFF	IDDOFF	
Type	I	I	I	I	I	I	I	

Card 3 1 2

Variable	IDROFF							
Type	I							

*INCLUDE

Card 4 1 2 3 4 5 6 7 8

Variable	FCTMAS	FCTTIM	FCTLEN	FCTTEM	INCOUT1			
Type	F	F	F	A	I			

Card 5

Variable	TRANID							
Type	I							
Default	0							

VARIABLE

DESCRIPTION

FILENAME	File name of file to be included in this keyword file, 80 characters maximum. If the STAMPED_PART option is active, this is the DYNAIN file containing the results from metal stamping.
BEAMDF	LS-DYNA beam element type. Defaults to type 2.
SHELLDF	LS-DYNA shell element type. Defaults to type 21.
SOLIDDF	LS-DYNA solid element type. Defaults to type 18.
PID	Part ID of crash part for remapping.
THICK	Thickness remap: EQ.0: map thickness EQ.1: do not map thickness
PSTRN	Plastic strain remap: EQ.0: map plastic strain EQ.1: do not plastic strain
STRAIN	Strain remap: EQ.0: map strains EQ.1: do not map strains
STRESS	Stress tensor remap: EQ.0: map stress tensor EQ.1: do not map stress tensor

INCOU	Set to 1 to save the mapped data to a file called dyna.inc, which contains the mapped data for the part that is being mapped. This option is useful to do mapping using INCLUDE_STAMPED_PART and then save the mapped data for future use.
N1S	First of 3 nodes needed to reorient the stamped part.
N2S	Second of 3 nodes needed to reorient the stamped part.
N3S	Third of 3 nodes needed to reorient the stamped part.
N1C	First of 3 nodes needed to reorient the crash model part.
N2C	Second of 3 nodes needed to reorient the crash model part.
N3C	Third of 3 nodes needed to reorient the crash model part.
ISYM	Symmetric switch EQ.0: no symmetric mapping EQ.1: yz plane symmetric mapping EQ.2: zx plane symmetric mapping EQ.3: zx and yz planes symmetric mapping EQ.4: user defined symmetric plane mapping
X01, Y01, Z01	First point in the symmetric plane (required if ISYM.NE.0)
X02, Y02, Z02	Second point in the symmetric plane
X03, Y03, Z03	Third point in the symmetric plane
IDNOFF	Offset to node ID.
IDEOFF	Offset to element ID.
IDPOFF	Offset to part ID, nodal rigid body ID, and constrained nodal set ID.
IDMOFF	Offset to material ID.
IDSOFF	Offset to set ID.
IDFOFF	Offset to function ID or table ID.
IDDOFF	Offset to any ID defined through DEFINE except the FUNCTION option.
IDROFF	Offset to section ID, hourglass ID, and any equation of state ID.
FCTMAS	Mass transformation factor. For example, FCTMAS=1000. when the original mass units are in tons and the new unit is kg.
FCTTIM	Time transformation factor. For example, FCTTIM=.001 when the original time units are in milliseconds and the new time unit is seconds.
FCTLEN	Length transformation factor.

*INCLUDE

<u>VARIABLE</u>	<u>DESCRIPTION</u>
FCTTEM	Temperature transformation factor consisting of a four character flag: FtoC (Fahrenheit to Centigrade), CtoF, FtoK, KtoF, KtoC, and CtoK.
INCOUT1	Set to 1 for the creation of a file, DYNA.INC, which contains the transformed data. The data in this file can be used in future include files and should be checked to ensure that all the data was transformed correctly.
TRANID	Transformation ID, if 0 no transformation will be applied. See the input DEFINE_TRANSFORM.

Remarks:

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

```
*TITLE
full car model
*INCLUDE
carfront.k
*INCLUDE
carback.k
*INCLUDE
occupantcompartment.k
*INCLUDE
dummy.k
*INCLUDE
bag.k
*CONTACT
...
*END
```

Note that the command *END terminates the include file.

The carfront.k file can again be subdivided into rightrail.k, leftrail.k, battery.k, 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
```

When symmetric mapping is used (ISYM is not zero), the three points should not be in one line.

If ISYM == 0, 1, 2, or 3, only the first point (X01, Y01, Z01) is needed

If ISYM==4, all the three points are needed

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

2. n1s,n2s,n3s,n1c,n2c,n3c are used for transforming the stamped part to the crashed part, such that it is in the same position as the crashed part. If the stamped part is in the same position as the crashed part then n1s,n2s,n3s,n4s,n1c,n2c,n3c can all be set to 0. Note: If these 6 nodes are input as 0, LS-DYNA will not transform the stamped part.

***INCLUDE**

***INITIAL**

The keyword ***INITIAL** provides a way of initializing velocities and detonation points. The keyword control cards in this section are defined in alphabetical order:

***INITIAL_CFD**

***INITIAL_DETONATION**

***INITIAL_FOAM_REFERENCE_GEOMETRY**

***INITIAL_GAS_MIXTURE**

***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**

***INITIAL_VELOCITY_RIGID_BODY**

and:

***INITIAL_VELOCITY_GENERATION**

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

***INITIAL_VOID_OPTION**

***INITIAL_VOLUME_FRACTION**

***INITIAL_VOLUME_FRACTION_GEOMETRY**

*INITIAL

*INITIAL_CFD

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

Card Format (1 of 3)

Card 1 1 2 3 4 5 6 7 8

Variable	U	V	W	T	H	RHO	Z1	Z2
Type	F	F	F	F	F	F	F	F
Default	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0

Card Format (2 of 3)

Card 2 1 2 3 4 5 6 7 8

Variable	Z3	Z4	Z5	Z6	Z7	Z8	Z9	Z10
Type	F	F	F	F	F	F	F	F
Default	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0

Card Format (3 of 3)

Card 3 1 2 3 4 5 6 7 8

Variable	K	EPS						
Type	F	F						
Default	0.0	0.0						

VARIABLE

DESCRIPTION

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

*INITIAL

*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						

<u>VARIABLE</u>	<u>DESCRIPTION</u>
PID	Part ID of high explosive material to be lit, see *PART. However, two other options are available: EQ,-1: an acoustic boundary, also *BOUNDARY_USA_SURFACE, EQ, 0: all high explosive materials are considered.
X	x-coordinate of detonation point, see Figure 16.1.
Y	y-coordinate of detonation point.
Z	z-coordinate of detonation point.

VARIABLE	DESCRIPTION
LT	Lighting time for detonation point. This time is ignored for an acoustic boundary.
PEAK	Peak pressure, p_o , of incident pressure pulse, see remark below.
DECAY	Decay constant, τ
XS	x-coordinate of standoff point, see Figure 16.1.
YS	y-coordinate of standoff point
ZS	z-coordinate of standoff point
NID	Reference node ID near structure

Remarks:

For solid elements (not acoustic) two options are available. If the control card option, *CONTROL_EXPLOSIVE_SHADOW, is not used the lighting time for an explosive element is computed using the distance from the center of the element to the nearest detonation point, L_d ; the detonation velocity, D ; and the lighting time for the detonator, t_d :

$$t_L = t_d + \frac{L_d}{D}$$

The detonation velocity for this default option is taken from the element whose lighting time is computed and does not account for the 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-PREPOST. The lighting times may be displayed at time=0., state 1, by plotting component 7 (a component normally reserved for plastic strain) for the explosive material. The lighting times are stored as negative numbers. The negative lighting time is replaced by the burn fraction when the element ignites.

Line detonations may be approximated by using a sufficient number of detonation points to define the line. Two many detonation points may result in significant initialization cost.

The pressure versus time curve for the acoustic option is defined by:

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

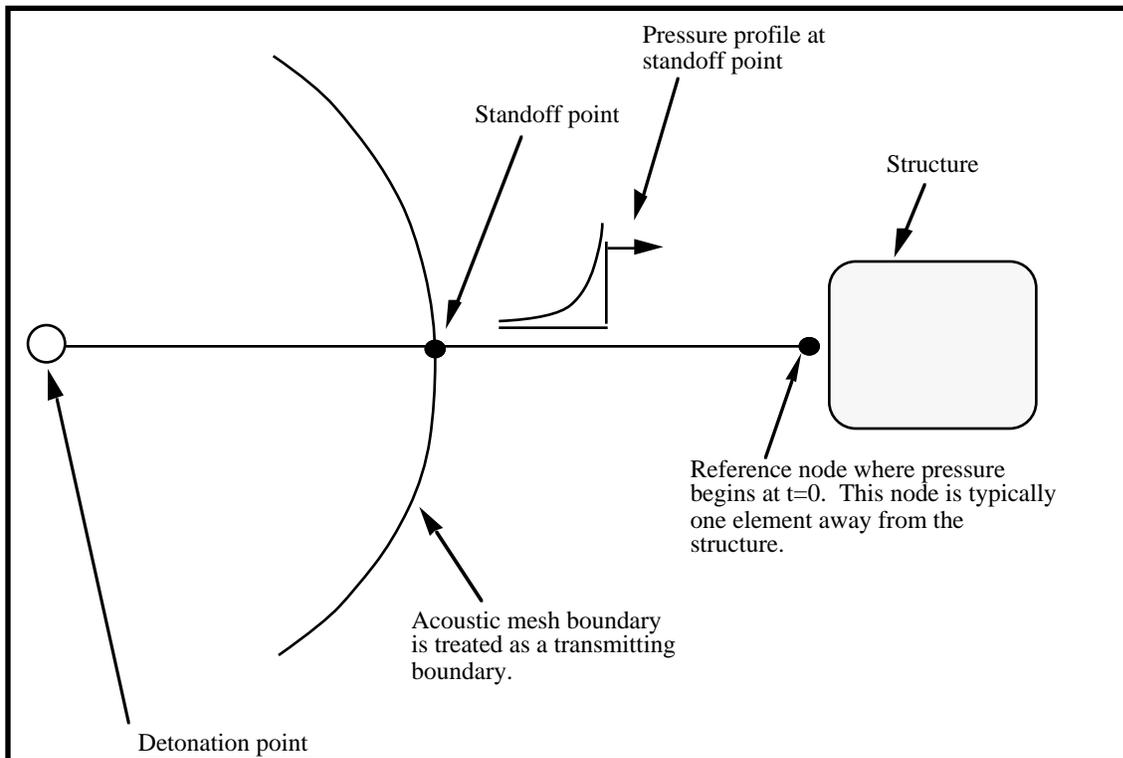


Figure 16.1 Initialization of the initial pressures due to an explosive disturbance is performed in the acoustic media. LS-DYNA automatically determines the acoustic mesh boundary and applies the pressure time history to the boundary. This option is only applicable to the acoustic element formulation,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

***INITIAL_FOAM_REFERENCE_GEOMETRY**

Purpose: The reference configuration allows stresses to be initialized in the material model, *MAT_LOW_DENSITY_FOAM. To use this option, the geometry of the foam material is defined in a deformed configuration. The stresses in the low density foam then depend only on the deformation gradient matrix:

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

where X_j is the undeformed configuration. By using this option, dynamic relaxation can be avoided once a deformed configuration is obtained usually on the first run of a particular problem.

Card Format (I8,3E16.0)

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

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

VARIABLE**DESCRIPTION**

NID	Node number
X	x coordinate
Y	y coordinate
Z	z coordinate

*INITIAL

*INITIAL_GAS_MIXTURE

Purpose: The command is used to initialize the temperature and the density of different gas species in *MAT_GAS_MIXTURE.

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	SID	STYPE	MMGID	TEMP				
Type	I	I	I	F				
Default	none	0	none	none				

Card 2 1 2 3 4 5 6 7 8

Variable	RO1	RO2	RO3	RO4	RO5	RO6	RO7	RO8
Type	F	F	F	F	F	F	F	F
Default	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0

VARIABLE	DESCRIPTION
-----------------	--------------------

SID Set ID for initialization

STYPE Set type:
 EQ.0: part set,
 EQ.1: part ID

MMGID Multi-material group ID of the material that is to be initialized

RO1-RO8 Initial densities for up to eight different gas species

***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

*INITIAL_STRAIN_SHELL

Purpose: Initialize strain tensor and inner and outer through thickness integration points at element center. This option is primarily for multi-stage metal forming operations where the accumulated strain is of interest.

Define as many shell elements in this section as desired. The input is assumed to terminate when a new keyword is detected. These strain tensors are defined at the inner and outer integration points and are used for post-processing only. There is no interpolation with this option and the strains are defined in the global cartesian coordinate system. The *DATABASE_EXTENT_BINARY flag STRFLG must be set to unity for this option to work.

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	EID							
Type	I							
Default	none							

Define two cards below, one for the inner integration point and the other for the outer integration point, respectively.

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

Variable	EPSxx	EPSyy	EPSzz	EPSxy	EPSyz	EPSzx		
Type	F	F	F	F	F	F		

VARIABLE

DESCRIPTION

EID

Element ID

EPSij

Define the ij strain component. The strains are defined in the GLOBAL cartesian system.

***INITIAL_STRESS_BEAM**

Purpose: Initialize stresses and plastic strains in the Hughes-Liu beam elements.

Define as many beams in this section as desired. The input is assumed to terminate when a new keyword is detected.

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	EID	RULE	NPTS	LOCAL				
Type	I	I	I	I				
Default	none	none	none	0				

Define NTPS cards below, one per integration point.

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

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

VARIABLE

DESCRIPTION

EID

Element ID

RULE

Integration rule type number:

EQ.1.0: 1 x 1 Gauss quadrature

EQ.2.0: 2 x 2 Gauss quadrature (default beam),

EQ.3.0: 3 x 3 Gauss quadrature,

EQ.4.0: 3 x 3 Lobatto quadrature,

EQ.5.0: 4 x 4 Gauss quadrature.

*INITIAL

<u>VARIABLE</u>	<u>DESCRIPTION</u>
NPTS	Number of integration points output.
LOCAL	Coordinate system for stresses: EQ.0: stress components are defined in the global coordinate system. EQ.1: stress components are defined in the local beam system. In th local system components SIG22, SIG33, and SIG23 are set to 0.0.
SIGIJ	Define the IJ stress component.
EPS	Effective plastic strain

***INITIAL_STRESS_SHELL**

Purpose: Initialize stresses and plastic strains for shell elements.

Define as many shell elements in this section as desired. The input is assumed to terminate when a new keyword is detected. It is not necessary for the location of the through thickness integration points to match those used in the elements which are initialized. The data will be interpolated by LS-DYNA.

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	EID	NPLANE	NTHICK	NHISV				
Type	I	I	I	I				
Default	none	none	none	0				

Define NPLANE X NTHICK cards below, one (if NHISV=0) per integration point. Include optional cards as necessary to define the NHISV history variable. For each through thickness point define NPLANE points. NPLANE should be either 1 or 4 corresponding to either 1 or 4 Gauss integration points. If four integration points are specified, they should be ordered such that their in plane parametric coordinates are at:

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

respectively.

Card Set 2., 1 2 3 4 5 6 7 8

Variable	T	SIGXX	SIGYY	SIGZZ	SIGXY	SIGYZ	SIGZX	EPS
Type	F	F	F	F	F	F	F	F

Optional

Variable	HISV1	HISV2	HISV3	HISV4	HISV5	HISV6	HISV7	HISV8
Type	F	F	F	F	F	F	F	F

*INITIAL

<u>VARIABLE</u>	<u>DESCRIPTION</u>
EID	Element ID
NPLANE	Number of in plane integration points being output.
NTHICK	Number of through thickness integration points.
NHISV	Number of number of additional history variables.
T	Parametric coordinate of through thickness integration point. between -1 and 1 inclusive.
SIGij	Define the ij stress component. The stresses are defined in the GLOBAL cartesian system.
EPS	Effective plastic strain
HISVn	Define the nth history variable. The stresses are defined in the GLOBAL cartesian system.

***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	SIGXX	SIGYY	SIGZZ	SIGXY	SIGYZ	SIGZX	EPS	
Type	F	F	F	F	F	F	F	

*INITIAL

<u>VARIABLE</u>	<u>DESCRIPTION</u>
EID	Element ID
NINT	Number of integration points either 1 or 8.
SIGij	Define the IJ stress component. Stresses are defined in the GLOBAL cartesian system.
EPS	Effective plastic strain

***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

*INITIAL_VEHICLE_KINEMATICS

Purpose: Define initial kinematical information for a vehicle. In its initial orientation, the vehicle's yaw, pitch, and roll axes must be aligned with the global axes. Successive simple rotations are taken about these body fixed axes.

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	GRAV	PSID	XO	YO	ZO	XF	YF	ZF
Type	I	I	F	F	F	F	F	F
Default	none	none	0.	0.	0.	0.	0.	0.

Card 2 1 2 3 4 5 6 7 8

Variable	VX	VY	VZ	AAXIS	BAXIS	CAXIS		
Type	F	F	F	I	I	I		
Default	0.	0.	0.	0	0	0		

Card 3 1 2 3 4 5 6 7 8

Variable	AANG	BANG	CANG	WA	WB	WC		
Type	F	F	F	F	F	F		
Default	0.	0.	0.	0.	0.	0.		

VARIABLE	DESCRIPTION
GRAV	Gravity direction code. EQ.1. Global +x direction. EQ.-1. Global -x direction. EQ.2. Global +y direction. EQ.-2. Global -y direction. EQ.3 Global +z direction. EQ.-3 Global -z direction. Note: this must be the same for all vehicles present in the model.
PSID	Part set ID.
XO	x-coordinate of initial position of mass center.
YO	y-coordinate of initial position of mass center.
ZO	z-coordinate of initial position of mass center.
XF	x-coordinate of final position of mass center.
YF	y-coordinate of final position of mass center.
ZF	z-coordinate of final position of mass center.
VX	x-component of mass center velocity.
VY	y-component of mass center velocity.
VZ	z-component of mass center velocity.
AAXIS	First rotation axis code. EQ.1. Initially aligned with global x-axis. EQ.2. Initially aligned with global y-axis. EQ.3. Initially aligned with global z-axis.
BAXIS	Second rotation axis code.
CAXIS	Third rotation axis code.
AANG	Rotation angle about the first rotation axis (degrees).
BANG	Rotation angle about the second rotation axis (degrees).
CANG	Rotation angle about the third rotation axis (degrees).
WA	Angular velocity component for the first axis (radian/second).
WB	Angular velocity component for the second axis (radian/second).
WC	Angular velocity component for the third axis (radian/second).

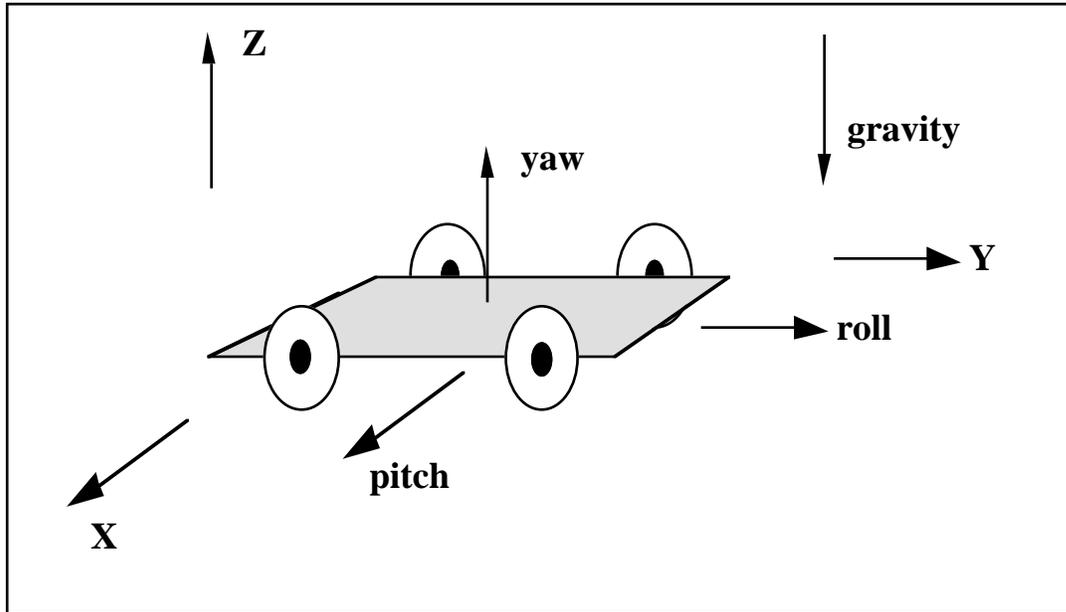


Figure 16.1. The vehicle pictured is to be oriented with a successive rotation sequence about the yaw, pitch, and roll axes, respectively. Accordingly, AAXIS=3, BAXIS=1, and CAXIS=2. The direction of gravity is given by GRAV=-3.

***INITIAL_VELOCITY**

Purpose: Define initial nodal point translational velocities using nodal set ID's. This may also be used for sets in which some nodes have other velocities. See NSIDEX below.

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	NSID	NSIDEX	BOXID	IRIGID				
Type	I	I	I	I				
Default	none	0	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.		

*INITIAL

<u>VARIABLE</u>	<u>DESCRIPTION</u>
NSID	Nodal set ID, see *SET_NODES, containing nodes for initial velocity: If NSID = 0 the initial velocity is applied to all nodes.
NSIDEX	Nodal set I, see *SET_NODES, containing nodes that are exempted from the imposed velocities and may have other initial velocities.
BOXID	All nodes in box which belong to NSID are initialized. Nodes outside the box are not initialized. Exempted nodes are initialized to velocities defined by VXE, VYE, and VZE below regardless of their location relative to the box.
IRIGID	Option to overwrite rigid body velocities defined on *PART_INERTIA and *CONSTRAINED_NODAL_RIGID_BODY_INERTIA cards. GE.1: part set ID, containing ID of parts to overwrite. Centre of gravity of part must lie within box BOXID. If BOXID is not defined then all parts defined in the set are overwritten. EQ.-1: Overwrite velocities for all *PART_INERTIA's and *CONSTRAINED_NODAL_RIGID_BODY_INERTIA 's with a centre of gravity within box BOXID. If BOXID is not defined then all are overwritten. EQ.-2: Overwrite velocities for all *PART_INERTIA's and *CONSTRAINED_NODAL_RIGID_BODY_INERTIA's.
VX	Initial velocity in x-direction
VY	Initial velocity in y-direction
VZ	Initial velocity in z-direction
VXR	Initial rotational velocity about the x-axis
VYR	Initial rotational velocity about the y-axis
VZR	Initial rotational velocity about the z-axis
VXE	Initial velocity in x-direction of exempted nodes
VYE	Initial velocity in y-direction of exempted nodes
VZE	Initial velocity in z-direction of exempted nodes
VXRE	Initial rotational velocity in x-direction of exempted nodes
VYRE	Initial rotational velocity in y-direction of exempted nodes
VZRE	Initial rotational velocity in z-direction of exempted nodes

Remarks:

1. This generation input must not be used with *INITIAL_VELOCITY_GENERATION keyword.
2. If a node is initialized on more than one input card set, then the last set input will determine its velocity. However, if the nodal velocity is also specified on a *INITIAL_VELOCITY_NODE card, then the velocity specification on this card will be used.
3. Unless the option IRIGID is specified rigid bodies, initial velocities given in *PART_INERTIA will overwrite generated initial velocities. The IRIGID option will cause the rigid body velocities specified on the *PART_INERTIA input to be overwritten. To directly specify the motion of a rigid body without using the keyword, *PART_INERTIA, which also requires the definition of the mass properties, use the keyword option, *INITIAL_VELOCITY_RIGID_BODY.
4. Nodes which belong to rigid bodies must have motion consistent with the translational and rotational velocity of the center of gravity (c.g.) of the rigid body. During initialization the rigid body translational and rotational rigid body momentum's are computed based on the prescribed nodal velocity field. From this rigid body momentum, the translational and rotational velocities of the nodal points are computed and reset to the new values. These new values may or may not be the same as the values prescribed for the nodes that make up the rigid body. Sometimes this occurs in single precision due to numerical round-off. If a problem like this occurs specify the velocity using the keyword: *INITIAL_VELOCITY_RIGID_BODY.

***INITIAL**

***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_RIGID_BODY**

Purpose: Define the initial translational and rotational velocities at the center of gravity (c.g.) for a rigid body or a nodal rigid body. This input overrides all other velocity input for the rigid body and the nodes which define the rigid body.

Card 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 the rigid body or the nodal rigid body.
VX	Initial translational velocity at the c.g. in global x-direction.
VY	Initial translational velocity at the c.g. in global y-direction.
VZ	Initial translational velocity at the c.g. in global z-direction.
VXR	Initial rotational velocity at the c.g. about the global x-axis.
VYR	Initial rotational velocity at the c.g. about the global y-axis.
VZR	Initial rotational velocity at the c.g. about the global z-axis.

See remarks 3 and 4 on the *INITIAL_VELOCITY input description.

*INITIAL

*INITIAL_VELOCITY_GENERATION

Purpose: Define initial velocities for rotating and translating bodies. Caution: Rigid body velocities cannot be reinitialized after dynamic relaxation by setting PHASE=1 since rigid body velocities are always restored to the values that existed prior to dynamic relaxation. Reinitialization of velocities after dynamic relaxation is only for nodal points of deformable bodies.

Card Format

Card 1 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. WARNING: If a part ID of a rigid body is specified only the nodes that belong to elements of the rigid body are initialized. Nodes defined under the keyword *CONSTRAINED_EXTRA_NODES are not initialized.

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.

<u>VARIABLE</u>	<u>DESCRIPTION</u>
VZ	Initial translational velocity in global z-direction. .
XC	x-coordinate on rotational axis.
YC	y-coordinate on rotational axis.
ZC	z-coordinate on rotational axis.
NX	x-direction cosine.
NY	y-direction cosine.
NZ	z-direction cosine.
PHASE	Flag specifying phase of the analysis the velocities apply to: EQ.0. Velocities applied immediately, EQ.1. Velocities applied after dynamic relaxation.

Remarks:

1. This generation input must not be used with *INITIAL_VELOCITY or *INITIAL_VELOCITY_NODE options.
2. The velocities are initialized in the order the *INITIAL_VELOCITY_GENERATION input is defined. Later input via the *INITIAL_VELOCITY_GENERATION keyword may overwrite the velocities previously set.
3. For rigid bodies, initial velocities given in *PART_INERTIA will overwrite generated initial velocities.
4. Nodes which belong to rigid bodies must have motion consistent with the translational and rotational velocity of the rigid body. During initialization the rigid body translational and rotational rigid body momentum's are computed based on the prescribed nodal velocities. From this rigid body motion the velocities of the nodal points are computed and reset to the new values. These new values may or may not be the same as the values prescribed for the node.

***INITIAL**

***INITIAL_VOID_OPTION**

Available options are:

PART

SET

Purpose: Define initial voided part set ID's or part numbers. Void materials cannot be created during the calculation. Fluid elements which are evacuated, e.g., by a projectile moving through the fluid, during the calculation are approximated as fluid elements with very low densities. The constitutive properties of fluid materials used as voids must be identical to those of the materials which will fill the voided elements during the calculation. Mixing of two fluids with different properties is not permitted with this option.

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	PSID/PID								
Type	I								
Default	none								
Remark	1								

VARIABLE

DESCRIPTION

PSID/PID

Part set ID or part ID, see also *SET_PART:

Remark:

This void option and multiple materials per element, see *ALE_MULTI-MATERIAL_GROUP are incompatible and cannot be used together in the same run.

***INITIAL_VOLUME_FRACTION**

Purpose: Define initial volume fraction of different materials in multi-material ALE, or in single material and void, models.

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	EID	VF1	VF2	VF3	VF4	VF5	VF6	VF7
Type	I	F	F	F	F	F	F	F
Default	none	0.0	0.0	0.0	0.0	0.0	0.0	0.0

VARIABLE

DESCRIPTION

EID	Element ID.
VF1	Volume fraction of multi-material group 1, AMMGID=1.
VF2	Volume fraction of multi-material group 2. Only needed in simulations with 3 material groups. Otherwise VF2=1-VF1.
VF3	Volume fraction of multi-material group 3, AMMGID=3.
VF4	Volume fraction of multi-material group 4, AMMGID=4.
VF5	Volume fraction of multi-material group 5, AMMGID=5.
VF6	Volume fraction of multi-material group 6, AMMGID=6.
VF7	Volume fraction of multi-material group 7, AMMGID=7.

*INITIAL

*INITIAL_VOLUME_FRACTION_GEOMETRY

Purpose: Define initial volume fraction of different materials in multi-material ALE, or in single material and void, models,

Card1 Format

Card 1 1 2 3 4 5 6 7 8

Variable	SID_ALE	ST_ALE	GROUP					
Type	I	I	I					
Default	none	0.0	0.0					

VARIABLE

DESCRIPTION

SID_ALE	Part set ID or part ID, see also *SET_PART:Set ID, of the fluid elements to be processed
ST_ALE	Set Type: EQ.0: part set EQ.1: part.
GROUP	Primitive Group Id

Card2 Format

Card 1 1 2 3 4 5 6 7 8

Variable	GEOTYPE	IN-OUT	GR_FILL					
Type	I	I	I					
Default	none	0	none					

*INITIAL

VARIABLE	DESCRIPTION
----------	-------------

SID	Part or set part Id of the elements shell defining the geometry to be partially or completely filled, see *PART and *SET_PART
-----	---

SSTYPE	EQ.0: Part Set EQ.1: Part
--------	------------------------------

IOPT	EQ.0: if the normal vectors are defined toward the inside tank EQ.1: if the normal vectors are defined toward the outside tank
------	---

Card3 Format if GEOTYPE=2 (Set_Segment)

Card 1 1 2 3 4 5 6 7 8

Variable	SID	IOPT						
Type	I	I						
Default	none	0						

VARIABLE	DESCRIPTION
----------	-------------

SID	Segment Set ID, see *SET_SEGMENT
-----	----------------------------------

IOPT	EQ.0: if the normal vectors to the segments are defined toward the inside geometry EQ.1: if the normal vectors to the segments are defined toward the outside geometry
------	---

Card3 Format if GEOTYPE=3 (Plane)

Card 1 1 2 3 4 5 6 7 8

Variable	X 0	Y0	Z0	X1	Y1	Z1		
Type	F	F	F	F	F	F		
Default	none	none	none	none	none	none		

<u>VARIABLE</u>	<u>DESCRIPTION</u>
X0	X-coordinate of a spatial point on the plane
Y0	Y-coordinate of a spatial point on the plane
Z0	Z-coordinate of a spatial point on the plane
X1	X-coordinate of a normal vector to the plane
Y1	Y-coordinate of a normal vector to the plane
Z1	Z-coordinate of a normal vector to the plane

Card3 Format if GEOTYPE=4 (Cylinder and Cone)

Card 1 1 2 3 4 5 6 7 8

Variable	X 0	Y0	Z0	X1	Y1	Z1	R1	R2
Type	F	F	F	F	F	F	F	F
Default	none							

<u>VARIABLE</u>	<u>DESCRIPTION</u>
X0	X-coordinate of the center of the lower base of the cone
Y0	Y-coordinate of the center of the lower base of the cone
Z0	Z-coordinate of the center of the lower base of the cone
X1	X-coordinate of the center of the upper base of the cone
Y1	Y-coordinate of the center of the upper base of the cone
Z1	Z-coordinate of the center of the upper base of the cone
R1	Radius of the lower base of the cone
R2	Radius of the upper base of the cone

*INITIAL

Card3 Format if GEOTYPE=5 (Rectangular Box)

Card 1 1 2 3 4 5 6 7 8

Variable	X 0	Y0	Z0	X1	Y1	Z1		
Type	F	F	F	F	F	F		
Default	none	none	none	none	none	none		

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

X0	X_max coordinate of the box
Y0	Y_max coordinate of the box
Z0	Z_max coordinate of the box
X1	X_min coordinate of the box
Y1	Y_min coordinate of the box
Z1	Z_min coordinate of the box

Card3 Format if GEOTYPE=6 (Sphere)

Card 1 1 2 3 4 5 6 7 8

Variable	X 0	Y0	Z0	R0				
Type	F	F	F	F				
Default	none	none	none	none				

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

X0	X-coordinate of the center of the sphere
Y0	Y-coordinate of the center of the sphere
Z0	Z-coordinate of the center of the sphere
R0	Radius of the sphere

In case a new geometries are defined the following cards are needed

Card 4 and Card 5

should be defined as Card 2 and Card 3.

Card 6 and Card 7

should be defined as Card 2 and Card 3.

Example 1

2 material groups.

A tank defined by shell elements *ELEMENT_SHELL is intersecting a block of ALE mesh. The goal is fill the inside tank with fluid material, the outside with air and the intersected element with a partial volume.

In this example we have 2 material groups, water and air. Water is defined by group 1 and air by group 2.

These 2 groups are defined by the following card:

*ALE_MULTI-MATERIAL_GROUP

1,1

2,1

Part 1 for water and Part 2 for air, first line is the first group, second line is the second group. The shell elements are defined by Part 3, and the normal vectors are oriented toward the tank. The lsdyna input file should contain the following cards.

***KEYWORD**

***PART**

water

1,1,1,1

***PART**

Air

2,2,2,2

***PART**

Tank

***INITIAL**

3,3,3

***ALE_MULTI-MATERIAL_GROUP**

1.1

2.1

***ELEMENT_SOLID**

1,1,,,,,,,,

2,1,,,,,

.....

***ELEMENT_SHELL**

1,3,,,,,

***INITIAL_VOLUME_FRACTION_GEOMETRY**

1,1,2

1,0,1

3,1,0

***END**

Explanations of *INITIAL_VOLUME_FRACTION_GEOMETRY cards

CARD 1

All ALE elements of Part 1 in *ELEMENT_SOLID will be processed, These elements are initially filled with air, group 2.

SID_ALE=1, SSTYPE_ALE=1, GROUP=2

CARD 2

1,0,1

ALE elements inside the tank (normal vectors are oriented inward) will be filled with material group 1, water.

GETYPE=1, SID=3, SSTYPE=1, IOPT=0

CARD 3

3,1,0

The shell elements are defined by Part 3 , the normal vectors to the segments are inward.

SID=3, SSTYPE=1, IOPT=0

Example 2

3 material groups.

Let us consider 3 material groups problem.

Group 1 : air

Group 2: water

Group 3: gas

The aim is to fill the tank with water up to a level defined by a plane (x0, y0, z0, nx,ny,nz), and gas for the remaining volume of the tank. The outside volume of the tank is filled with air. The normal vectors of the shell elements defining the tank are oriented inward.

***KEYWORD**

***PART**

water

1,1,1,1

***PART**

air

2,2,2,2

***PART**

tank

3,3,3

***PART**

gas

***INITIAL**

4,4,4,4

***ALE_MULTI-MATERIAL_GROUP**

1,1

2,1

4,1

***ELEMENT_SOLID**

1,1,,,,

2,1,,,,,

..

***ELEMENT_SHELL**

1,3,,,,

2,3,,,,

..

***INITIAL_VOLUME_FRACTION_GEOMETRY**

1,1,2

1,0,1

3,1

3,0,3

420.,270.,190.,0.,0.,1.

1,1,2

3,1

***END**

Explanations of *INITIAL_VOLUME_FRACTION_GEOMETRY cards

CARD 1

1,1,2

ALE elements of Part 1, initially filled with material group 2, air, will be processed.

SID_ALE=1, SSTYPE_ALE=1, GROUP=2

CARD 2

1,0,1

ALE elements inside the tank are filled with material group 1, water, (normal vectors are oriented inward)

GEOTYPE=1, IN_OPT=0, GR_FILL=1

CARD 3

3,1,0

The tank is defined by Part 3, and the normal vectors are oriented inward.

SID=3, SSTYPE=1, IOPT=0

CARD 4

3,0,3

The ALE elements outside the plane defined below, (upper volume) are filled with material group 3, gas.

GOEMTYPE=3, IN_OPT=0, FILL_GR=3

CARD 5

420.,270.,190.,0.,0.,1.

The plane is defined by a spatial point coordinates (420.,270.,190.) and a normal vector to the plane, (0.,0.,1.)

GEOMTYPE=1, IN_OPT=1, FILL_GR=2

CARD 7

3,1,0

The tank is defined by Part 3, and the normal vectors are oriented inward

SID=3,SSTYPE=1,IOPT=0

Example 3

Explosion in air, using a spherical charge in a rectangular mesh.

In this example, we have 2 material groups, air and HE, air is group 1, and HE group 2. We insert a spherical volume of HE inside the rectangular air mesh.

***KEYWORD**

***INITIAL**

***ELEMENT_SOLID**

1,1,,,

2,1,,,

***ALE_MULTI-MATERIAL_GROUP**

1,1

2,1

***INITIAL_VOLUME_FRACTION_GEOMETRY**

1,0,

6,0,2

0.,0.,0.,5.5

***END**

Explanations of *INITIAL_VOLUME_FRACTION_GEOMETRY cards

CARD 1

1,1,1

Element of Part 1 , initially filled with air, will be processed.

SID_ALE=1, SSTYPE_ALE=1, FILL_GR=2

CARD 2

6,0.2

The inside volume of the sphere will be filled with material group 2, HE.

GEOMTYPE=1, IN_OPT=1, FILL_GR=2

CARD 3

0.,0.,0.,5.5

The sphere is defined by a center (0.,0.,0.) and radius R=5.5

***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.

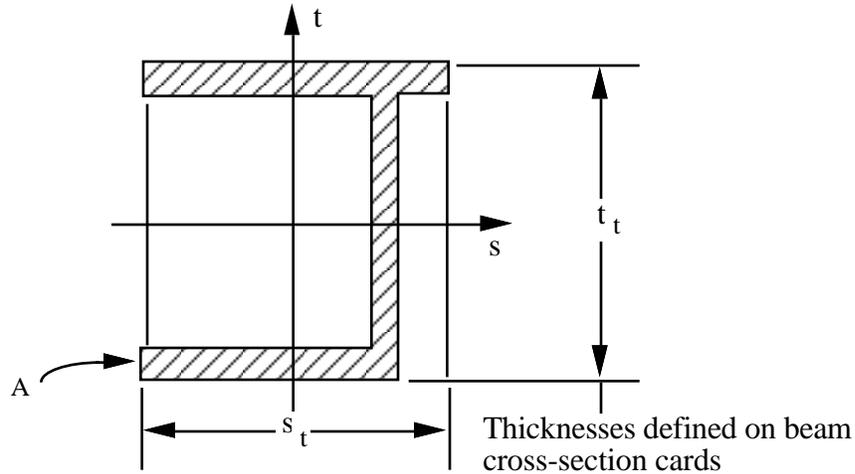
	1	2	3	4	5	6	7	8
Variable	W	TF	D	TW	SREF	TREF		
Type	F	F	F	F	F	F		
Default	none	none	none	none	1.0	0.0		

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 _f , flange thickness
D	d, depth
TW	t _w , web thickness
SREF	s _{ref} , location of reference surface normal to s, for the Hughes-Liu beam only. This option is only useful if the beam is connected to a shell or another beam on its outer surface, see also *SECTION_BEAM.
TREF	t _{ref} , location of reference surface normal to t, for the Hughes-Liu beam only. This option is only useful if the beam is connected to a shell or another beam on its outer surface, see also *SECTION_BEAM.
S	Normalized s coordinate of integration point, $-1 \leq s \leq 1$.
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.

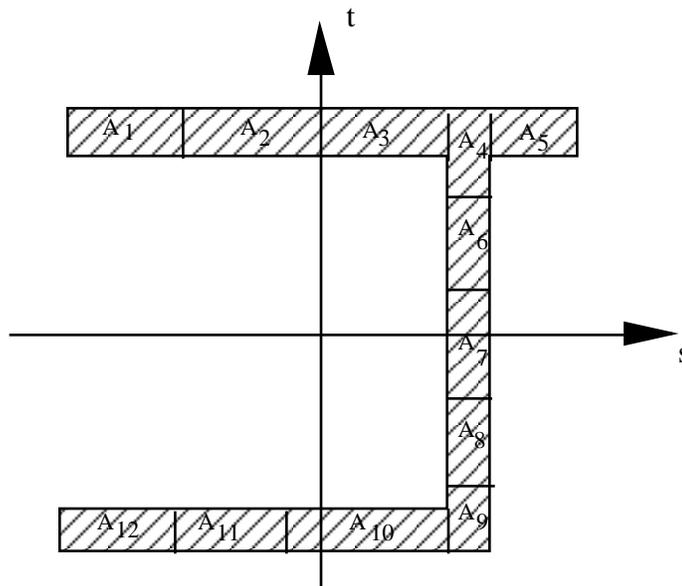


Figure 17.2. Definition of integration points for user defined integration rule.

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

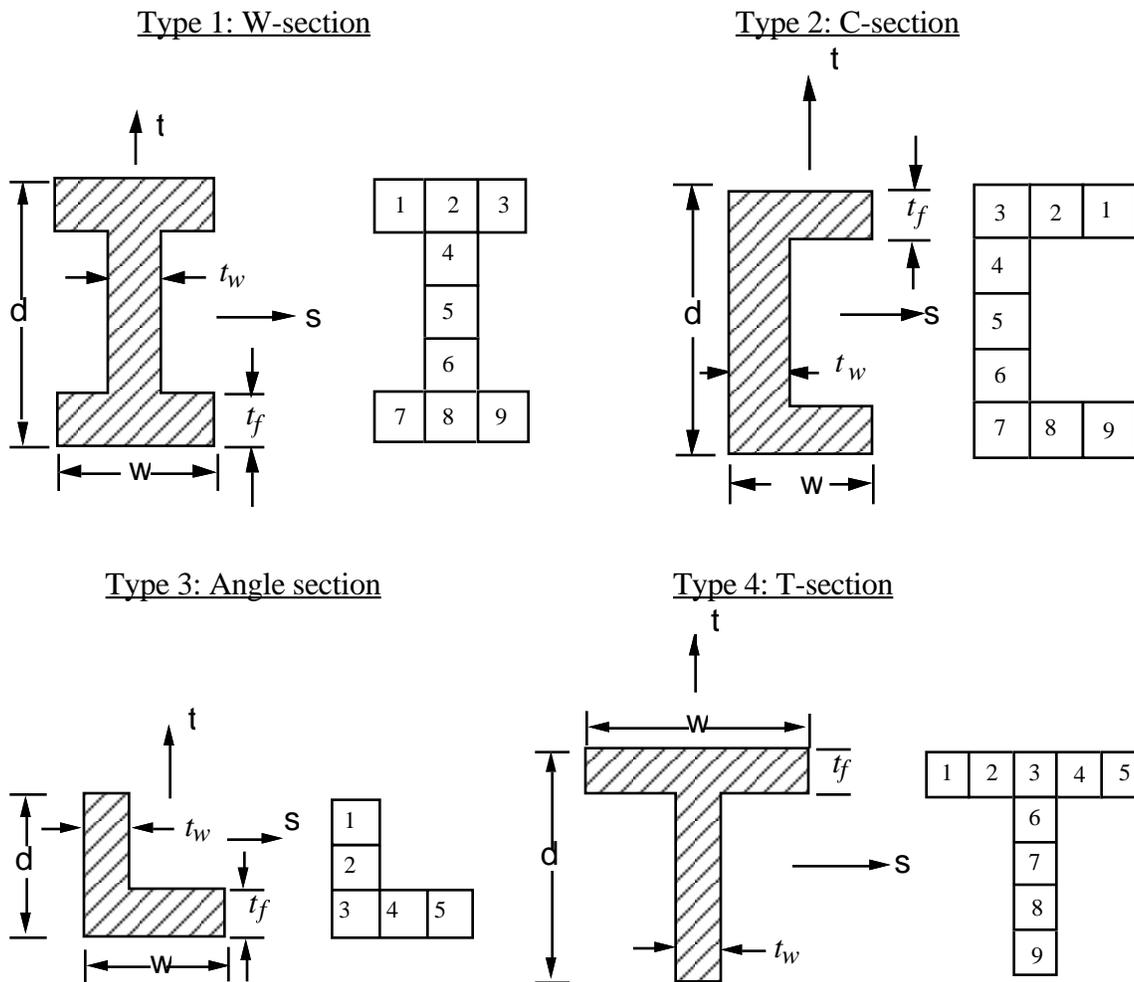


Figure 17.3a. Standard beam cross sections.

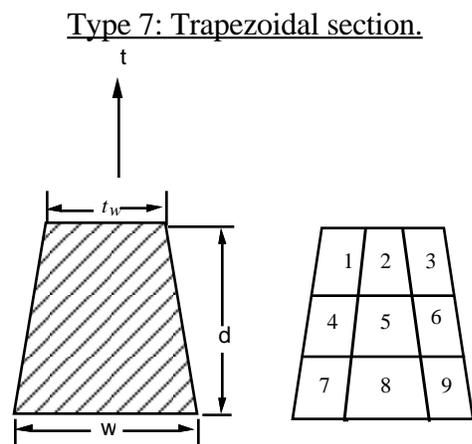
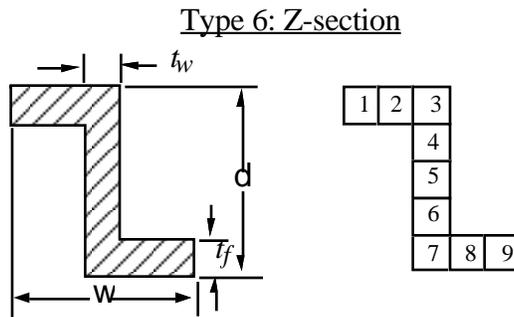
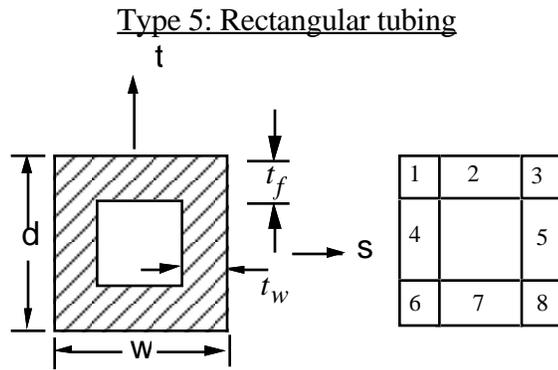


Figure 17.3b. Standard beam cross sections.

*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 PID specified on the element card. The material type is not allowed to change, see *PART. The average mass density for the shell element is based on a weighted average of the density of each layer that is used through the thickness. When modifying the constitutive constants through the thickness, it is often necessary to defined unique part IDs without elements that are referenced only by the user integration rule. These additional part IDs only provide a density and constitutive constants with local material axes (if used) and orientation angles taken from the PID referenced on the element card. In defining a PID for an integration point, it is okay to reference a solid element PID.

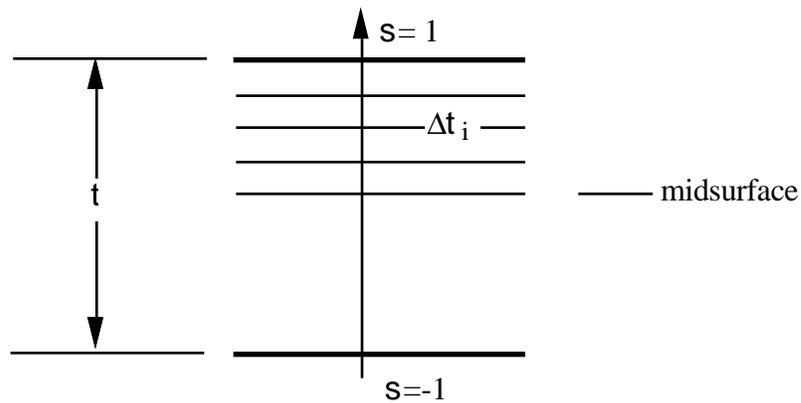


Figure 17.4. In the user defined shell integration rule the ordering of the integration points is arbitrary.

***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							

VARIABLE

DESCRIPTION

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						

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

<u>VARIABLE</u>	<u>DESCRIPTION</u>
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:

LSDYNA

NIKE3D

NASTRAN

SEAMLESS

and for *OPTION2*:

THICKNESS

NOTHICKNESS

See the remarks below.

Purpose: Define a material subset for an implicit springback calculation in LS-DYNA (LS-NIKE3D can be used but this option is not recommended) and any nodal constraints to eliminate rigid body degrees-of-freedom.

Card Format

1 2 3 4 5 6 7 8

Variable	PSID	NSHV						
Type	I	I						

VARIABLE

DESCRIPTION

PSID

Part set ID for springback, see *SET_PART.

NSHV

Number of additional shell history variables to be initialized. The shell stresses and plastic strains are written to the interface file. If NSHV is nonzero, the shell formulations and constitutive models should not change between runs.

*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					

<u>VARIABLE</u>	<u>DESCRIPTION</u>
NID	Node ID, see *NODE.
TC	Tranlational Constraint: EQ.0: no constraints, EQ.1: constrained x displacement, EQ.2: constrained y displacement, EQ.3: constrained z displacement, EQ.4: constrained x and y displacements, EQ.5: constrained y and z displacements, EQ.6: constrained z and x displacements. EQ.7: constrained x, y, and z displacements.
RC	Rotational constraint: EQ.0: no constraints, EQ.1: constrained x rotation, EQ.2: constrained y rotation, EQ.3: constrained z rotation, EQ.4: constrained x and y rotations, EQ.5: constrained y and z rotations, EQ.6: constrained z and x rotations, EQ.7: constrained x, y, and z rotations.

***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 LS-DYNA and NASTRAN in which case the shell element thickness is not an output. The file name for LS-DYNA is “dynain” and for NASTRAN is “nastin.” The *CONTROL_ADAPTIVITY is available for LS-DYNA.
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_LSDYNA.
 - (2) Prepare a new input deck including the file “dynain.”
 - (3) Add the keyword *ELEMENT_TRIM to this new deck.
 - (4) Add the keyword *DEFINE_CURVE_TRIM to this new deck.
 - (5) Run this new input deck with i=input_file_name. The adaptive constraints are eliminated by remeshing and the trimming is performed.
 - (6) In case this new trimmed mesh is needed, run a zero termination time job and output the file generated via the keyword, *INTERFACE_SPRINGBACK_LSDYNA.

Remarks for Seamless Springback:

In seamless springback 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_LSDYNA to generate a “dynain” file after forming, and then perform a second simulation running LS-DYNA in fully implicit mode for springback. See Appendix 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_BLAST**
- *LOAD_BODY_OPTION**
- *LOAD_BODY_GENERALIZED**
- *LOAD_BRODE**
- *LOAD_DENSITY_DEPTH**
- *LOAD_HEAT_GENERATION_OPTION**
- *LOAD_MASK**
- *LOAD_NODE_OPTION**
- *LOAD_RIGID_BODY**
- *LOAD_SEGMENT**
- *LOAD_SEGMENT_SET**
- *LOAD_SHELL_OPTION**
- *LOAD_SSA**
- *LOAD_SUPERPLASTIC_FORMING**
- *LOAD_THERMAL_OPTION**

*LOAD

*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								

VARIABLE

DESCRIPTION

EID/ESID	Beam ID (EID) or beam set ID (ESID), see *ELEMENT_BEAM or *SET_BEAM.
DAL	Direction of applied load: EQ.1: along r-axis of beam, EQ.2: along s-axis of beam, EQ.3: along t-axis of beam.
LCID	Load curve ID, see *DEFINE_CURVE.
SF	Load curve scale factor. This is for a simple modification of the function values of the load curve.

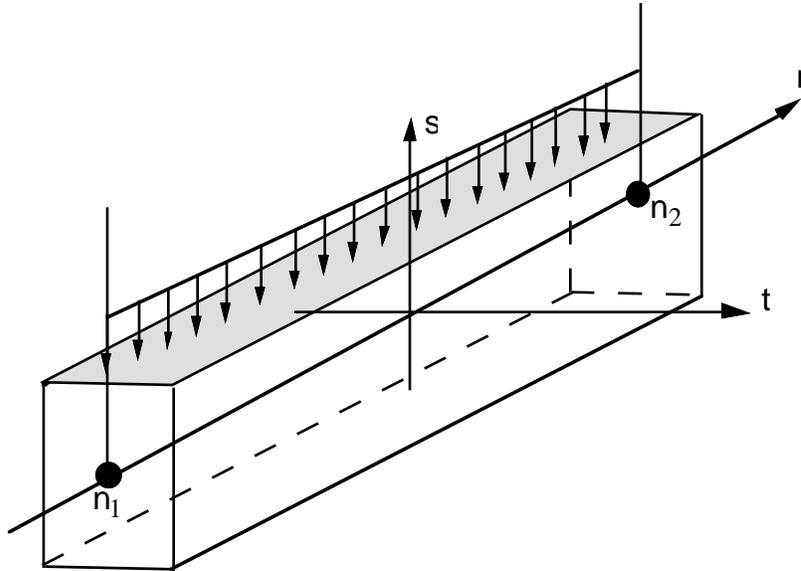


Figure 19.1. Applied traction loads are given in force per unit length. The s and t directions are defined on the *ELEMENT_BEAM keyword.

*LOAD

*LOAD_BLAST

Purpose: Define an airblast function for the application of pressure loads due to explosives in conventional weapons. The implementation is based on a report by Randers-Pehrson and Bannister [1997] where it is mentioned that this model is adequate for use in engineering studies of vehicle responses due to the blast from land mines. This option determines the pressure values when used in conjunction with the keywords: *LOAD_SEGMENT, *LOAD_SEGMENT_SET, or *LOAD_SHELL.

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	WGT	XBO	YBO	ZBO	TBO	IUNIT	ISURF	
Type	F	F	F	F	F	I	I	
Default	none	0.0	0.0	0.0	0.0	2	2	

Card 2 1 2 3 4 5 6 7 8

Variable	CFM	CFL	CFT	CFP				
Type	F	F	F	F				
Default	0.0	0.0	0.0	0.0				

VARIABLE

DESCRIPTION

WGT	Equivalent mass of TNT.
XBO	x-coordinate of point of explosion.
YBO	y-coordinate of point of explosion.
ZBO	z-coordinate of point of explosion.
TBO	Time-zero of explosion.

<u>VARIABLE</u>	<u>DESCRIPTION</u>
IUNIT	Unit conversion flag. EQ.1: feet, pounds, seconds, psi EQ.2: meters, kilograms, seconds, Pascals (default) EQ.3: inch, dozens of slugs, seconds, psi EQ.4: centimeters, grams, microseconds, Megabars EQ.5: user conversions will be supplied (see Card 2)
ISURF	Type of burst. EQ.1: surface burst - hemispherical charge situated on the surface EQ.2: air burst - spherical charge at least one charge diameter away from the surface (default)
CFM	Conversion factor - pounds per LS-DYNA mass unit.
CFL	Conversion factor - feet per LS-DYNA length units.
CFT	Conversion factor - milliseconds per LS-DYNA time unit.
CFP	Conversion factor - psi per LS-DYNA pressure unit.

Remarks:

1. A minimum of two load curves, even if unreferenced, must be present in the model.

*LOAD

*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							

<u>VARIABLE</u>	<u>DESCRIPTION</u>
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 ρ is the mass density, ω is the angular velocity vector, and r is a position vector from the origin to point P. Although the angular velocity may vary with time, the effects of angular acceleration are not included.

5. Angular velocities are useful for studying transient deformation of spinning three-dimensional objects. Typical applications have included stress initialization during dynamic relaxation where the initial rotational velocities are assigned at the completion of the initialization, and this option ceases to be active.

*LOAD

*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 ρ is the mass density, ω is the angular velocity vector, and r is a position vector from the origin to point P. Although the angular velocity may vary with time, the effects of angular acceleration are included.

5. Angular velocities are useful for studying transient deformation of spinning three-dimensional objects. Typical applications have included stress initialization during dynamic relaxation where the initial rotational velocities are assigned at the completion of the initialization, and this option ceases to be active.

*LOAD

*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				

<u>VARIABLE</u>	<u>DESCRIPTION</u>
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:

- 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					

<u>VARIABLE</u>	<u>DESCRIPTION</u>
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, PID, are considered. If no ID is given all elements of part ID, PID, are included. When the active list of elements are updated, elements outside the box will no longer have pressure applied, i.e., the current configuration is always used.
LCIDM	Curve ID defining the mask. This curve gives (x,y) pairs of points in a local coordinate system defined by the vector ID, VID2. Generally, the curve should form a closed loop, i.e., the first point is identical to the last point, and the curve should be flagged as a DATTYP=1 curve in the *DEFINE_CURVE section. If no curve ID is given, all elements of part ID, PID, are included with the exception of those deleted by the box. The mask works like the trimming option, i.e., see DEFINE_CURVE_TRIM and Figure 10.5.
VID2	Vector ID used to project the masking curve onto the surface of part ID, PID. The origin of this vector determines the origin of the local system that the coordinates of the PID are transformed into prior to determining the pressure distribution in the local system. This curve must be defined if LCIDM is nonzero. See Figure 10.5.
INOUT	If 0, elements whose center falls inside the projected curve are considered. If 1, elements whose center falls outside the projected curve are considered.
ICYCLE	Number of time steps between updating the list of active elements (default=200). The list update can be quite expensive and should be done at a reasonable interval. The default is not be appropriate for all problems.

Remarks:

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

***LOAD_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).

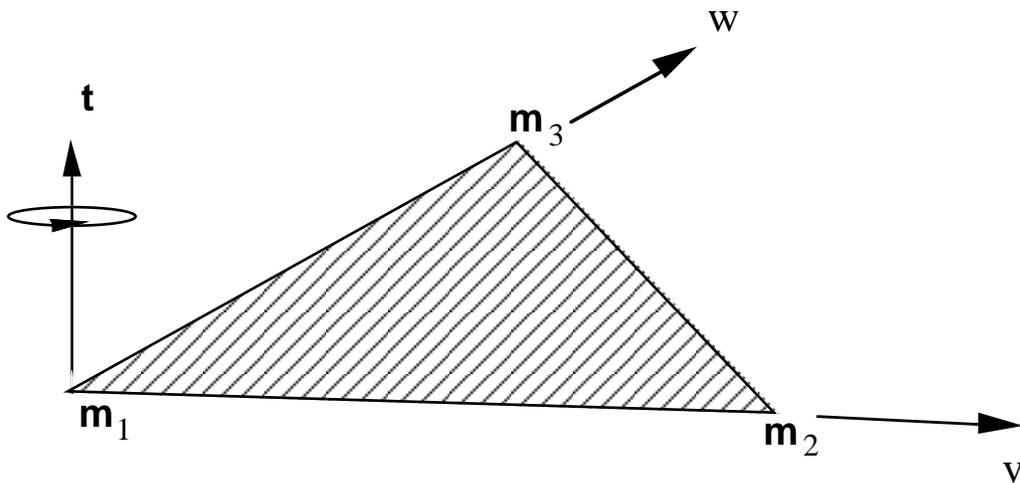


Figure 19.2. Follower force and moment acting on a plane defined by nodes m_1 , m_2 , and m_3 . In this case, the load is applied to node m_1 ; i.e., $m=m_1$. A positive force acts in the positive t -direction, and a positive moment puts a counterclockwise torque about the normal vector. The positive t -direction is found by the cross product $t = v \times w$ where v and w are vectors as shown.

*LOAD

*LOAD_RIGID_BODY

Purpose: Apply a concentrated nodal force to a rigid body. The force is applied at the center of mass or a moment is applied around a global axis. As an option, local axes can be defined for force or moment directions.

Card Format

	1	2	3	4	5	6	7	8
Variable	PID	DOF	LCID	SF	CID	M1	M2	M3
Type	I	I	I	F	I	I	I	I
Default	none	none	none	1.	0	0	0	0
Remark					1	2		

VARIABLE

DESCRIPTION

PID	Part ID of the rigid body, see *PART_OPTION.
DOF	Applicable degrees-of-freedom: EQ.1: x-direction of load action, EQ.2: y-direction of load action, EQ.3: z-direction of load action, EQ.4: follower force, see remark 2 on next page, EQ.5: moment about the x-axis, EQ.6: moment about the y-axis, EQ.7: moment about the z-axis. EQ.8: follower moment, see remark 2.
LCID	Load curve ID, see *DEFINE_CURVE. GT.0: force as a function of time, LT.0: force as a function of the absolute value of the rigid body displacement.
SF	Load curve scale factor
CID	Coordinate system ID
M1	Node 1 ID. Only necessary if DOF.EQ.4 or 8, see remark 2 on next page.

*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 If LCID is input as -2, then the ConWep function is used to determine the pressure for the segments, see *LOAD_BLAST.
3. The load curve multipliers may be used to increase or decrease the pressure. The time value is not scaled.
4. The activation time, AT, is the time during the solution that the pressure begins to act. Until this time, the pressure is ignored. The function value of the load curves will be evaluated at

*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				

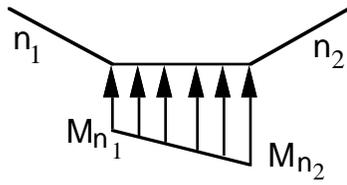
VARIABLE

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 If LCID is input as -2, then the ConWep function is used to determine the pressure for the segments, see *LOAD_BLAST.
3. The load curve multipliers may be used to increase or decrease the pressure. The time value is not scaled.
4. The activation time, AT, is the time during the solution that the pressure begins to act. Until this time, the pressure is ignored. The function value of the load curves will be evaluated at the offset time given by the difference of the solution time and AT i.e., (solution time-AT). Relative displacements that occur prior to reaching AT are ignored. Only relative displacements that occur after AT are prescribed.



2-Dimensional Definition for axisymmetric, plane stress, and plane strain geometries

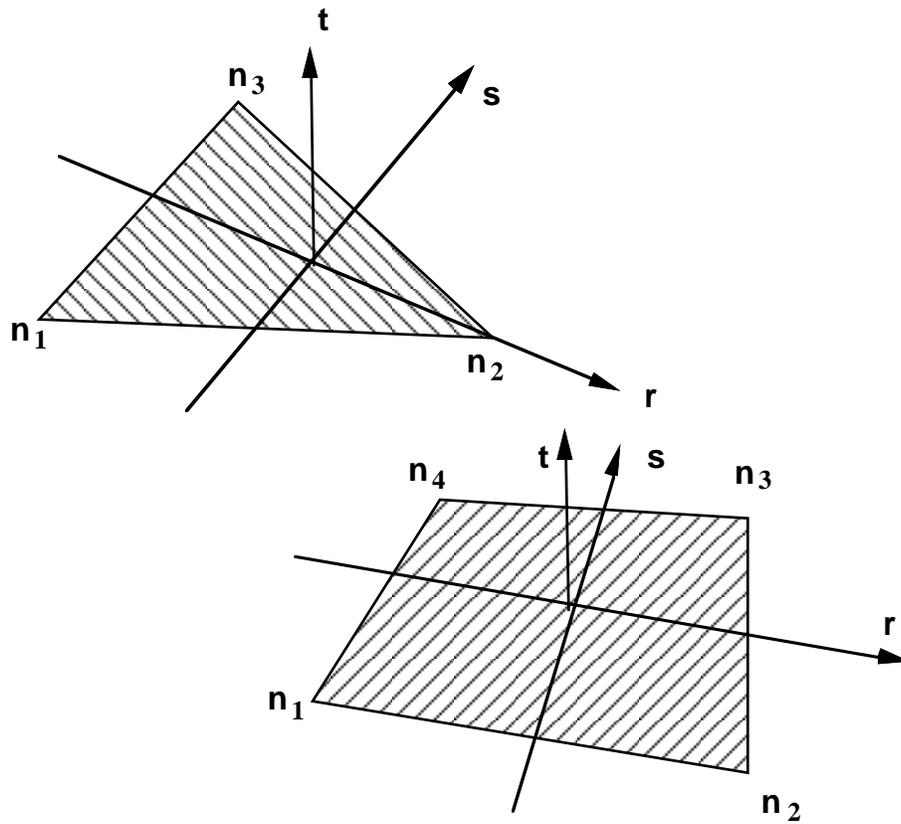


Figure 19.3. Nodal numbering for pressure cards. Positive pressure acts in the negative t -direction. For two dimensional problems repeat the second node for the third and fourth nodes in the segment definitions.

*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 If LCID is input as -2, then the ConWep function is used to determine the pressure for the segments, see *LOAD_BLAST.
3. The load curve multipliers may be used to increase or decrease the pressure. The time value is not scaled.

*LOAD

*LOAD_SSA

Purpose: The Sub-Sea Analysis capability allows a simple way of loading the structure to account for the effects of the primary explosion and the subsequent bubble oscillations.

Define one card.

Card 1 1 2 3 4 5 6 7 8

Variable	VS	DS	REFL	ZB	ZSURF	FPSID	PSID	
Type	F	F	F	F	F	I	I	
Default	none	none	0.	0.	0.	0	0	

Define two cards for each explosive charge. This input is terminated by the next “*” keyword card.

Card 1 1 2 3 4 5 6 7 8

Variable	A	ALPHA	GAMMA	KTHETA	KAPPA			
Type	F	F	F	F	F			
Default	none	none	none	none	none			

Card 2 1 2 3 4 5 6 7 8

Variable	XS	YS	ZS	W	TDELY	RAD	CZ	
Type	F	F	F	F	F	F	F	
Default	none	none	none	none	none	none	none	

VARIABLE	DESCRIPTION
VS	Sound speed in fluid
DS	Density of fluid
REFL	Consider reflections from sea floor. EQ.0: off EQ.1: on
ZB	Z coordinate of sea floor if REFL=1, otherwise, not used.
ZSURF	Z coordinate of sea surface
FPSID	Part set ID of parts subject to flood control. Use the *PART_SET_COLUMN option where the parameters A1 and A2 must be defined as follows: Parameter A1: Flooding status: EQ.1.0: Fluid on both sides. EQ.2.0: Fluid outside, air inside. EQ.3.0: Air outside, fluid inside. EQ.4.0: Material or part is ignored. Parameter A2: Tubular outer diameter of beam elements. For shell elements this input must be greater than zero for loading.
PSID	Part IDs of parts defining the wet surface. The elements defining these parts must have 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	α , shock pressure parameter
GAMMA	γ , time constant parameter
KTHETA	K_θ , time constant parameter
KAPPA	κ , ratio of specific heat capacities
XS	X coordinate of charge
YS	Y coordinate of charge
ZS	Z coordinate of charge
W	Weight of charge

*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 θ below are functions of the type and weight W of the explosive charge and the distance Q from the charge.

$$P_{peak} = A \left[\frac{W^{1/3}}{Q} \right]^\alpha$$

$$\theta = K_\theta W^{1/3} \left[\frac{W^{1/3}}{Q} \right]^\gamma$$

where A , α , γ , and K_θ are constants for the explosive being used.

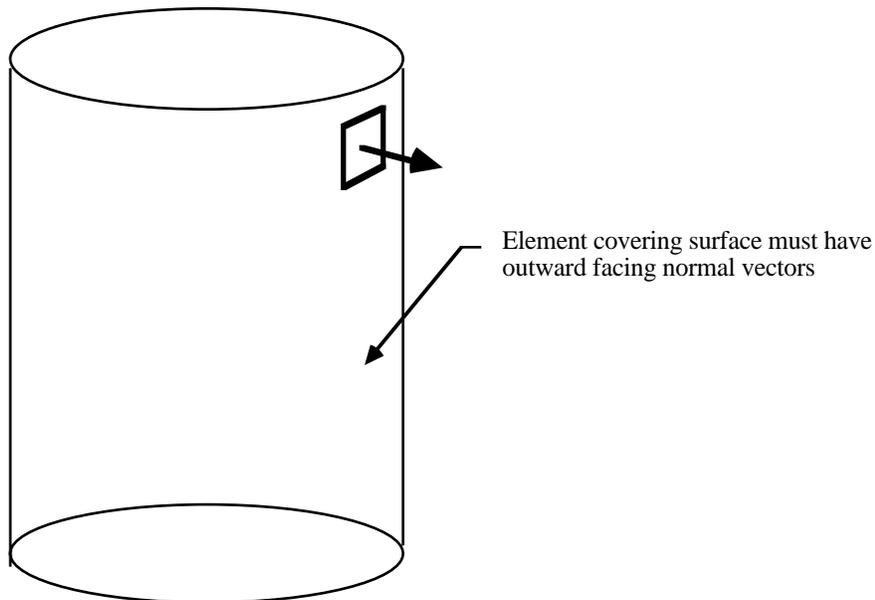


Figure 19.4. The shell elements interacting with the fluid must be numbered such that their outward normal vector points into the fluid media.

***LOAD_SUPERPLASTIC_FORMING**

Purpose: Perform superplastic forming (SPF) analyses. This option can be applied to both solid and shell elements. The pressure loading controlled by the load curve ID given below is scaled to maintain a constant maximum strain rate.

This option must be used with material model 64, *MAT_RATE_SENSITIVE_POWERLAW_PLASTICITY, for strain rate sensitive, powerlaw plasticity. For the output of data, see *DATABASE_SUPERPLASTIC_FORMING. Mass scaling is recommended in SPF applications.

Card Format

	1	2	3	4	5	6	7	8
Variable	LCP1	CSP1	NCP1	LCP2	CSP2	NCP2		
Type	I	I	F	I	I	F		
Default	none	none	none.	none	none	none		
Remarks				1	1	1		

	1	2	3	4	5	6	7	8
Variable	ERATE	SCMIN	SCMAX	NCYL				
Type	F	F	F	I				
Default	none	none	none.	0				
Remarks				2				

*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.

<i>*LOAD_THERMAL_CONSTANT</i>	- Thermal load type 1
<i>*LOAD_THERMAL_CONSTANT_NODE</i>	- Thermal load type 1
<i>*LOAD_THERMAL_LOAD_CURVE</i>	- Thermal load type 2
<i>*LOAD_THERMAL_TOPAZ</i>	- Thermal load type 3
<i>*LOAD_THERMAL_VARIABLE</i>	- Thermal load type 4
<i>*LOAD_THERMAL_VARIABLE_NODE</i>	- 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.						

<u>VARIABLE</u>	<u>DESCRIPTION</u>
NSID	Nodal set ID containing nodes for initial temperature (see *SET_NODES): EQ.0: all nodes are included:
NSIDEX	Nodal set ID containing nodes that are exempted from the imposed temperature (optional).
BOXID	All nodes in box which belong to NSID are initialized. Others are excluded (optional).
T	Temperature
TE	Temperature of exempted nodes (optional)

***LOAD_THERMAL_CONSTANT_NODE**

Purpose: Define nodal temperature that remains constant for the duration of the calculation. The reference temperature state is assumed to be a null state with this option. A nodal temperature state, read in above and held constant throughout the analysis, dynamically loads the structure. Thus, the temperature defined can also be seen as a relative temperature to a surrounding or initial temperature.

Card Format

	1	2	3	4	5	6	7	8
Variable	NID	T						
Type	I	F						
Default	none	0.						

VARIABLE

DESCRIPTION

NID	Node ID
T	Temperature, see remark below.

Remark:

1. The temperature range for the constitutive constants in the thermal materials must include the reference temperature of zero. If not termination will occur with a temperature out-of-range error immediately after the execution phase is entered.

*LOAD

*LOAD_THERMAL_LOAD_CURVE

Purpose: Nodal temperatures will be uniform throughout the model and will vary according to a load curve. The temperature at time=0 becomes the reference temperature for the thermal material. The reference temperature is obtained from the optional curve for dynamic relaxation if this curve is used. The load curve option for dynamic relaxation is useful for initializing preloads.

Card Format

1 2 3 4 5 6 7 8

Variable	LCID	LCIDDR						
Type	I	I						
Default	none	0						

VARIABLE

DESCRIPTION

LCID

Load curve ID, see *DEFINE_CURVE, to define temperature versus time.

LCIDDR

An optional load curve ID, see *DEFINE_CURVE, to define temperature versus time during the dynamic relaxation phase.

***LOAD_THERMAL_TOPAZ**

Purpose: Nodal temperatures will be read in from the TOPAZ3D database. This file is defined in the EXECUTION SYNTAX, see INTRODUCTION.

*LOAD

*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			

VARIABLE	DESCRIPTION
-----------------	--------------------

- | | |
|--------|--|
| NSID | Nodal set ID containing nodes (see *SET_NODE_OPTION):
EQ.0: all nodes are included. |
| NSIDEX | Nodal set ID containing nodes that are exempted (optional), see *SET_NODE_OPTION. |
| BOXID | All nodes in box which belong to NSID are initialized. Others are excluded. |
| TS | Scaled temperature. |

<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_{scale} , is the scaled temperature, and, T_{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				

<u>VARIABLE</u>	<u>DESCRIPTION</u>
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_{scale} is the scaled temperature

T_{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:

- 0** - Solids,
- 1H** - Hughes-Liu beam,
- 1B** - Belytschko resultant beam,
- 1I** - Belytschko integrated solid and tubular beams,
- 1T** - Truss,
- 1D** - Discrete beam,
- 1SW** - Spotweld beam,
- 2** - Shells,
- 3** - Thick shells.
- 4** - Special airbag element.
- 5** - SPH element.

***MAT_ADD_EROSION**

***MAT_NONLOCAL**

TYPE 1: ***MAT_ELASTIC_(OPTION)** [0,1H,1B,1I,1T,2,3,5]

TYPE 2: ***MAT_OPTIONTROPIC_ELASTIC** [0,2,3]

TYPE 3: ***MAT_PLASTIC_KINEMATIC** [0,1H,1I,1T,2,3,5]

TYPE 4: ***MAT_ELASTIC_PLASTIC_THERMAL** [0,1H,2,3]

TYPE 5: ***MAT_SOIL_AND_FOAM** [0]

TYPE 6: ***MAT_VISCOELASTIC** [0,1H,5]

TYPE 7: ***MAT_BLATZ-KO_RUBBER** [0,2]

TYPE 8: ***MAT_HIGH_EXPLOSIVE_BURN** [0,5]

TYPE 9: ***MAT_NULL** [0,1,2,5]

TYPE 10: ***MAT_ELASTIC_PLASTIC_HYDRO_(OPTION)** [0,5]

TYPE 11: ***MAT_STEINBERG** [0,5]

TYPE 11: ***MAT_STEINBERG_LUND** [0,5]

TYPE 12: ***MAT_ISOTROPIC_ELASTIC_PLASTIC** [0,2,3]

TYPE 13: ***MAT_ISOTROPIC_ELASTIC_FAILURE** [0]

TYPE 14: ***MAT_SOIL_AND_FOAM_FAILURE** [0,5]

TYPE 15: ***MAT_JOHNSON_COOK** [0,2,3,5]

TYPE 16: ***MAT_PSEUDO_TENSOR** [0,5]

TYPE 17: ***MAT_ORIENTED_CRACK** [0]

TYPE 18: ***MAT_POWER_LAW_PLASTICITY** [0,1H,2,3,5]

TYPE 19: ***MAT_STRAIN_RATE_DEPENDENT_PLASTICITY** [0,2,3,5]

TYPE 20: ***MAT_RIGID** [0,1H,1B,1T,2,3]

*MAT

TYPE 21: *MAT_ORTHOTROPIC_THERMAL [0,2,3]
TYPE 22: *MAT_COMPOSITE_DAMAGE [0,2,3]
TYPE 23: *MAT_TEMPERATURE_DEPENDENT_ORTHOTROPIC [0,2,3]
TYPE 24: *MAT_PIECEWISE_LINEAR_PLASTICITY [0,1H,2,3,5]
TYPE 25: *MAT_GEOLOGIC_CAP_MODEL [0,5]
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_SHAPE_MEMORY [0,5]
TYPE 31: *MAT_FRAZER_NASH_RUBBER_MODEL [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 [0,5]
TYPE 36: *MAT_3-PARAMETER_BARLAT [2,5]
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_MATERIAL_MODELS
TYPE 51: *MAT_BAMMAN [0,2,3,5]
TYPE 52: *MAT_BAMMAN_DAMAGE [0,2,3,5]
TYPE 53: *MAT_CLOSED_CELL_FOAM [0]
TYPE 54-55: *MAT_ENHANCED_COMPOSITE_DAMAGE [2]
TYPE 57: *MAT_LOW_DENSITY_FOAM [0,5]
TYPE 58: *MAT_LAMINATED_COMPOSITE_FABRIC [2]
TYPE 59: *MAT_COMPOSITE_FAILURE_OPTION_MODEL [0,2]
TYPE 60: *MAT_ELASTIC_WITH_VISCOSITY [0,2,5]
TYPE 61: *MAT_KELVIN-MAXWELL_VISCOELASTIC [0,5]
TYPE 62: *MAT_VISCOUS_FOAM [0]
TYPE 63: *MAT_CRUSHABLE_FOAM [0,5]
TYPE 64: *MAT_RATE_SENSITIVE_POWERLAW_PLASTICITY [0,2,3,5]
TYPE 65: *MAT_MODIFIED_ZERILLI_ARMSTRONG [0,2,5]
TYPE 66: *MAT_LINEAR_ELASITC_DISCRETE_BEAM

TYPE 67: *MAT_NONLINEAR_ELASITC_DISCRETE_BEAM
TYPE 68: *MAT_NONLINEAR_PLASITC_DISCRETE_BEAM
TYPE 69: *MAT_SID_DAMPER_DISCRETE_BEAM
TYPE 70: *MAT_HYDAULIC_GAS_DAMPER_DISCRETE_BEAM
TYPE 71: *MAT_CABLE_DISCRETE_BEAM
TYPE 72: *MAT_CONCRETE_DAMAGE [0,5]
TYPE 73: *MAT_LOW_DENSITY_VISCOUS_FOAM [0]
TYPE 74: *MAT_ELASTIC_SPRING_DISCRETE_BEAM
TYPE 75: *MAT_BILKHU/DUBOIS_FOAM [0,5]
TYPE 76: *MAT_GENERAL_VISCOELASTIC [0,2,5]
TYPE 77: *MAT_HYPERELASTIC_RUBBER [0,5]
TYPE 77: *MAT_OGDEN_RUBBER [0]
TYPE 78: *MAT_SOIL_CONCRETE [0]
TYPE 79: *MAT_HYSTERETIC_SOIL [0,5]
TYPE 80: *MAT_RAMBERG-OSGOOD [0]
TYPE 81: *MAT_PLASTICITY_WITH_DAMAGE [2,3]
TYPE 83: *MAT_FU_CHANG_FOAM [0,5]
TYPE 84: *MAT_WINFRITH_CONCRETE [0]
TYPE 84: *MAT_WINFRITH_CONCRETE_REINFORCEMENT [0]
TYPE 86: *MAT_ORTHOTROPIC_VISCOELASTIC [2]
TYPE 87: *MAT_CELLULAR_RUBBER [0,5]
TYPE 88: *MAT_MTS [0,2,5]
TYPE 89: *MAT_PLASTICITY_POLYMER [2]
TYPE 90: *MAT_ACOUSTIC [0]
TYPE 91: *MAT_SOFT_TISSUE_{OPTION} [0,2]
TYPE 93: *MAT_ELASTIC_6DOF_SPRING_DISCRETE_BEAM
TYPE 94: *MAT_INELASTIC_SPRING_DISCRETE_BEAM
TYPE 95: *MAT_INELASTC_6DOF_SPRING_DISCRETE_BEAM
TYPE 96: *MAT_BRITTLE_DAMAGE [0]
TYPE 97: *MAT_GENERAL_JOINT_DISCRETE_BEAM
TYPE 98: *MAT_SIMPLIFIED_JOHNSON_COOK [0,1H,1B,1T,2,3]
TYPE 99: *MAT_SIMPLIFIED_JOHNSON_COOK_ORTHOTROPIC_DAMAGE
TYPE 100: *MAT_SPOTWELD_{OPTION} [1SW]
TYPE 101: *MAT_GEPLASTIC_SRATE2000a [2]
TYPE 102: *MAT_INV_HYPERBOLIC_SIN [0]
TYPE 103: *MAT_ANISOTROPIC_VISCOPLASTIC [0,2]

*MAT

TYPE 103: *MAT_ANISOTROPIC_PLASTIC
TYPE 104: *MAT_DAMAGE_1 [0,2]
TYPE 105: *MAT_DAMAGE_2 [0,2]
TYPE 106: *MAT_ELASTIC_VISCOPLASTIC_THERMAL [0,2]
TYPE 110: *MAT_JOHNSON_HOLMQUIST_CERAMICS [0]
TYPE 111: *MAT_JOHNSON_HOLMQUIST_CONCRETE [0]
TYPE 112: *MAT_FINITE_ELASTIC_STRAIN_PLASTICITY [0,5]
TYPE 114: *MAT_LAYERED_LINEAR_PLASTICITY [2,3]
TYPE 115: *MAT_UNIFIED_CREEP [0,5]
TYPE 116: *MAT_COMPOSITE_LAYUP [2]
TYPE 117: *MAT_COMPOSITE_MATRIX [2]
TYPE 118: *MAT_COMPOSITE_DIRECT [2]
TYPE 119: *MAT_GENERAL_NONLINEAR_6DOF_DISCRETE_BEAM
TYPE 120: *MAT_GURSON [2]
TYPE 120: *MAT_GURSON_RCDC [2]
TYPE 121: *MAT_GENERAL_NONLINEAR_1DOF_DISCRETE_BEAM
TYPE 122: *MAT_HILL_3RC [2]
TYPE 123: *MAT_MODIFIED_PIECEWISE_LINEAR_PLASTICITY [2,3]
TYPE 124: *MAT_PLASTICITY_COMPRESSION_TENSION [0,5]
TYPE 126: *MAT_MODIFIED_HONEYCOMB [0]
TYPE 127: *MAT_ARRUDA_BOYCE_RUBBER [0,5]
TYPE 128: *MAT_HEART_TISSUE [0]
TYPE 129: *MAT_LUNG_TISSUE [0]
TYPE 130: *MAT_SPECIAL_ORTHOTROPIC [2]
TYPE 139: *MAT_MODIFIED_FORCE_LIMITED [1B]
TYPE 140: *MAT_VACUUM [0]
TYPE 141: *MAT_RATE_SENSITIVE_POLYMER
TYPE 142: *MAT_TRANSVERSELY_ANISOTROPIC_CRUSHABLE_FOAM [0]
TYPE 143: *MAT_WOOD [0]
TYPE 144: *MAT_PITZER_CRUSHABLE_FOAM [0]
TYPE 145: *MAT_SCHWER_MURRAY_CAP_MODEL
TYPE 146: *MAT_1DOF_GENERALIZED_SPRING
TYPE 147: *MAT_FHWA_SOIL [0]
TYPE 147: *MAT_FHWA_NEBRASKA [0]
TYPE 148: *MAT_GAS_MIXTURE [0]
TYPE 150: *MAT_CFD_{*OPTION*}

TYPE 154: *MAT_DESHPANDE_FLECK_FOAM
TYPE 161: *MAT_COMPOSITE_MSC [0]
TYPE 163: *MAT_MODIFIED_CRUSHABLE_FOAM [0]
TYPE 176: *MAT_QUASILINEAR_VISCOELASTIC [0]
TYPE 177: *MAT_HILL_FOAM [0]
TYPE 178: *MAT_VISCOELASTIC_HILL_FOAM [0]
TYPE 179: *MAT_LOW_DENSITY_SYNTHETIC_FOAM [0]
TYPE 181: *MAT_SIMPLIFIED_RUBBER [0]
TYPE 191: *MAT_SEISMIC_BEAM [1B]
TYPE 192: *MAT_SOIL_BRICK [0]
TYPE 193: *MAT_DRUCKER_PRAGER [0]
TYPE 194: *MAT_RC_SHEAR_WALL [2]
TYPE 195: *MAT_CONCRETE_BEAM [1H]
TYPE 196: *MAT_GENERAL_SPRING_DISCRETE_BEAM
TYPE 197: *MAT_SEISMIC_ISOLATOR
TYPE 198: *MAT_JOINTED_ROCK

For the discrete (type 6) beam elements, which are used to model complicated dampers and multi-dimensional spring-damper combinations, the following material types are available:

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 74: *MAT_ELASTIC_SPRING_DISCRETE_BEAM [1D]
TYPE 93: *MAT_ELASTIC_6DOF_SPRING_DISCRETE_BEAM [1D]
TYPE 94: *MAT_INELASTIC_SPRING_DISCRETE_BEAM [1D]
TYPE 95: *MAT_INELASTIC_6DOF_SPRING_DISCRETE_BEAM [1D]
TYPE 119: *MAT_GENERAL_NONLINEAR_6DOF_DISCRETE_BEAM [1D]
TYPE 121: *MAT_GENERAL_NONLINEAR_1DOF_DISCRETE_BEAM [1D]
TYPE 146: *MAT_1DOF_GENERALIZED_SPRING [1D]
TYPE 196: *MAT_GENERAL_SPRING_DISCRETE_BEAM [1D]

For the discrete springs and dampers thirteen material types are available. In the structured input separate type numbers are assigned to this element class.

TYPE 1: *MAT_SPRING_ELASTIC
TYPE 2: *MAT_DAMPER_VISCOUS

*MAT

TYPE 3: *MAT_SPRING_ELASTOPLASTIC
TYPE 4: *MAT_SPRING_NONLINEAR_ELASTIC
TYPE 5: *MAT_DAMPER_NONLINEAR_VISCOUS
TYPE 6: *MAT_SPRING_GENERAL_NONLINEAR
TYPE 7: *MAT_SPRING_MAXWELL
TYPE 8: *MAT_SPRING_INELASTIC
TYPE 13: *MAT_SPRING_TRILINEAR_DEGRADING
TYPE 14: *MAT_SPRING_SQUAT_SHEARWALL
TYPE 15: *MAT_SPRING_MUSCLE

For the seatbelts one material is available. No type numbers were used for this material type:

*MAT_SEATBELT

For incompressible CFD analysis, or for coupled incompressible fluid-structure interaction problems, the *MAT_CFD_OPTION keyword may be used to specify fluid properties. The fluid properties may be defined only for solid and shell elements.

TYPE 150: *MAT_CFD_OPTION

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. In the structured input separate type numbers are assigned to the thermal property definitions.

*MAT_THERMAL_OPTION

TYPE 1: *MAT_THERMAL_ISOTROPIC
TYPE 2: *MAT_THERMAL_ORTHOTROPIC
TYPE 3: *MAT_THERMAL_ISOTROPIC_TD
TYPE 4: *MAT_THERMAL_ORTHOTROPIC_TD
TYPE 5: *MAT_THERMAL_ISOTROPIC_PHASE_CHANGE
TYPE 6: *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.

An additional option **_TITLE** may be appended to all the *MAT keywords. If this option is used then an additional line is read for each section in 80a format which can be used to describe the material. At present LS-DYNA does not make use of the title. Inclusion of titles gives greater clarity to input decks.

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
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	Bricks	Beams	Thin Shells	Thick Shells	Strain-Rate Effects				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
						Failure						
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	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										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	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 Gl Glass Hy Hydro-dyn Mt Metal Pl Plastic Rb Rubber Sl Soil/Conc
51	Bamman (Temp /Rate Dependent Plasticity)	Y	Y	Y	Y	Y				Gn
52	Bamman Damage	Y	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	Laminated Composite Fabric		Y							
59	Composite Failure (Plasticity Based)	Y	Y			Y				Cm, Cr
60	Elastic with Viscosity (Viscous Glass)	Y	Y			Y		Y		Gl
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 SI Soil/Conc
71	Cable Discrete Beam (Elastic)		Y							
72	Concrete Damage	Y				Y	Y	Y		SI
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			SI
79	Hysteretic Soil (Elasto-Perfectly Plastic)	Y					Y			SI
80	Ramberg Osgood									
81	Plasticity with Damage (Elasto-Plastic)			Y	Y	Y	Y			Mt, Pl
83	Fu Chang Foam	Y				Y	Y			Fm
84	Winfrith Concrete	Y								
84	Winfrith Concrete Reinforcement	Y								
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
91	Soft Tissue	Y	Y							

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
93	Elastic 6DOF Spring Discrete Beam		Y							
94	Inelastic Spring Discrete Beam		Y							
95	Inelastic 6DOF Spring Discrete Beam		Y							
96	Brittle Damage	Y				Y	Y			
97	General Joint Discrete Beam		Y							
98	Simplified Johnson Cook	Y	Y	Y	Y					
99	Simplified Johnson Cook Orthotropic Damage									
100	Spotweld		Y							
101	GEPLASTIC Srate2000a			Y						
102	Inv Hyperbolic Sin	Y								
103	Anisotropic Viscoplastic	Y		Y						
104	Damage 1	Y		Y						
105	Damage 2	Y		Y						
106	Elastic Viscoplastic Thermal	Y		Y					Y	
107										
108										
109										
110	Johnson Holmquist Ceramics	Y								
111	Johnson Holmquist Concrete	Y								
112	Finite Elastic Strain Plasticity	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
114	Layered Linear Plasticity			Y	Y		
115	Unified Creep	Y					
116	Composite Layup			Y			
117	Composite Matrix			Y			
118	Composite Direct			Y			
120	Gurson			Y			
121	Generalized Nonlinear 1DOF Discrete Beam		Y				
122	Hill 3RC						
123	Modified Piecewise Linear Plasticity			Y	Y		
124	Plasticity Compression Tension	Y					
126	Modified Honeycomb	Y					
127	Arruda Boyce Rubber	Y					
128	Heart Tissue	Y					
129	Lung Tissue	Y					
130	Special Orthotropic			Y			
139	Modified Force Limited		Y				
140	Vacuum						
141	Rate Sensitive Polymer						
142	Transversely Anisotropic Crushable Foam						
143	Wood						
144	Pitzer Crushable Foam						

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
145	Schwer Murray Cap Model									
146	1DOF Generalized Spring									
147	FHWA Soil									
147	FHWA Soil Nebraska									
148	Gas Mixture									
150	CFD									
154	Deshpande Fleck Foam									
161	Composite MSC		Y							
163	Modified Crushable Foam									
176	Quasilinear Viscoelastic									
177	Hill Foam									
178	Viscoelastic Hill Foam									
179	Low Density Synthetic Foam									
181	Simplified Rubber									
191	Seismic Beam		Y							
192	Soil Brick	Y								
193	Drucker Prager	Y								
194	RC Shear Wall			Y						
195	Concrete Beam		Y							
196	General Spring Discrete Beam		Y							

*MAT

Material number	Material Title	Bricks	Beams	Thin Shells	Thick Shells	Strain-Rate Effects	Failure	Equation-of-State	Thermal Effects	Notes:
										Gn General
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								
DS13	Spring Trilinear Degrading									
DS14	Spring Squat Shearwall									
DS15	Spring Muscle									
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		
TH5	Thermal Isotropic (Phase Change)	Y	Y					Y		
TH6	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	FAILTM
Type	F	F	F	F	F	F	F	F
Default	none	none						

VARIABLE

DESCRIPTION

- 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, σ_{\max} .
SIGVM	Equivalent stress at failure, $\bar{\sigma}_{\max}$.
EPSP1	Principal strain at failure, ϵ_{\max} .
EPSSH	Shear strain at failure, γ_{\max} .
SIGTH	Threshold stress, σ_0 .
IMPULSE	Stress impulse for failure, K_f .
FAILTM	Failure time. When the problem time exceeds the failure time, the material is removed.

The criteria for failure besides failure time 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 σ_1 is the maximum principal stress, and σ_{\max} is the principal stress at failure.
3. $\sqrt{\frac{3}{2} \sigma'_{ij} \sigma'_{ij}} \geq \bar{\sigma}_{\max}$, where σ'_{ij} are the deviatoric stress components, and $\bar{\sigma}_{\max}$ is the equivalent stress at failure.
4. $\epsilon_1 \geq \epsilon_{\max}$, where ϵ_1 is the maximum principal strain, and ϵ_{\max} is the principal strain at failure.
5. $\gamma_1 \geq \gamma_{\max}$, where γ_1 is the shear strain, and γ_{\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 σ_1 is the maximum principal stress, σ_0 is a specified threshold stress, $\sigma_1 \geq \sigma_0 \geq 0$, and K_f is the stress impulse for failure. Stress values below the threshold value are too low to cause fracture even for very long duration loadings.

These failure models apply only to solid elements with one point integration in 2 and 3 dimensions.

*MAT_NONLOCAL

In nonlocal failure theories the failure criterion depends on the state of the material within a radius of influence which surrounds the integration point. An advantage of nonlocal failure is that mesh size sensitivity on failure is greatly reduced leading to results which converge to a unique solution as the mesh is refined. Without a nonlocal criterion, strains will tend to localize randomly with mesh refinement leading to results which can change significantly from mesh to mesh. The nonlocal failure treatment can be a great help in predicting the onset and the evolution of material failure. This option can be used with two and three-dimensional solid elements, and three-dimensional shell elements. The implementation is available for under integrated elements, which have one integration point at their center. Shells are assumed to have multiple integration points through their thickness. This is a new option and should be used with caution. This option applies to a subset of elastoplastic materials that include a damage - based failure criterion.

Define the following cards:

Card 1 1 2 3 4 5 6 7 8

Variable	IDNL	PID	P	Q	L	NFREQ		
Type	I	I	I	I	F	I		
Default	none	none	none	none	none	none		

Card 2

Variable	NL1	NL2	NL3	NL4	NL5	NL6	NL7	NL8
Type	I	I	I	I	I	I	I	I
Default	none							

Define one card for each symmetry plane. Up to six symmetry planes can be defined. The next "*" card terminates this input.

Cards 3,...

Variable	XC1	YC1	ZC1	XC2	YC2	ZC2		
Type	F	F	F	F	F	F		
Default	none	none	none	none	none	none		

<u>VARIABLE</u>	<u>DESCRIPTION</u>
IDNL	Nonlocal material input ID.
PID	Part ID for nonlocal material.
P	Exponent of weighting function. A typical value might be 8 depending somewhat on the choice of L. See equations below.
Q	Exponent of weighting function. A typical value might be 2. See equations below.
L	Characteristic length. This length should span a few elements. See the equations below.
NFREQ	Number of time steps between update of neighbours. The nearest neighbour search can add significant computational time so it is suggested that NFREQ be set to value of 10 to 100 depending on the problem. This parameter may be somewhat problem dependent.
NL1,...,NL8	Define up to eight history variable ID's for nonlocal treatment.
XC1, YC1,ZC1	Coordinate of point on symmetry plane.
XC2, YC2, ZC2	Coordinate of a point along the normal vector.

Remarks:

The memory usage for this option can vary during the duration of the calculation. It is recommended that additional memory be requested by using the *CONTROL_NONLOCAL input. Usually, a value of 10 should be okay.

For elastoplastic material models in LS-DYNA which use the plastic strain as a failure criterion, the first history variable, which does not count the six stress components, is the plastic strain. In this case the variable NL1=1 and NL2 - NL8=0. See the table below, which lists the history variable ID's for a subset of materials.

Material Model Name	Effective Plastic Strain Location	Damage Parameter Location
PLASTIC_KINEMATIC	1	N/A
JOHNSON_COOK	1	6
PIECEWISE_LINEAR_PLASTICITY	1	N/A
PLASTICITY_WITH_DAMAGE	1	2
MODIFIED_ZERILLI-ARMSTRONG	1	N/A
DAMAGE_1	1	4
DAMAGE_2	1	2
MODIFIED_PIECEWISE_LINEAR_PLAST	1	N/A
PLASTICITY_COMPRESSION_TENSION	1	N/A
JOHNSON_HOLMQUIST_CONCRETE	1	2
GURSON	1	2

In applying the nonlocal equations to shell elements, integration points lying in the same plane within the radius determined by the characteristic length are considered. Therefore, it is important to define the connectivity of the shell elements consistently within the part ID, e.g., so that the outer integration points lie on the same surface.

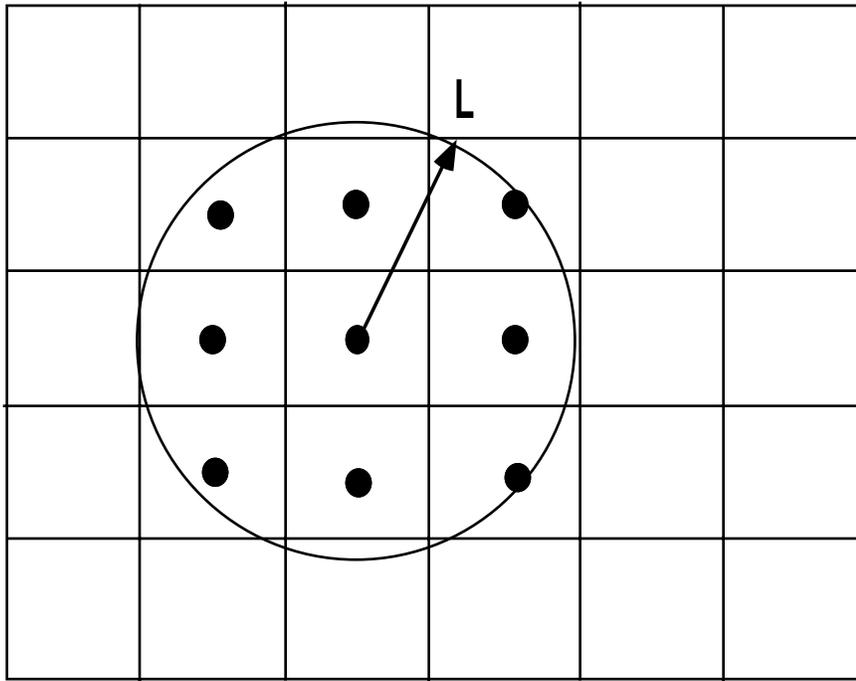
The equations and our implementation are based on the implementation by Worswick and Lalbin [1999] of the nonlocal theory to Pijaudier-Cabot and Bazant [1987]. Let Ω_r be the neighborhood of radius, L , of element e_r and $\{e_i\}_{i=1,\dots,N_r}$ the list of elements included in Ω_r , then

$$\dot{f}_r = \dot{f}(x_r) = \frac{1}{W_r} \int_{\Omega_r} \dot{f}_{local} w(x_r - y) dy \approx \frac{1}{W_r} \sum_{i=1}^{N_r} \dot{f}_{local}^i w_{ri} V_i$$

where

$$W_r = W(x_r) = \int w(x_r - y)dy \approx \sum_{i=1}^{N_r} w_{ri} V_i$$
$$w_{ri} = w(x_r - y_i) = \frac{1}{\left[1 + \left(\frac{\|x_r - y_i\|}{L}\right)^p\right]^q}$$

Here \dot{f}_r and x_r are respectively the nonlocal rate of increase of damage and the center of the element e_r , and \dot{f}_{local}^i , V_i and y_i are respectively the local rate of increase of damage, the volume and the center of element e_i .



*MAT_ELASTIC_{OPTION}

This is Material Type 1. This is an isotropic elastic material and is available for beam, shell, and solid elements in LS-DYNA. A specialization of this material allows the modeling of fluids.

Options include:

<BLANK>

FLUID

such that the keyword cards appear:

*MAT_ELASTIC or MAT_001

*MAT_ELASTIC_FLUID or MAT_001_FLUID

The fluid option is valid for solid elements only.

Define the following card for all options:

Card 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						

<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.
DA	Axial damping factor (used for Belytschko-Schwer beam, type 2, only).
DB	Bending damping factor (used for Belytschko-Schwer beam, type 2, only).
K	Bulk Modulus (define for fluid option only).
VC	Tensor viscosity coefficient, values between .1 and .5 should be okay.
CP	Cavitation pressure (default = 1.0e+20).

Remarks:

The axial and bending damping factors are used to damp down numerical noise. The update of the force resultants, F_i , and moment resultants, M_i , includes the damping factors:

$$F_i^{n+1} = F_i^n + \left(1 + \frac{DA}{\Delta t}\right) \Delta F_i^{n+\frac{1}{2}}$$

$$M_i^{n+1} = M_i^n + \left(1 + \frac{DB}{\Delta t}\right) \Delta M_i^{n+\frac{1}{2}}$$

For the fluid option the bulk modulus (K) has to be defined as Young's modulus, and Poission'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, ρ 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. For orthotropic solids and isotropic frictional damping is available.

Options include:

ORTHO

ANISO

such that the keyword cards appear:

***MAT_ORTHOTROPIC ELASTIC or MAT_002** (4 cards follow)

***MAT_ANISOTROPIC ELASTIC or MAT_002_ANIS** (5 cards follow)

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	G	SIGF		
Type	F	F	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>
MID	Material identification. A unique number has to be chosen.
RO	Mass density.

Define for the ORTHO option only:

EA	E_a , Young's modulus in a-direction.
EB	E_b , Young's modulus in b-direction.
EC	E_c , Young's modulus in c-direction.
PRBA	ν_{ba} , Poisson's ratio ba.
PRCA	ν_{ca} , Poisson's ratio ca.
PRCB	ν_{cb} , Poisson's ratio cb.
GAB	G_{ab} , shear modulus ab.
GBC	G_{bc} , shear modulus bc.
GCA	G_{ca} , shear modulus ca.

Due to symmetry define the upper triangular Cij's for the ANISO option only:

C11	The 1,1 term in the 6×6 anisotropic constitutive matrix. Note that 1 corresponds to the <i>a</i> material direction
C12	The 1,2 term in the 6×6 anisotropic constitutive matrix. Note that 2 corresponds to the <i>b</i> material direction
.	.
.	.
.	.
C66	The 6,6 term in the 6×6 anisotropic constitutive matrix.

Define for both options:

AOPT	Material axes option, see Figure 20.1:
------	--

VARIABLE	DESCRIPTION
	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. This option is for solid elements only.
	EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR.
	EQ. 3.0: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle, BETA, from a line in the plane of the element defined by the cross product of the vector v with the element normal. The plane of a solid element is the midsurface between the inner surface and outer surface defined by the first four nodes and the last four nodes of the connectivity of the element, respectively.
	EQ. 4.0: locally orthotropic in cylindrical coordinate system with the material axes determined by a vector v, and an originating point, P, which define the centerline axis. This option is for solid elements only.
G	Shear modulus for frequency independent damping. Frequency independent damping is based of a spring and slider in series. The critical stress for the slider mechanism is SIGF defined below. For the best results, the value of G should be 250-1000 times greater than SIGF. This option applies only to solid elements.
SIGF	Limit stress for frequency independent, frictional, damping.
XP YP ZP	Define coordinates of point p for AOPT = 1 and 4.
A1 A2 A3	Define components of vector a for AOPT = 2.
V1 V2 V3	Define components of vector v for AOPT = 3 and 4.
D1 D2 D3	Define components of vector d for AOPT = 2:
BETA	Material angle in degrees for AOPT = 3, may be overridden on the element card, see *ELEMENT_SHELL_BETA or *ELEMENT_SOLID_ORTHO.
REF	Use reference geometry to initialize the stress tensor. The reference 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:

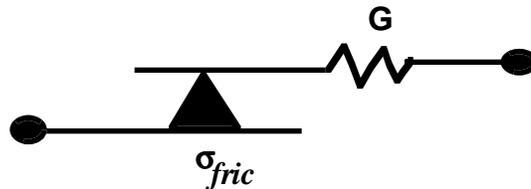
$$\underset{\sim}{C} = \underset{\sim}{T}^T \underset{\sim}{C} \underset{\sim}{T}$$

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:

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

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

The frequency independent damping is obtained by the having a spring and slider in series as shown in the following sketch:



This option applies only to orthotropic solid elements and affects only the deviatoric stresses.

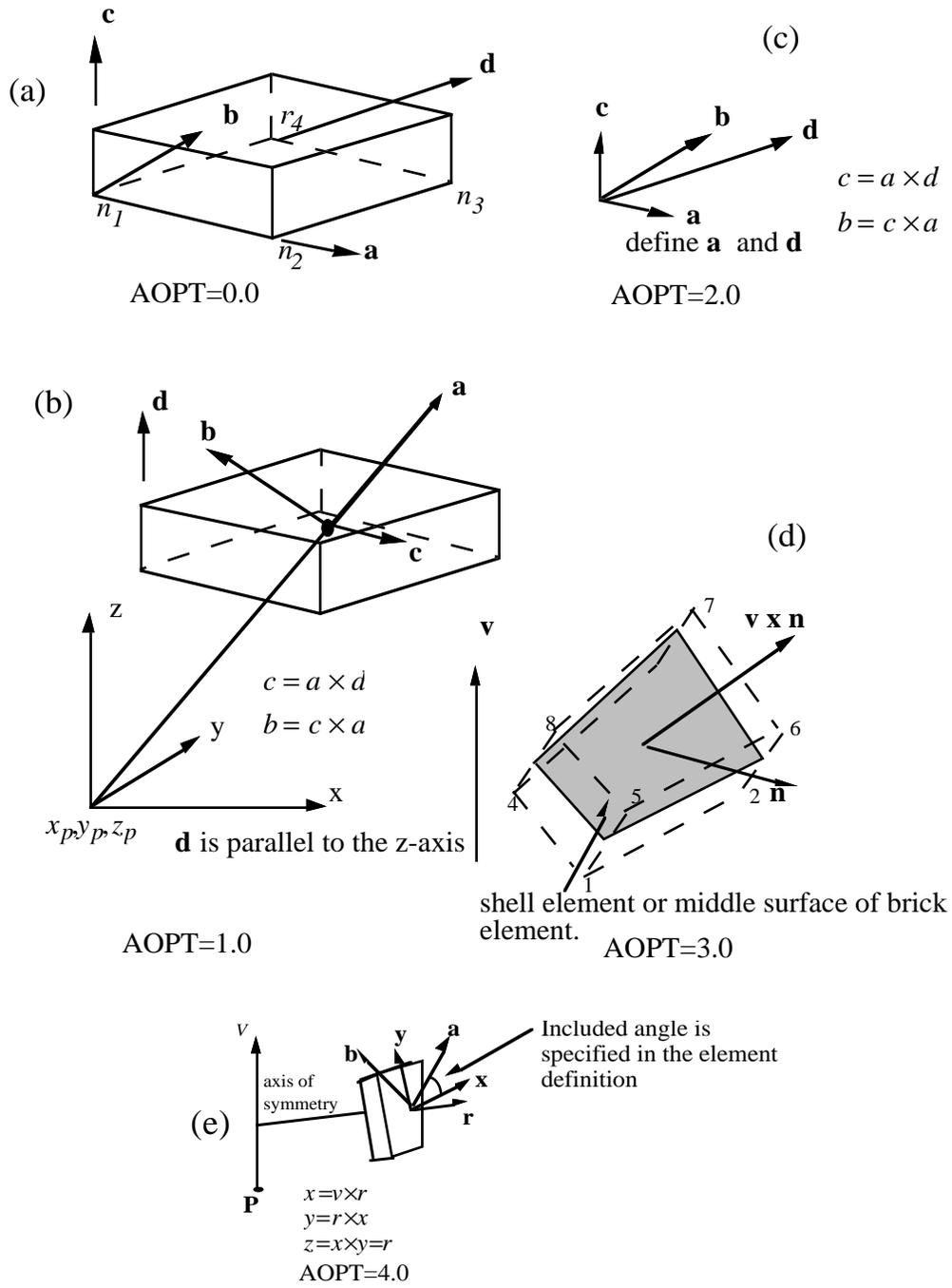


Figure 20.1. Options for determining principal material axes: (a) AOPT = 0.0, (b) AOPT = 1.0, (c) AOPT = 2.0, (d) AOPT = 3.0, and (e) AOPT=4.0 for brick elements.

***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				

<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, see Figure 20.2.
BETA	Hardening parameter, $0 < \beta' < 1$. See comments below.

<u>VARIABLE</u>	<u>DESCRIPTION</u>
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 β' between 0 and 1. For β' 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.

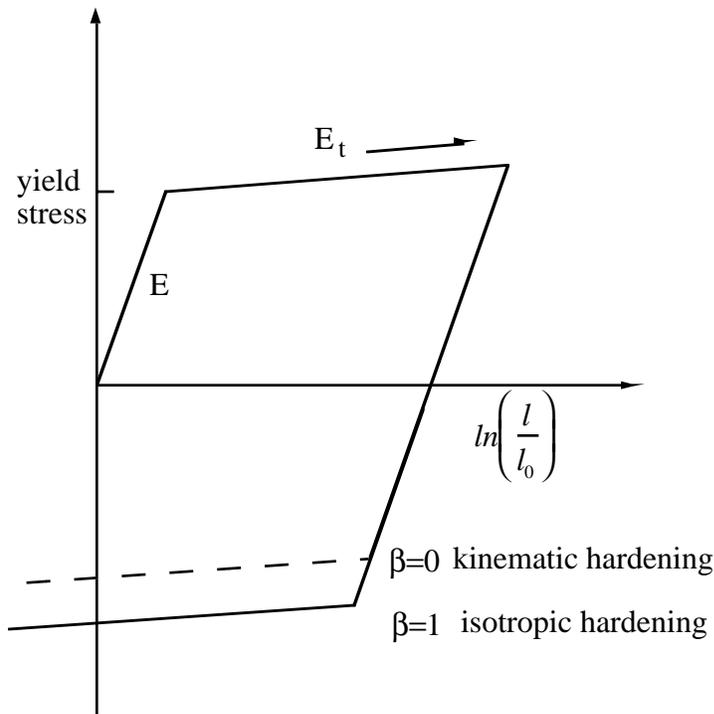


Figure 20.2. Elastic-plastic behavior with kinematic and isotropic hardening where l_0 and l are undeformed and deformed lengths of uniaxial tension specimen. E_t is the slope of the bilinear stress strain curve.

*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.
EPI	Corresponding plastic hardening moduli.

Remarks:

At least two temperatures and their corresponding material properties must be defined. The analysis will be terminated if a material temperature falls outside the range defined in the input. If a thermoelastic material is considered, do not define SIGY and ETAN. The coefficient of thermal expansion is defined as the instantaneous value. Thus, the thermal strain rate becomes:

$$\dot{\epsilon}_{ij}^T = \alpha \dot{T} \delta_{ij}$$

***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						

VARIABLE

DESCRIPTION

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 do 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].

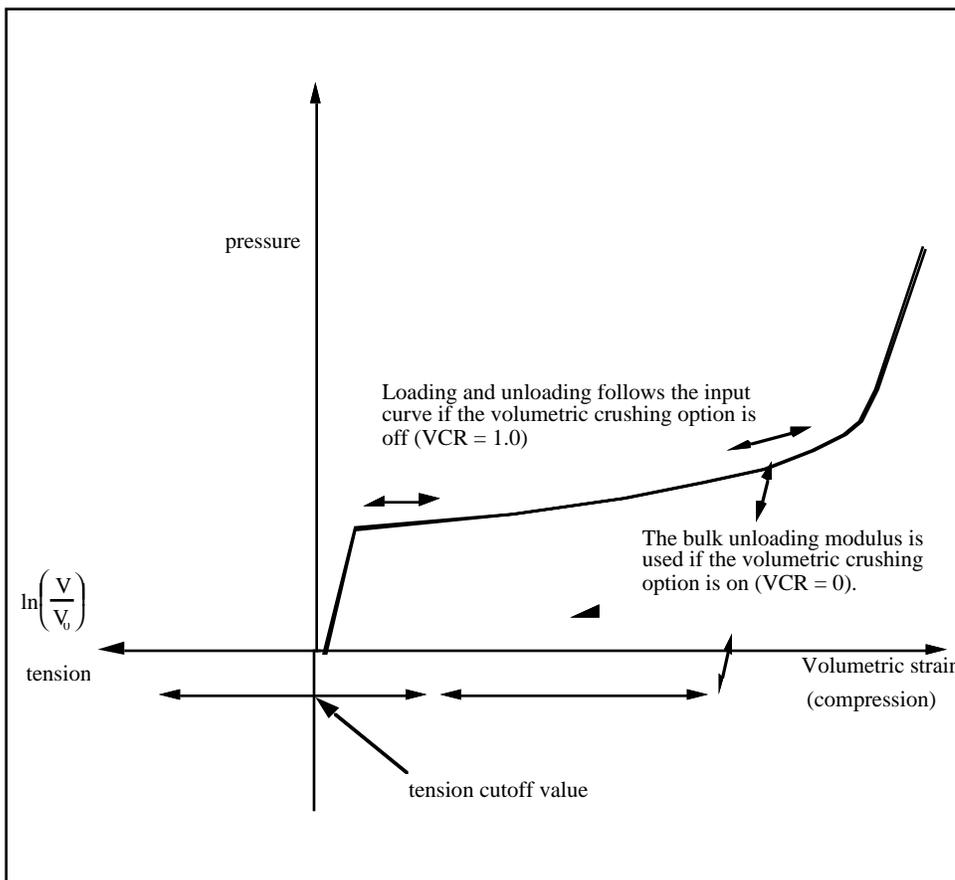


Figure 20.3. Pressure versus volumetric strain curve for soil and crushable foam model. The volumetric strain is given by the natural logarithm of the relative volume, V.

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

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

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

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

On the yield surface $J_2 = \frac{1}{3} \sigma_y^2$ where σ_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		

<u>VARIABLE</u>	<u>DESCRIPTION</u>
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_∞ .
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				

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 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 ν is Poisson's ratio, which is set to .463 internally. This stress measure is transformed to the Cauchy stress, σ_{ij} , according to the relationship

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

where F_{ij} is the deformation gradient tensor. 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

<u>VARIABLE</u>	<u>DESCRIPTION</u>
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	σ_y , yield stress (BETA=2.0 only).

Remarks:

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

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

where p_{eos} , is the pressure from the equation of state (either types 2 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 σ_y . If yielding has occurred, i.e., $\phi > 0$, the deviatoric trial stress is scaled to obtain the final deviatoric stress at time $n+1$:

$$s_{ij}^{n+1} = \frac{\sigma_y}{\sqrt{3J_2}} * s_{ij}^{n+1}$$

If $\phi \leq 0$, 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 (≤ 0.0).
MU	Dynamic viscosity coefficient μ (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.

VARIABLE	DESCRIPTION
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:

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

$$\sigma_{ij} = \mu \epsilon'_{ij}$$

$$\left[\frac{N}{m^2} \right] \approx \left[\frac{N}{m^2} s \right] \left[\frac{1}{S} \right]$$

is computed for nonzero μ where ϵ'_{ij} is the deviatoric strain rate. μ is the dynamic viscosity with unit of [Pascal*second].

2. The null material has no shear stiffness and hourglass control must be used with great care. In some applications, the default hourglass coefficient might lead to significant energy losses. In general for fluid(s), the hourglass coefficient QM should be small (in the range 1.0E-4 to 1.0E-6 in the SI unit system for the standard default IHQ choice).
3. The Null material has no yield strength and behaves in a fluid-like manner.
4. The pressure cut-off, PC, must be defined to allow for a material to “numerically” cavitate. In other words, when a material undergoes dilatation above certain magnitude, it should no longer be able to resist this dilatation. Since dilatation stress or pressure is negative, setting PC limit to a very small negative number would allow for the material to cavitate once the pressure in the material goes below this negative value.

*MAT_ELASTIC_PLASTIC_HYDRO_{OPTION}

This is Material Type 10. This material allows the modeling of an elastic-plastic hydrodynamic material.

Options include:

<BLANK>

SPALL

The keyword card can appear in two ways:

*MAT_ELASTIC_PLASTIC_HYDRO or MAT_010

*MAT_ELASTIC_PLASTIC_HYDRO_SPALL or MAT_010_SPALL

Card 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	-∞	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 (≤ 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, β , = 1. The yield strength is calculated as

$$\sigma_y = \sigma_0 + E_h \bar{\epsilon}^p + (a_1 + p a_2) \max[p, 0]$$

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, σ_{max} , exceeds the limiting value $-p_{cut}$. Note that the negative sign is required because p_{cut} is measured positive in compression, while σ_{max} is positive in tension. Once spall is detected with this model, the deviatoric stresses are reset to zero, and no hydrostatic tension ($p < 0$) is permitted. If tensile pressures are calculated, they are reset to 0 in the spalled material. Thus, the spalled material behaves as a rubble or incohesive material. The hydrostatic tension spall model, SPALL=3, detects spall if the pressure becomes more tensile than the specified limit, p_{cut} . Once spall is detected the deviatoric stresses are reset to zero, and nonzero values of pressure are required to be compressive (positive). If hydrostatic tension ($p < 0$) is subsequently calculated, the pressure is reset to 0 for that element.

This model is applicable to a wide range of materials, including those with pressure-dependent yield behavior. 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.

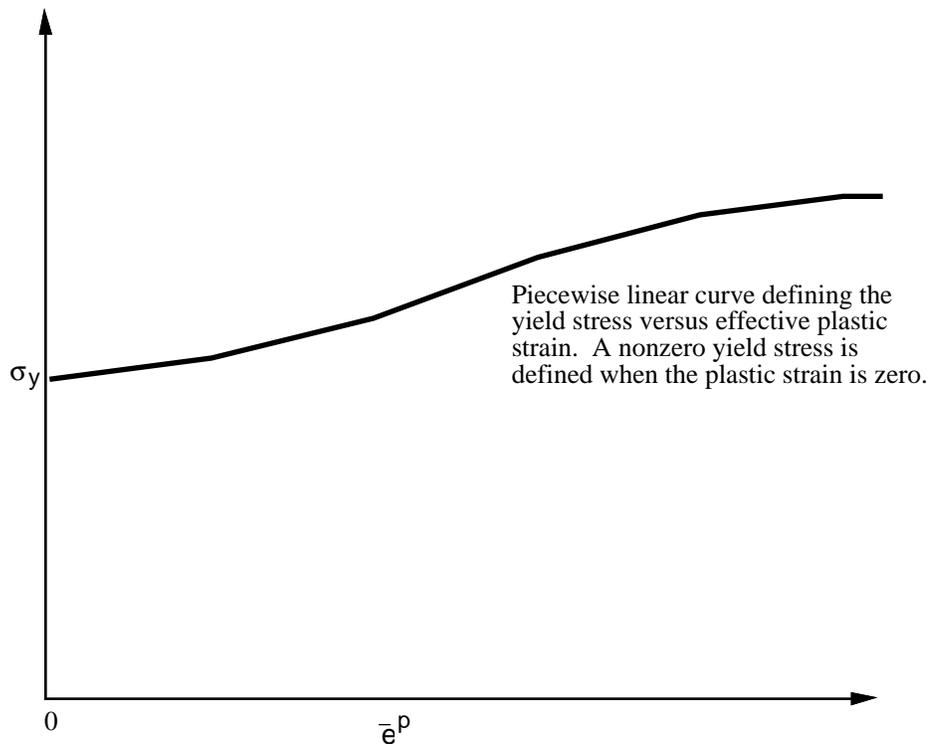


Figure 20.4. Effective stress versus effective plastic strain curve.

*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

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
G0	Basic shear modulus.
SIGO	σ_o , see defining equations.
BETA	β , see defining equations.
N	n, see defining equations.
GAMA	γ_i , initial plastic strain, see defining equations.
SIGM	σ_m , see defining equations.
B	b, see defining equations.
BP	b', see defining equations.
H	h, see defining equations.
F	f, see defining equations.
A	Atomic weight (if = 0.0, R' must be defined).
TMO	T_{mo} , see defining equations.
GAMO	γ_o , see defining equations.
SA	a, see defining equations.
PC	p_{cut} or $-\sigma_f$ (default=-1.e+30)
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 μ coefficients for the cold compression energy fit. Default is η .

VARIABLE	DESCRIPTION
MMN	μ_{\min} or η_{\min} . Optional μ or η minimum value.
MMX	μ_{\max} or η_{\max} . Optional μ or η maximum value.
EC0,...EC9	Cold compression energy coefficients (optional).

Remarks:

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

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

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

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 σ_y is given by:

$$\sigma_y = \sigma'_0 \left[1 + b'pV^{1/3} - h \left(\frac{E_i - E_c}{3R'} - 300 \right) \right] e^{-E_i/E_m - E_i}$$

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

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

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

If the coefficients EC_0, \dots, EC_9 are not defined above, LS-DYNA will fit the cold compression energy to a ten term polynomial expansion either as a function of μ or η depending on the input variable, FLAG, as:

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

$$E_c(\mu^i) = \sum_{i=0}^9 EC_i \mu^i$$

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, σ_{max} , exceeds the limiting value $-p_{cut}$. Note that the negative sign is required because p_{cut} is measured positive in compression, while σ_{max} is positive in tension. Once spall is detected with this model, the deviatoric stresses are reset to zero, and no hydrostatic tension ($p < 0$) is permitted. If tensile pressures are calculated, they are reset to 0 in the spalled material. Thus, the spalled material behaves as a rubble or incohesive material. The hydrostatic tension spall model, $SPALL=3$, detects spall if the pressure becomes more tensile than the specified limit, p_{cut} . Once spall is detected the deviatoric stresses are reset to zero, and nonzero values of pressure are required to be compressive (positive). If hydrostatic tension ($p < 0$) is subsequently calculated, the pressure is reset to 0 for that element.

This model is applicable to a wide range of materials, including those with pressure-dependent yield behavior. 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.

The keyword cards can appear in two ways:

***MAT_STEINBERG_LUND or MAT_011_LUND**

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		

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
G0	Basic shear modulus.
SIGO	σ_0 , see defining equations.
BETA	β , see defining equations.
N	n, see defining equations.
GAMA	γ_i , initial plastic strain, see defining equations.
SIGM	σ_m , see defining equations.
B	b, see defining equations.
BP	b', see defining equations.
H	h, see defining equations.
F	f, see defining equations.
A	Atomic weight (if = 0.0, R' must be defined).
TMO	T_{m0} , see defining equations.
GAMO	γ_0 , see defining equations.
SA	a, see defining equations.
PC	p_{cut} or $-\sigma_f$ (default=-1.e+30)

VARIABLE	DESCRIPTION
SPALL	Spall type: EQ. 0.0: default set to "2.0", EQ. 1.0: $p \geq p_{\min}$, 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 μ coefficients for the cold compression energy fit. Default is η .
MMN	μ_{\min} or η_{\min} . Optional μ or η minimum value.
MMX	μ_{\max} or η_{\max} . Optional μ or η maximum value.
EC0,...EC9	Cold compression energy coefficients (optional).
UK	Activation energy for rate dependent model.
C1	Exponent prefactor in rate dependent model.
C2	Coefficient of drag term in rate dependent model.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				

<u>VARIABLE</u>	<u>DESCRIPTION</u>
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 (≤ 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 Δt below.
TREM	Δt for element removal: EQ.0.0: Δt is not considered (default), GT.0.0: element eroded if element time step size falls below Δt .

Remarks:

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

The element erosion option based on Δt must be used cautiously with the contact options. Nodes to surface contact is recommended with all nodes of the eroded brick elements included in the node list. As the elements are eroded the mass remains and continues to interact with the master surface.

***MAT_SOIL_AND_FOAM_FAILURE**

This is Material Type 14. The input for this model is the same as for *MATERIAL_SOIL_AND_FOAM (Type 5); however, when the pressure reaches the failure pressure, the element loses its ability to carry tension. It should be used only in situations when soils and foams are confined within a structure or when geometric boundaries are present.

***MAT_JOHNSON_COOK**

This is Material Type 15. The Johnson/Cook strain and temperature sensitive plasticity is sometimes used for problems where the strain rates vary over a large range and adiabatic temperature increases due to plastic heating cause material softening. When used with solid elements this model requires an equation-of-state. If thermal effects and damage are unimportant, the much less expensive *MAT_SIMPLIFIED_JOHNSON_COOK model is recommended. The simplified model can be used with beam elements.

Card 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	Strain rate normalization factor. This value depends on the time units. Input 1.0 for units of seconds, 0.001 for units of milliseconds, 0.000001 for microseconds, etc.
CP	Specific heat

<u>VARIABLE</u>	<u>DESCRIPTION</u>
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 option. This input applies to solid elements only since it is always necessary to iterate for the shell element plane stress condition. EQ. 0.0: no iterations (default), EQ. 1.0: accurate iterative solution for plastic strain. Much more expensive than default.
D1-D5	Failure parameters, see equations below.

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

$$\dot{\epsilon}^* = \frac{\dot{\bar{\epsilon}}^p}{\epsilon_0} \text{ effective plastic strain rate for } \epsilon_0 = 1 \text{ s}^{-1}$$

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

Constants for a variety of materials are provided in [Johnson and Cook 1983]. A fully viscoplastic formulation is optional (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 σ_y with sufficient accuracy to avoid iteration.

The strain at fracture is given by

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

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

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

Fracture occurs when the damage parameter

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

reaches the value of 1.

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

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

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

***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 20.5. Examples of converting conventional triaxial compression data to this type of model are found in (Desai and Siriwardane, 1984). Note that under conventional triaxial compression conditions, the LS-DYNA input corresponds to an ordinate of $\sigma_1 - \sigma_3$ rather than the more widely used $\frac{\sigma_1 - \sigma_3}{2}$, where σ_1 is the maximum principal stress and σ_3 is the minimum principal stress.

This material combined with equation-of-state type 9 (saturated) has been used very successfully to model ground shocks and soil-structure interactions at pressures up to 100kbars (approximately 1.5×10^6 psi).

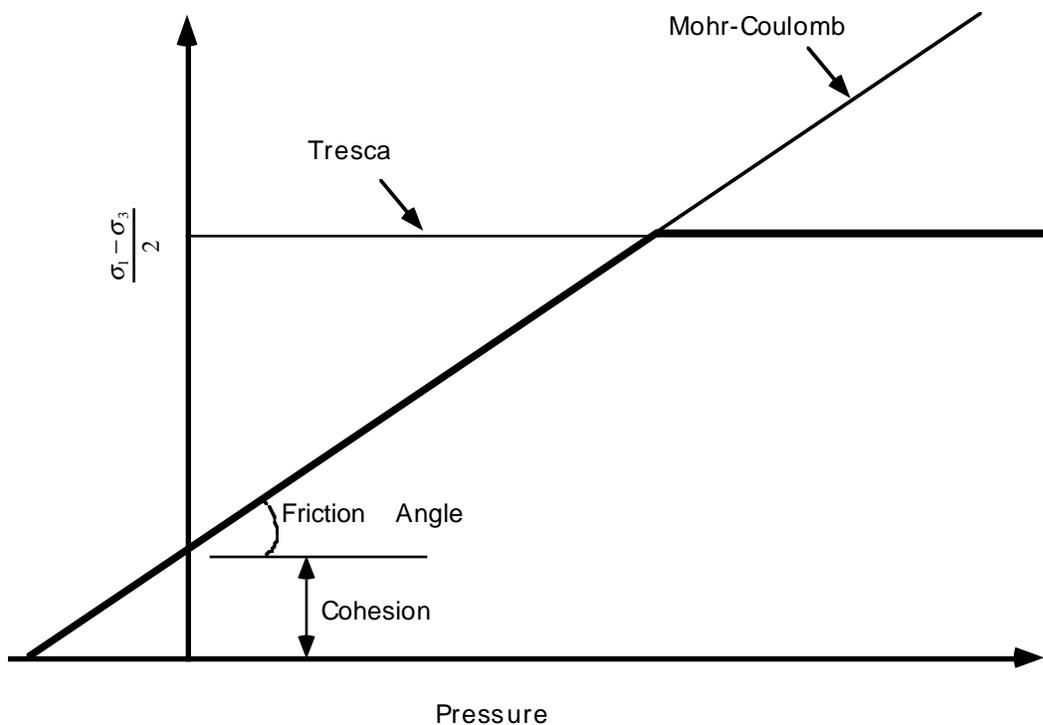


Figure 20.5. Mohr-Coulomb surface with a Tresca limit.

To invoke Mode I of this model, set a_0 , a_1 , a_2 , b_1 , a_{0f} , and a_{1f} to zero. The tabulated values of pressure should then be specified on cards 4 and 5, and the corresponding values of yield stress should be specified on cards 6 and 7. The parameters relating to reinforcement properties, initial yield stress, and tangent modulus are not used in this response mode, and should be set to zero.

Simple tensile failure

Note that a_{1f} is reset internally to $1/3$ even though it is input as zero; this defines a failed material curve of slope $3p$, 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 (σ_1) in the element exceeds the tensile cut-off (σ_{cut}). For every time step that $\sigma_1 > \sigma_{cut}$ the yield strength is

scaled back by a fraction of the distance between the two curves until after 20 time steps the yield strength is defined by the failed curve. The only way to inhibit this feature is to set σ_{cut} arbitrarily large.

Response Mode II. Two Curve Model with Damage and Failure

This approach uses two yield versus pressure curves of the form

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

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

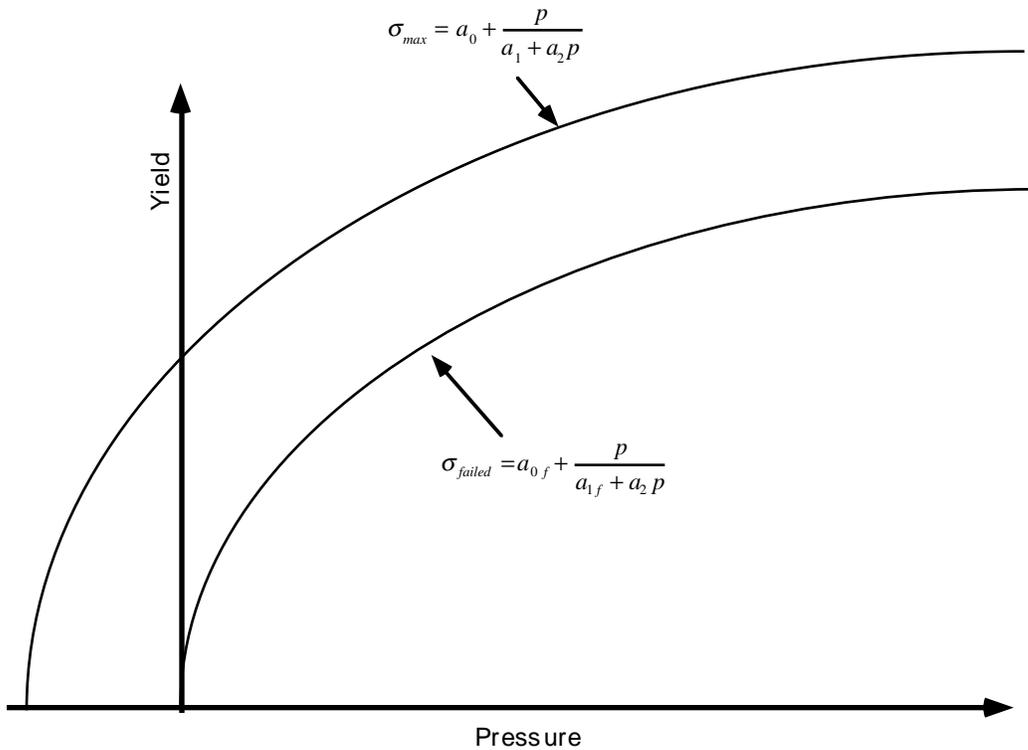


Figure 20.6. Two-curve concrete model with damage and failure.

MODE II. A: Simple tensile failure

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

Mode II.B: Tensile failure plus plastic strain scaling

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

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

where σ_{max} and σ_{failed} are found as shown in Figure 20.6. This yield strength is then subject to scaling for tensile failure as described above. This type of model allows the description of a strain hardening or softening material such as concrete.

Mode II.C: Tensile failure plus damage scaling

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

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

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

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

$$a_0 = \frac{f'_c}{4}$$

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

$$a_2 = \frac{1}{3f'_c}$$

$$a_{0f} = 0$$

$$a_{1f} = 0.385$$

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

<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 | 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, σ_y , is a function of the effective plastic strain, ϵ_{eff}^p , and the plastic hardening modulus, E_p :

$$\sigma_y = \sigma_0 + E_p \epsilon_{eff}^p$$

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, σ_f , the element fails on a plane perpendicular to the direction of the maximum principal stress. The normal stress and the two shear stresses on that plane are then reduced to zero. This stress reduction is done according to a delay function that reduces the stresses gradually to zero over a small number of time steps. This delay function procedure is used to reduce the ringing that may otherwise be introduced into the system by the sudden fracture.

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						

<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.
K	Strength coefficient.
N	Hardening exponent.
SRC	Strain rate parameter, C, if zero, rate effects are ignored.
SRP	Strain rate parameter, P, if zero, rate effects are ignored.

VARIABLE	DESCRIPTION
SIGY	Optional input parameter for defining the initial yield stress, σ_y . Generally, this parameter is not necessary and the strain to yield is calculated as described below. LT.0.02: $\epsilon_{yp} = SIGY$ 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, σ_y , is a function of plastic strain and obeys the equation:

$$\sigma_y = k \epsilon^n = k (\epsilon_{yp} + \bar{\epsilon}^p)^n$$

where ϵ_{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	

<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.

VARIABLE	DESCRIPTION
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 σ_0 as a function of the effective strain rate.
ETAN	Tangent modulus, E_t
LC2	Load curve ID defining Young's modulus as a function of the effective strain rate (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 σ_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, Δt_{max} . Generally, Δt_{max} goes down as the element becomes more distorted. To assure stability of time integration, the global LS-DYNA time step is the minimum of the Δt_{max} values calculated for all elements in the model. Using this option allows the selective deletion of elements whose time step Δt_{max} has fallen below the specified minimum time step, Δt_{crit} . Elements which are severely distorted often indicate that material has failed and supports little load, but these same elements may have very small time steps and therefore control the cost of the analysis. This option allows these highly distorted elements to be deleted from the calculation, and, therefore, the analysis can proceed at a larger time step, and, thus, at a reduced cost. Deleted elements do not carry any load, and are deleted from all applicable slide surface definitions. Clearly, this option must be judiciously used to obtain accurate results at a minimum cost.

A fully viscoplastic formulation is optional which incorporates the rate formulation within the yield surface. An additional cost is incurred but the improvement 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/RE
Type	I	F	F	F	F	F	F	C/F
Default	none	none	none	none	0	0	0	blank/none

Card 2

Variable	CMO	CON1	CON2					
Type	F	F	F					
Default	0	0	0					

Optional 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 5.4 (not CAL3D) coupling flag, n: EQ.0: use normal LS-DYNA rigid body updates, GT.0: the rigid body is coupled to MADYMO 5.4 ellipsoid number n, LT.0: the rigid body is coupled to MADYMO 5.4 plane number n .
COUPLE	Coupling option if applicable: EQ.-1: attach VDA surface in ALIAS (defined in the eighth field) and automatically generate a mesh for viewing the surface in LS-PREPOST. MADYMO 5.4 / CAL3D coupling option: EQ.0: the undeformed geometry input to LS-DYNA corresponds to the local system for MADYMO 5.4 / CAL3D. The finite element mesh is input, EQ.1: the undeformed geometry input to LS-DYNA corresponds to the global system for MADYMO 5.4 / CAL3D, EQ.2: generate a mesh for the ellipsoids and planes internally in LS-DYNA.
ALIAS	VDA surface alias name, see Appendix I.
RE	MADYMO 6.0.1 External Reference Number
CMO	Center of mass constraint option, CMO: EQ.+1.0: constraints applied in global directions, EQ. 0.0: no constraints, EQ. -1.0: constraints applied in local directions (SPC constraint).
CON1	First constraint parameter: <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. <u>If CMO=-1.0, then specify local coordinate system ID. See *DEFINE_COORDINATE_OPTION:</u> This coordinate system is fixed in time.

VARIABLE	DESCRIPTION
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 a and v, fixed in the rigid body which are used for output and the user defined airbag sensor subroutines. The output parameters are in the directions a, b, and c where the latter are given by the cross products $\mathbf{c}=\mathbf{a}\times\mathbf{v}$ and $\mathbf{b}=\mathbf{c}\times\mathbf{a}$. This input is optional.</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, ν , are used for determining sliding interface parameters if the rigid body interacts in a contact definition. Realistic values for these constants should be defined since unrealistic values may contribute to numerical problem in contact.

Constraint directions for rigid materials (CMO equal to +1 or -1) are fixed, that is, not updated, with time. To impose a constraint on a rigid body such that the constraint direction is updated as the rigid body rotates, use *BOUNDARY_PRESCRIBED_MOTION_RIGID_LOCAL.

If no constraints are specified for the rigid part (CMO=0) the nodes for the part are scanned to determine constraints on the part in global directions. If constraints are specified (CMO equal to +1 or -1) then the nodes are not scanned.

For coupling with MADYMO 5.4.1, only basic coupling is available.

The coupling flags (N and M) must match with SYSTEM and ELLIPSOID/PLANE in the MADYMO input file and the coupling option (COUPLE) must be defined.

For coupling with MADYMO 6.0.1, both basic and extended coupling are available:

- (1) Basic Coupling: The external reference number (RE) must match with the external reference number in the MADYMO XML input file. The coupling option (COUPLE) must be defined.
- (2) Extended Coupling: Under this option MADYMO will handle the contact between the MADYMO and LS-DYNA models. The external reference number (RE) and the coupling option (COUPLE) are not needed. All coupling surfaces that interface with the MADYMO models need to be defined in *CONTACT_COUPLING.

*MAT_ORTHOTROPIC_THERMAL

This is Material Type 21. A linearly elastic 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	ν_{ba} , Poisson's ratio, ba.
PRCA	ν_{ca} , Poisson's ratio, ca.
PRCB	ν_{cb} , Poisson's ratio, cb
GAB	G_{ab} , Shear modulus, ab.
GBC	G_{bc} , Shear modulus, bc.
GCA	G_{ca} , Shear modulus, ca.
AA	α_a , coefficients of thermal expansion in the a-direction.
AB	α_b , coefficients of thermal expansion in the b-direction.
AC	α_c , coefficients of thermal expansion in the c-direction.
AOPT	Material axes option (see <i>MAT_OPTION TROPIC_ELASTIC</i> for a more complete description): EQ. 0.0: locally orthotropic with material axes determined by element nodes 1, 2, and 4, as with <i>*DEFINE_COORDINATE_NODES</i> . EQ. 1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center; this is the a-direction. This option is for solid elements only. EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with <i>*DEFINE_COORDINATE_VECTOR</i> . EQ. 3.0: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle, BETA, from a line in the plane of the element defined by the cross product of the vector \mathbf{v} with the element normal. EQ. 4.0: locally orthotropic in cylindrical coordinate system with the material axes determined by a vector \mathbf{v} , and an originating point, P, which define the centerline axis. This option is for solid elements only.
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 \mathbf{v} for AOPT = 3.
D1,D2,D3	Components of vector \mathbf{d} for AOPT = 2.
BETA	Material angle in degrees for AOPT = 3, may be overridden on the element card, see *ELEMENT_SHELL_BETA or *ELEMENT_SOLID_ORTHO.
REF	Use reference geometry to initialize the stress tensor. The reference geometry is defined by the keyword:*INITIAL_FOAM_REFERENCE_GEOMETRY. 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 \mathbf{S} to the Green-St. Venant strain \mathbf{E} is

$$\mathbf{S} = \mathbf{C} \cdot \mathbf{E} = \mathbf{T}' \mathbf{C}_I \mathbf{T} \cdot \mathbf{E}$$

where \mathbf{T} is the transformation matrix [Cook 1974].

$$\mathbf{T} = \begin{bmatrix} l_1^2 & m_1^2 & n_1^2 & l_1 m_1 & m_1 n_1 & n_1 l_1 \\ l_2^2 & m_2^2 & n_2^2 & l_2 m_2 & m_2 n_2 & n_2 l_2 \\ l_3^2 & m_3^2 & n_3^2 & l_3 m_3 & m_3 n_3 & n_3 l_3 \\ 2l_1 l_2 & 2m_1 m_2 & 2n_1 n_2 & (l_1 m_2 + l_2 m_1) & (m_1 n_2 + m_2 n_1) & (n_1 l_2 + n_2 l_1) \\ 2l_2 l_3 & 2m_2 m_3 & 2n_2 n_3 & (l_2 m_3 + l_3 m_2) & (m_2 n_3 + m_3 n_2) & (n_2 l_3 + n_3 l_2) \\ 2l_3 l_1 & 2m_3 m_1 & 2n_3 n_1 & (l_3 m_1 + l_1 m_3) & (m_3 n_1 + m_1 n_3) & (n_3 l_1 + n_1 l_3) \end{bmatrix}$$

l_i, m_i, n_i are the direction cosines

$$\dot{x}_i = l_i x_1 + m_i x_2 + n_i x_3 \quad \text{for } i = 1, 2, 3$$

and \dot{x}_i denotes the material axes. The constitutive matrix \mathbf{C}_I 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

$$E^t = [E_{11}, E_{22}, E_{33}, E_{12}, E_{23}, E_{31},]$$

which include the local thermal strains which are integrated in time:

$$\epsilon_{aa}^{n+1} = \epsilon_{aa}^n + \alpha_a (T^{n+1} - T^n)$$

$$\epsilon_{bb}^{n+1} = \epsilon_{bb}^n + \alpha_b (T^{n+1} - T^n)$$

$$\epsilon_{cc}^{n+1} = \epsilon_{cc}^n + \alpha_c (T^{n+1} - T^n)$$

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 1987a,1987b]. Three failure criteria are possible, see Theoretical Manual. By using the user defined integration rule, see *INTEGRATION_SHELL, the constitutive constants can vary through the shell thickness. For all shells, except the DKT formulation, laminated shell theory can be activated to properly model the transverse shear deformation. Lamination theory is applied to correct for the assumption of a uniform constant shear strain through the thickness of the shell. For sandwich shells where the outer layers are much stiffer than the inner layers, the response will tend to be too stiff unless lamination theory is used. To turn on lamination theory see *CONTROL_SHELL.

Card 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							

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	BETA	
Type	F	F	F	F	F	F	F	
Default	0.0	0.0	0.0	0.0	0.0	0.0	0.0	

Card 5

Variable	SC	XT	YT	YC	ALPH	SN	SYZ	SZX
Type	F	F	F	F	F	F	F	F
Default	none							

<u>VARIABLE</u>	<u>DESCRIPTION</u>
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	ν_{ba} , Poisson ratio, ba.
PRCA	ν_{ca} , Poisson ratio, ca.
PRCB	ν_{cb} , Poisson ratio, cb.
GAB	G_{ab} , Shear modulus, ab.
GBC	G_{bc} , Shear modulus, bc.
GCA	G_{ca} , Shear modulus, ca.
KFAIL	Bulk modulus of failed material. Necessary for compressive failure.

VARIABLE	DESCRIPTION
AOPT	<p>Material axes option (see MAT_OPTION TROPIC_ELASTIC for a more complete description):</p> <p>EQ. 0.0: locally orthotropic with material axes determined by element nodes 1, 2, and 4, as with *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. This option is for solid elements only.</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: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle, BETA, from a line in the plane of the element defined by the cross product of the vector \mathbf{v} with the element normal.</p> <p>EQ. 4.0: locally orthotropic in cylindrical coordinate system with the material axes determined by a vector \mathbf{v}, and an originating point, P, which define the centerline axis. This option is for solid elements only.</p>
MACF	<p>Material axes change flag for brick elements:</p> <p>EQ.1.0: default,</p> <p>EQ.2.0: switch material axes a and b,</p> <p>EQ.3.0: switch material axes a and c.</p>
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.
BETA	Material angle in degrees for AOPT = 3, may be overridden on the element card, see *ELEMENT_SHELL_BETA or *ELEMENT_SOLID_ORTHO.
SC	Shear strength, ab plane, see 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
<i>ef(i)</i>	<i>tensile fiber mode</i>	<i>1 - elastic</i> <i>0 - failed</i>	81
<i>cm(i)</i>	<i>tensile matrix mode</i>		82
<i>ed(i)</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_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	BETA	
Type	F	F	F	F	F	F	F	

Define one set of constants on two cards for each temperature point. Up to 48 points (96 cards) can be defined. The next "*" card terminates the input.

Cards 1 for
Temperature
T_i

Variable	E _{Ai}	E _{Bi}	E _{Ci}	PRB _{Ai}	PRC _{Ai}	PRC _{Bi}		
Type	F	F	F	F	F	F		

Cards 2 for
Temperature
Ti

Variable	AAi	ABi	ACi	GABi	GBCi	GCAi	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 (see <i>MAT_OPTION TROPIC_ELASTIC</i> for a more complete description):</p> <p>EQ. 0.0: locally orthotropic with material axes determined by element nodes 1, 2, and 4, as with <i>*DEFINE_COORDINATE_NODES</i>.</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. This option is for solid elements only.</p> <p>EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with <i>*DEFINE_COORDINATE_VECTOR</i>.</p> <p>EQ. 3.0: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle, <i>BETA</i>, from a line in the plane of the element defined by the cross product of the vector \mathbf{v} with the element normal.</p> <p>EQ. 4.0: locally orthotropic in cylindrical coordinate system with the material axes determined by a vector \mathbf{v}, and an originating point, <i>P</i>, which define the centerline axis. This option is for solid elements only.</p>
REF	<p>Use reference geometry to initialize the stress tensor. The reference geometriy is defined by the keyword: <i>*INITIAL_FOAM_REFERENCE_GEOMETRY</i>. 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 \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.
BETA	Material angle in degrees for AOPT = 3, may be overridden on the element card, see <i>*ELEMENT_SHELL_BETA</i> or <i>*ELEMENT_SOLID_ORTHO</i> .

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	ν_{ba} , Poisson's ratio ba at temperature T_i .
PRCAi	ν_{ca} , Poisson's ratio ca at temperature T_i .
PRCBI	ν_{cb} , Poisson's ratio cb at temperature T_i .
AAi	α_a , coefficient of thermal expansion in a-direction at temperature T_i .
ABi	α_b , coefficient of thermal expansion in b-direction at temperature T_i .
ACi	α_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 .
T_i	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].

$$T = \begin{bmatrix} l_1^2 & m_1^2 & n_1^2 & l_1 m_1 & m_1 n_1 & n_1 l_1 \\ l_2^2 & m_2^2 & n_2^2 & l_2 m_2 & m_2 n_2 & n_2 l_2 \\ l_3^2 & m_3^2 & n_3^2 & l_3 m_3 & m_3 n_3 & n_3 l_3 \\ 2l_1 l_2 & 2m_1 m_2 & 2n_1 n_2 & (l_3 m_2 + l_1 m_1) & (m_1 n_2 + m_2 n_1) & (n_1 l_2 + n_2 l_1) \\ 2l_2 l_3 & 2m_2 m_3 & 2n_2 n_3 & (l_2 m_3 + l_3 m_2) & (m_2 n_3 + m_3 n_2) & (n_2 l_3 + n_3 l_2) \\ 2l_3 l_1 & 2m_3 m_1 & 2n_3 n_1 & (l_3 m_1 + l_1 m_3) & (m_3 n_1 + m_1 n_3) & (n_3 l_1 + n_1 l_3) \end{bmatrix}$$

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. If considering laminated or sandwich shells with nonuniform material properties (this is defined through the user specified integration rule), the model, MAT_LAYERED_LINEAR_PLASTICITY, is recommended. If solid elements are used and if the elastic strains before yielding are finite, the model, MAT_FINITE_ELASTIC_STRAIN_PLASTICITY, treats the elastic strains using a hyperelastic formulation.

Card 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;

<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: EQ.-1.0: Cowper-Symonds with deviatoric strain rate rather than total. EQ. 0.0: Scale yield stress (default), EQ. 1.0: Viscoplastic formulation.
EPS1-EPS8	Effective plastic strain values (optional 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 VP=-1, the deviatoric strain rates are used instead.

If the viscoplastic option is active, VP=1.0, and if SIGY is > 0 then the dynamic yield stress is computed from the sum of the static stress, $\sigma_y^s(\epsilon_{eff}^p)$, 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.

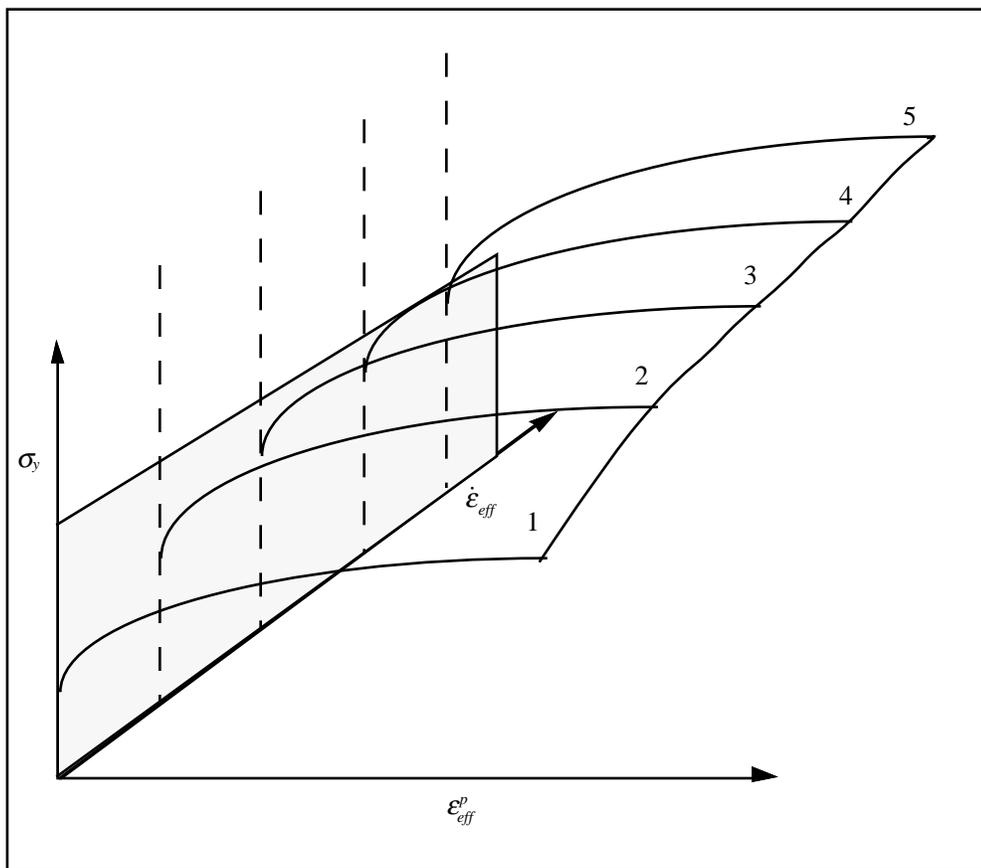


Figure 20.7. Rate effects may be accounted for by defining a table of curves. If a table ID is specified a curve ID is given for each strain rate, see *DEFINE_TABLE. Intermediate values are found by interpolating between curves. Effective plastic strain versus yield stress is expected. If the strain rate values fall out of range, extrapolation is not used; rather, either the first or last curve determines the yield stress depending on whether the rate is low or high, respectively.

***MAT_GEOLOGIC_CAP_MODEL**

This is Material Type 25. This 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, α .

VARIABLE	DESCRIPTION
THETA	Failure envelope linear coefficient, θ .
GAMMA	Failure envelope exponential coefficient, γ .
BETA	Failure envelope exponent, β .
R	Cap, surface axis ratio.
D	Hardening law exponent.
W	Hardening law coefficient.
X0	Hardening law exponent, X_0 .
C	Kinematic hardening coefficient, \bar{c} .
N	Kinematic hardening parameter.
PLOT	Save the following variable for plotting in TAURUS, to be labeled there as "effective plastic strain:" EQ.1: hardening parameter, κ , EQ.2: cap $-J_1$ axis intercept, $X(\kappa)$, EQ.3: volumetric plastic strain ε_V^p , EQ.4: first stress invariant, J_1 , EQ.5: second stress invariant, $\sqrt{J_2}$, EQ.6: not used, EQ.7: not used, EQ.8: response mode number, EQ.9: number of iterations.
FTYPE	Formulation flag: EQ.1: 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.

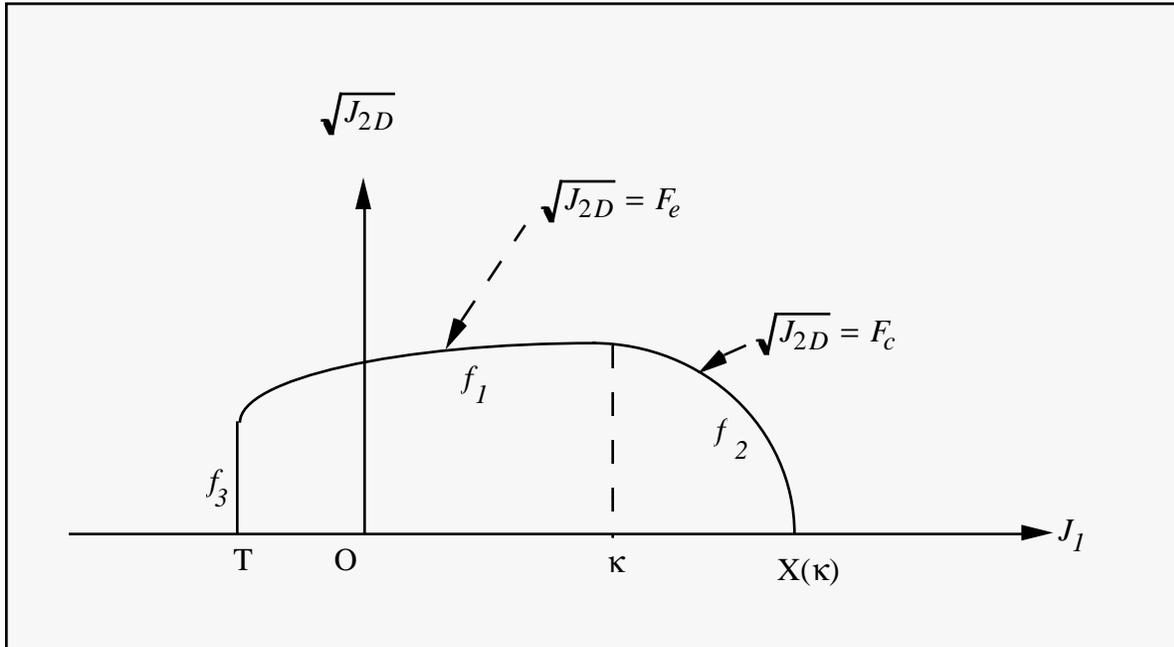


Figure 20.8. The yield surface of the two-invariant cap model in pressure $\sqrt{J_{2D}} - J_1$ space. Surface f_1 is the failure envelope, f_2 is the cap surface, and f_3 is the tension cutoff.

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

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

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

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

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

$$f_3 \equiv T - J_1,$$

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

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

$$\boldsymbol{\varepsilon} = \boldsymbol{\varepsilon}^e + \boldsymbol{\varepsilon}^p,$$

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

$$\boldsymbol{\sigma} = \mathbf{C}(\boldsymbol{\varepsilon} - \boldsymbol{\varepsilon}^p),$$

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

The yield condition may be written

$$\begin{aligned}
 f_1(s) &\leq 0 \\
 f_2(s, \kappa) &\leq 0 \\
 f_3(s) &\leq 0
 \end{aligned}$$

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

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

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

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-Columb models. However, the hardening law permits the cap surface to contract until the cap intersects the failure envelope at the stress point, and the cap remains at that point. The local normal to the yield surface is now vertical, and therefore the normality rule assures that no further plastic volumetric strain (dilatency) is created. Adjustment of the parameters that control the rate of cap contractions permits experimentally observed amounts of dilatency to be incorporated into the cap model, thus producing a constitutive law which better represents the physics to be modeled.

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

The inclusion of kinematic hardening results in hysteretic energy dissipation under cyclic loading conditions. Following the approach of Isenberg, et. al. [1978] a nonlinear kinematic hardening law is used for the failure envelope surface when nonzero values of α 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, α . The formulation including kinematic hardening is obtained by replacing the stress σ with the translated stress tensor $\eta \equiv \sigma - \alpha$ in all of the above equation. The history tensor α 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 σ and α 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 distant between the outside of the initial yield surface and the inside of the limit surface. In order for the limit surface of the kinematic hardening cap model to correspond with the failure envelope surface of the standard cap model, the scalar parameter a must be replaced $\alpha - N$ 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 α , β , θ , and γ are usually evaluated by fitting a curve through failure data taken from a set of triaxial compression tests. The parameters W, D, and X_0 define the cap hardening law. The value W represent the void fraction of the uncompressed sample and D governs the slope of the initial loading curve in hydrostatic compression. The value of R is the ration of major to minor axes of the quarter ellipse defining the cap surface. Additional details and guidelines for fitting the cap model to experimental data 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	μ , material viscosity coefficient. (default=.05) Recommended.
BULK	Bulk viscosity flag: EQ.0.0: bulk viscosity is not used. This is recommended. EQ.1.0: bulk viscosity is active and $\mu=0$ 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.

VARIABLE	DESCRIPTION
LCBC	Load curve ID, see *DEFINE_CURVE, for sigma-bc versus either relative volume or volumetric strain. Default LCBC=LCS. See notes below.
LCCA	Load curve ID, see *DEFINE_CURVE, or sigma-ca versus either relative volume or volumetric strain. Default LCCA=LCS. See notes below.
LCSR	Load curve ID, see *DEFINE_CURVE, for strain-rate effects defining the scale factor versus strain rate. This is optional. The curves defined above are scaled using this curve.
EAAU	Elastic modulus E_{aaU} in uncompressed configuration.
EBBU	Elastic modulus E_{bbU} in uncompressed configuration.
ECCU	Elastic modulus E_{ccU} in uncompressed configuration.
GABU	Shear modulus G_{abU} in uncompressed configuration.
GBCU	Shear modulus G_{bcU} in uncompressed configuration.
GCAU	Shear modulus G_{caU} in uncompressed configuration.
AOPT	Material axes option (see MAT_OPTION TROPIC_ELASTIC for a more complete description): EQ. 0.0: locally orthotropic with material axes determined by element nodes 1, 2, and 4, as with *DEFINE_COORDINATE_NODES. EQ. 1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center; this is the a-direction. EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR.
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 :

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

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

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

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

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

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

where

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

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 μ (MU) should be set to a small number (usually .02-.10 is okay). Alternatively, the two bulk viscosity coefficients on the control cards should be set to very small numbers to prevent the development of spurious pressures that may lead to undesirable and confusing results. The latter is not recommended since spurious numerical noise may develop.

The load curves define the magnitude of the average stress as the material changes density (relative volume), see Figure 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:

$$\epsilon_v = 1 - V$$

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:

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

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

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

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

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

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

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 λ is either unity or a value taken from the load curve number, LCSR, that defines λ as a function of strain-rate. Strain-rate is defined here as the Euclidean norm of the deviatoric strain-rate tensor.

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

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

where the deviatoric strain increment is defined as

$$\Delta \epsilon_{ij}^{dev} = \Delta \epsilon_{ij} - \frac{1}{3} \Delta \epsilon_{kk} \delta_{ij} .$$

Now a check is made to see if the yield stress for the fully compacted material is exceeded by comparing

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

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

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

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

to obtain the final value for the Cauchy stress

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

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

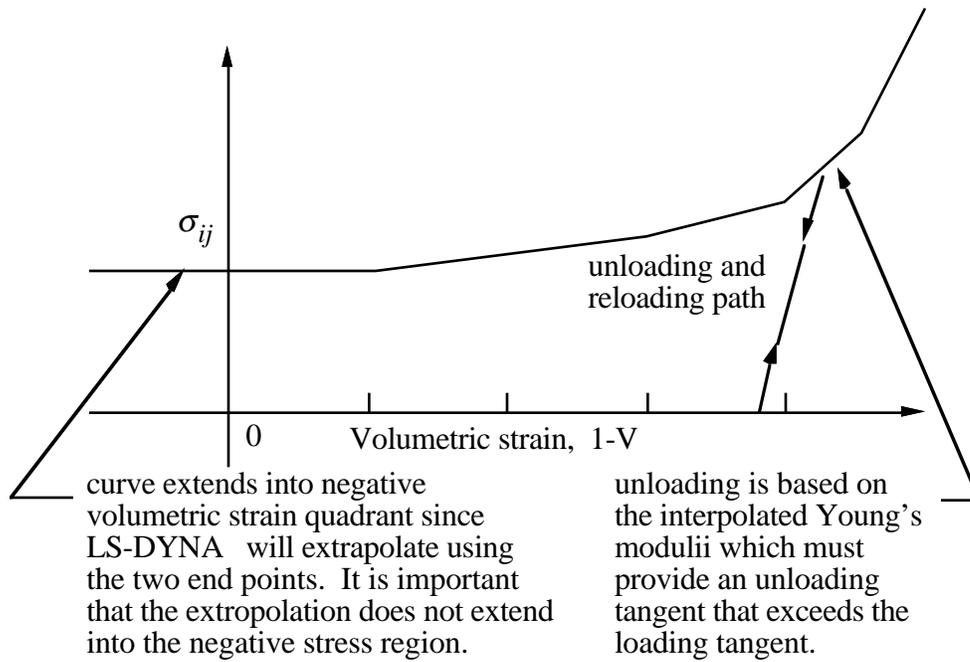


Figure 20.9. Stress quantity versus volumetric strain. Note that the “yield stress” at a volumetric strain of zero is non-zero. In the load curve definition, see *DEFINE_CURVE, the “time” value is the volumetric strain and the “function” value is the yield stress.

*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 (value between 0.49 and 0.5 is recommended, smaller values may not work).
A	Constant, see literature and equations defined below.
B	Constant, see literature and equations defined below.
REF	Use reference geometry to initialize the stress tensor. The reference 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.

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.

<u>VARIABLE</u>	<u>DESCRIPTION</u>
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:

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

where

$$C = 0.5 A + B$$

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

ν = Poisson's ratio

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

I, II, III = invariants of right Cauchy-Green Tensor C .

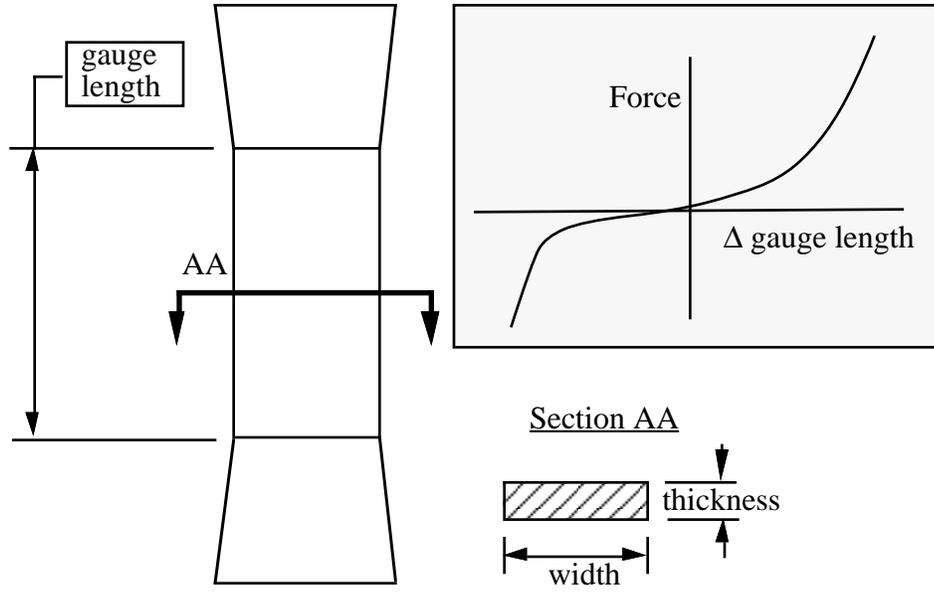
The load curve definition that provides the uniaxial data should give the change in gauge length, ΔL, versus the corresponding force. In compression both the force and the change in gauge length must be specified as negative values. In tension the force and change in gauge length should be input as positive values. The principal stretch ratio in the uniaxial direction, λ₁, is then given by

$$\lambda_1 = \frac{L_o + \Delta L}{L_o}$$

with L₀ 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.

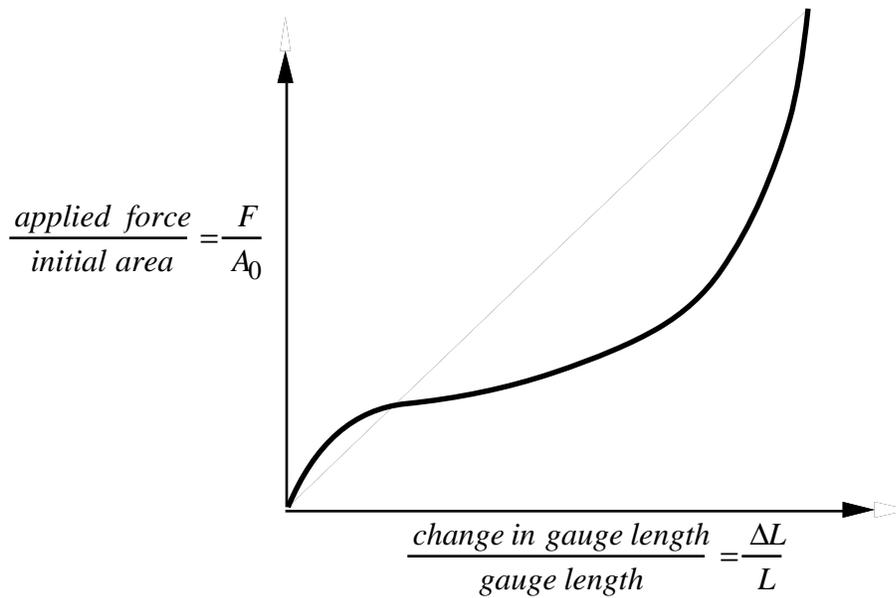


Figure 20.11 The stress versus strain curve can be used instead of the force versus the change in the gauge length by setting the gauge length, thickness, and width to unity (1.0) and defining the engineering strain in place of the change in gauge length and the nominal (engineering) stress in place 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^o triangular shell, the Belytschko-Tsay shell, and the fully integrated type 16 shell. For beams, the treatment is elastic-perfectly plastic, but for shell elements isotropic hardening is approximately modeled. For a detailed description we refer to the Theoretical Manual. Since the stresses are not computed in the resultant formulation, the stresses output to the binary databases for the resultant elements are zero.

Card 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		

<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	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					

<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
DF	Damping factor, see definition in notes below. A proper control for the timestep has to be maintained by the user!

<u>VARIABLE</u>	<u>DESCRIPTION</u>
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 (collapse load) versus strain/change in length (see AOPT) for the corresponding applied end moment. Define the same number as end moments. Each curve must contain the same number of points.
LPS1	Load curve ID for plastic moment versus rotation about s-axis at node 1. If zero, this load curve is ignored.
SFS1	Scale factor for plastic moment versus rotation curve about s-axis at node 1. Default = 1.0.
LPS2	Load curve ID for plastic moment versus rotation about s-axis at node 2. Default: is same as at node 1.
SFS2	Scale factor for plastic moment versus rotation curve about s-axis at node 2. Default: is same as at node 1.
YMS1	Yield moment about s-axis at node 1 for interaction calculations (default set to 1.0E+20 to prevent interaction).
YMS2	Yield moment about s-axis at node 2 for interaction calculations (default set to YMS1).
LPT1	Load curve ID for plastic moment versus rotation about t-axis at node 1. If zero, this load curve is ignored.
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.

VARIABLE	DESCRIPTION
YMT1	Yield moment about t-axis at node 1 for interaction calculations (default set to 1.0E+20 to prevent interactions)
YMT2	Yield moment about t-axis at node 2 for interaction calculations (default set to YMT1)
LPR	Load curve ID for plastic torsional moment versus rotation. If zero, this load curve is ignored.
SFR	Scale factor for plastic torsional moment versus rotation (default = 1.0).
YMR	Torsional yield moment for interaction calculations (default set to 1.0E+20 to prevent interaction)

Remarks:

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

Axial collapse occurs when the compressive axial load reaches the collapse load. Collapse load versus collapse deflection is specified in the form of a load curve. The points of the load curve are either (true strain, collapse force) or (change in length, collapse force). Both quantities should be entered as positive for all points, and will be interpreted as compressive. The first point should be (zero, initial collapse load).

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

Stiffness-proportional damping may be added using the damping factor λ . This is defined as follows:

$$\lambda = \frac{2 * \xi}{\omega}$$

where ξ is the damping factor at the reference frequency ω (in radians per second). For example if 1% damping at 2Hz is required

$$\lambda = \frac{2 * 0.01}{2\pi * 2} = 0.001592$$

If damping is used, a small timestep may be required. LS-DYNA does not check this so to avoid instability it may be necessary to control the timestep via a load curve. As a guide, the timestep required for any given element is multiplied by $0.3L/c\lambda$ when damping is present (L = element length, c = sound speed).

Moment Interaction:

Plastic hinges can form due to the combined action of moments about the three axes. This facility is activated only when yield moments are defined in the material input. A hinge forms when the following condition is first satisfied.

$$\left(\frac{M_r}{M_{ryield}}\right)^2 + \left(\frac{M_s}{M_{syield}}\right)^2 + \left(\frac{M_t}{M_{tyield}}\right)^2 \geq 1$$

where,

M_r, M_s, M_t = current moment

$M_{ryield}, M_{syield}, M_{tyield}$ = yield moment

Note that scale factors for hinge behavior defined in the input will also be applied to the yield moments: for example, M_{syield} in the above formula is given by the input yield moment about the local axis times the input scale factor for the local s axis. For strain-softening characteristics, the yield moment should generally be set equal to the initial peak of the moment-rotation load curve.

On forming a hinge, upper limit moments are set. These are given by

$$M_{rupper} = \text{MAX}\left(M_r, \frac{M_{ryield}}{2}\right)$$

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.

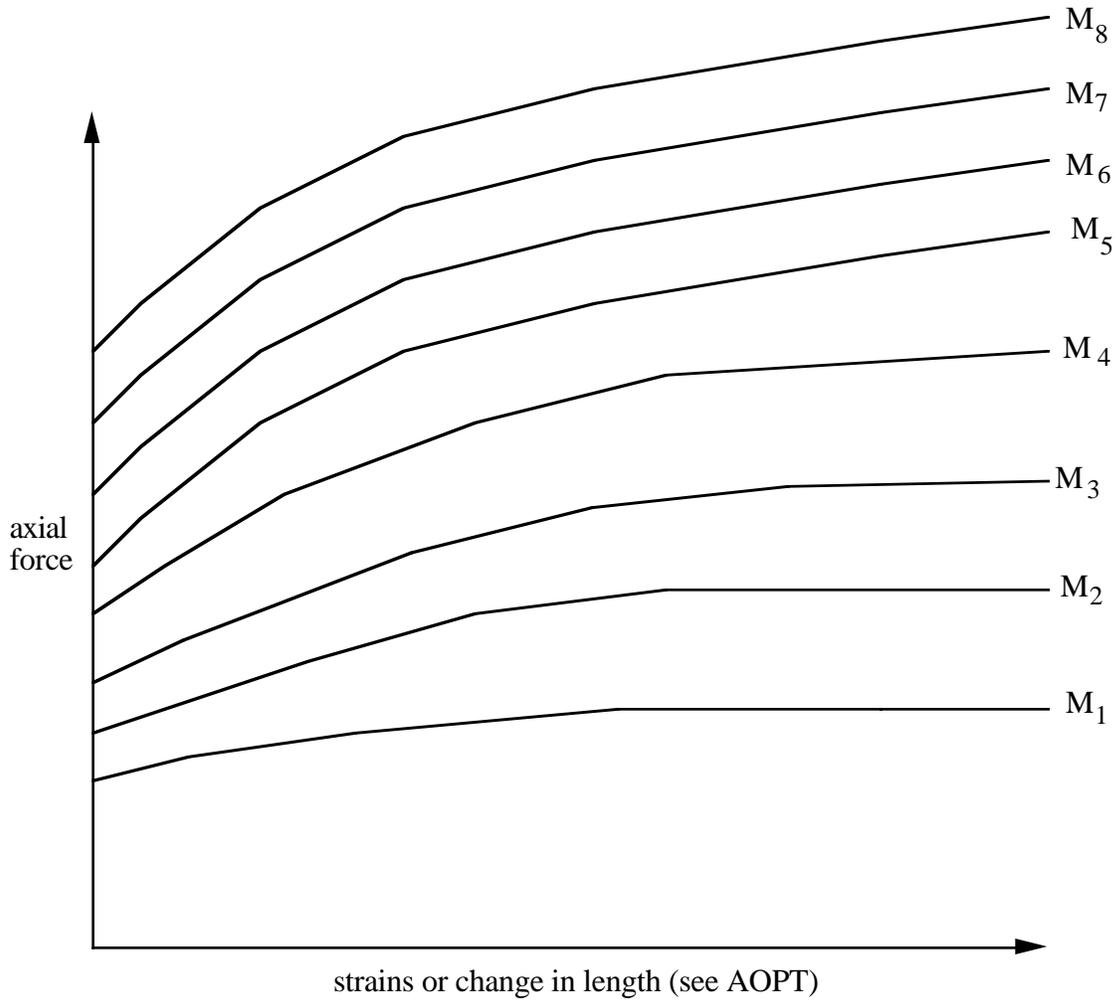


Figure 20.12. The force magnitude is limited by the applied end moment. For an intermediate value of the end moment LS-DYNA interpolates between the curves to determine the allowable force value.

*MAT_SHAPE_MEMORY

This is material type 30. This material model describes the superelastic response present in shape-memory alloys (SMA), that is the peculiar material ability to undergo large deformations with a full recovery in loading-unloading cycles (See Figure 20.13). The material response is always characterized by a hysteresis loop. See the references by [Auricchio, Taylor and Lubliner, 1997] and [Auricchio and Taylor, 1997].

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	SIG_ASS	SIG_ASF	SIG_SAS	SIG_SAF	EPSL	ALPHA	YMRT	
Type	F	F	F	F	F	F	F	
Default	none	none	none	none	none	0.0	0.0	

VARIABLE

DESCRIPTION

MID Material identification

RO Density

E Young's modulus

PR Poisson's ratio

SIG_ASS Starting value for the forward phase transformation (conversion of austenite into martensite) in the case of a uniaxial tensile state of stress. A load curve for SIG_ASS as a function of temperature is specified by using the negative of the load curve ID number.

VARIABLE	DESCRIPTION
SIG_ASF	Final value for the forward phase transformation (conversion of austenite into martensite) in the case of a uniaxial tensile state of stress. SIG_ASF as a function of temperature is specified by using the negative of the load curve ID number.
SIG_SAS	Starting value for the reverse phase transformation (conversion of martensite into austenite) in the case of a uniaxial tensile state of stress. SIG_SAS as a function of temperature is specified by using the negative of the load curve ID number.
SIG_SAF	Final value for the reverse phase transformation (conversion of martensite into austenite) in the case of a uniaxial tensile state of stress. SIG_SAF as a function of temperature is specified by using the negative of the load curve ID number.
EPSL	Recoverable strain or maximum residual strain. It is a measure of the maximum deformation obtainable all the martensite in one direction.
ALPHA	Parameter measuring the difference between material responses in tension and compression (set alpha = 0 for no difference). Also, see the following Remark.
YMRT	Young's modulus for the martensite if it is different from the modulus for the austenite. Defaults to the austenite modulus if it is set to zero.

Remarks:

The material parameter alpha, α , measures the difference between material responses in tension and compression. In particular, it is possible to relate the parameter α to the initial stress value of the austenite into martensite conversion, indicated respectively as $\sigma_s^{AS,+}$ and $\sigma_s^{AS,-}$, according to the following expression:

$$\alpha = \frac{\sigma_s^{AS,-} - \sigma_s^{AS,+}}{\sigma_s^{AS,-} + \sigma_s^{AS,+}}$$

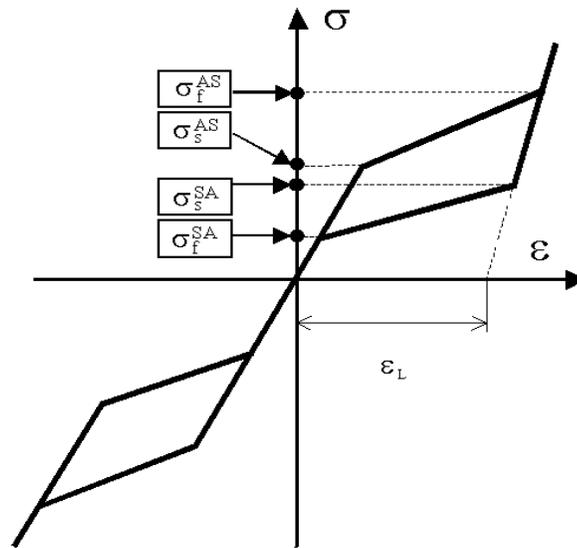


Figure 20.13: Pictorial representation of superelastic behavior for a shape-memory material.

In the following, the results obtained from a simple test problem is reported. The material properties are set as:

E	60000 MPa
nu	0.3
sig_AS_s	520 MPa
sig_AS_f	600 MPa
sig_SA_s	300 MPa
sig_SA_f	200 MPa
epsL	0.07
alpha	0.12
ymrt	50000 MPa

The investigated problem is the complete loading-unloading test in tension and compression. The uniaxial Cauchy stress versus the logarithmic strain is plotted in Figure 20.14.

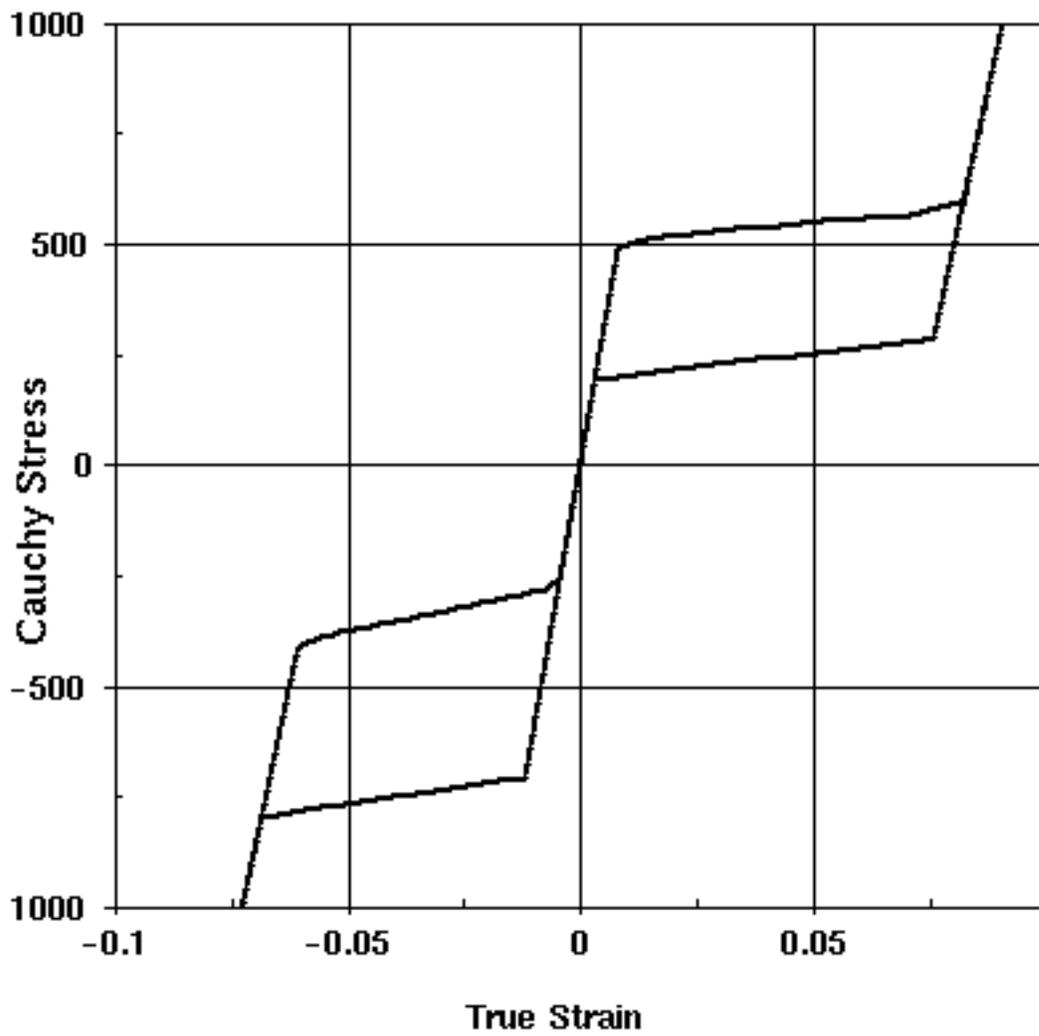


Figure 20.14. Complete loading-unloading test in tension and compression

*MAT_FRAZER_NASH_RUBBER_MODEL

This is Material Type 31. This model defines rubber from uniaxial test data. It is a modified form of the hyperelastic constitutive law first described in [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				

VARIABLE

DESCRIPTION

- 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 C₁₀₀ (EQ.1.0 if term is in the least squares fit.).
- C200 C₂₀₀ (EQ.1.0 if term is in the least squares fit.).

VARIABLE	DESCRIPTION
C300	C ₃₀₀ (EQ.1.0 if term is in the least squares fit.).
C400	C ₄₀₀ (EQ.1.0 if term is in the least squares fit.).
C110	C ₁₁₀ (EQ.1.0 if term is in the least squares fit.).
C210	C ₂₁₀ (EQ.1.0 if term is in the least squares fit.).
C010	C ₀₁₀ (EQ.1.0 if term is in the least squares fit.).
C020	C ₀₂₀ (EQ.1.0 if term is in the least squares fit.).
EXIT	Exit option: EQ. 0.0: stop if strain limits are exceeded (recommended), NE. 0.0: continue if strain limits are exceeded. The curve is then extrapolated.
EMAX	Maximum strain limit, (Green-St, Venant Strain).
EMIN	Minimum strain limit, (Green-St, Venant Strain).
REF	Use reference geometry to initialize the stress tensor. The reference 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, ΔL , and the corresponding force . In compression both the force and the change in gauge length must be specified as negative values. In tension the force and change in gauge length should be input as positive values. The principal stretch ratio in the uniaxial direction, λ_1 , is then given by

$$\lambda_1 = \frac{L_o + \Delta L}{L_o}$$

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	LCID	
Type	F	F	F	F	F	F	F	

Card 3

Variable	AOPT	OFFANG						
Type	F	F						

Card 4

Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card 5

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, ν .
K	k, strength coefficient, see notes below.
EO	ϵ_0 , strain corresponding to the initial yield, see notes below.
N	n, hardening exponent for yield strength.
M	m, flow potential exponent in Barlat's Model.
A	a, anisotropy coefficient in Barlat's Model.
B	b, anisotropy coefficient in Barlat's Model.
C	c anisotropy coefficient in Barlat's Model.
F	f, anisotropy coefficient in Barlat's Model.
G	g, anisotropy coefficient in Barlat's Model.
H	h, anisotropy coefficient in Barlat's Model.
LCID	Option load curve ID defining effective stress versus effective plastic strain. If nonzero, this curve will be used to define the yield stress.
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: locally orthotropic material axes determined by offsetting the material axes by an angle, OFFANG, from a line determined by taking the cross product of the vector \mathbf{v} with the normal to the plane of a shell element or midsurface of a brick.
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 Φ is defined as:

$$\Phi = |S_1 - S_2|^m + |S_2 - S_3|^m + |S_3 - S_1|^m = 2\bar{\sigma}^m$$

where $\bar{\sigma}$ is the effective stress and $S_{i=1,2,3}$ are the principal values of the symmetric matrix $S_{\alpha\beta}$,

$$S_{xx} = [c(\sigma_{xx} - \sigma_{yy}) - b(\sigma_{zz} - \sigma_{xx})]/3$$

$$S_{yy} = [a(\sigma_{yy} - \sigma_{zz}) - c(\sigma_{xx} - \sigma_{yy})]/3$$

$$S_{zz} = [b(\sigma_{zz} - \sigma_{xx}) - a(\sigma_{yy} - \sigma_{zz})]/3$$

$$S_{yz} = f\sigma_{yz}$$

$$S_{zx} = g\sigma_{zx}$$

$$S_{xy} = h\sigma_{xy}$$

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 ϵ_0 is the strain corresponding to the initial yield stress and ϵ^p is the plastic strain.

*MAT_BARLAT_YLD96

This is Material Type 33. This model was developed by Barlat, Maeda, Chung, Yanagawa, Brem, Hayashida, Lege, Matsui, Murtha, Hattori, Becker, and Makosey [1997] for modeling anisotropic material behavior in forming processes in particular for aluminum alloys. This model is available for shell elements only.

Card 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				A1	A2	A3		
Type				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, ν .
K	k , strength coefficient or a in Voce, see notes below.
EO	ϵ_0 , strain corresponding to the initial yield or b in Voce, see notes below.
N	n , hardening exponent for yield strength or c in Voce.
ESR0	ϵ_{SR0} , in powerlaw rate sensitivity.
M	m , exponent for strain rate effects
HARD	Hardening option: LT. 0.0: absolute value defines the load curve ID. EQ. 1.0: powerlaw EQ. 2.0: Voce
A	Flow potential exponent.
C1	$c1$, see equations below.
C2	$c2$, see equations below.

VARIABLE	DESCRIPTION
C3	c3, see equations below.
C4	c4, see equations below.
AX	ax, see equations below.
AY	ay, see equations below.
AZ0	az0, see equations below.
AZ1	az1, see equations below.
AOPT	Material axes option: EQ. 0.0: locally orthotropic with material axes determined by element nodes as shown in Figure 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. 2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR. EQ. 3.0: locally orthotropic material axes determined by offsetting the material axes by an angle, OFFANG, from a line determined by taking the cross product of the vector \mathbf{v} with the normal to the plane of the element.
OFFANG	Offset angle for AOPT = 3.
A1 A2 A3	Components of vector a for AOPT = 2.
V1 V2 V3	Components of vector v for AOPT = 3.
D1 D2 D3	Components of vector d for AOPT = 2.

Remarks:

The yield stress σ_y is defined three ways. The first, the Swift equation, is given in terms of the input constants as:

$$\sigma_y = k(\varepsilon_0 + \varepsilon^p)^n \left(\frac{\dot{\varepsilon}}{\varepsilon_{SR0}} \right)^m$$

The second, the Voce equation, is defined as:

$$\sigma_y = a - be^{-c\varepsilon^p}$$

and the third option is to give a load curve ID that defines the yield stress as a function of effective plastic strain. The yield function Φ is defined as:

$$\Phi = \alpha_1 |s_1 - s_2|^a + \alpha_2 |s_2 - s_3|^a + \alpha_3 |s_3 - s_1|^a = 2\sigma_y^a$$

where s_i is a principle component of the deviatoric stress tensor where in vector notation:

$$\underset{\sim}{s} = \underset{\sim}{L} \underset{\sim}{\sigma}$$

and $\underset{\sim}{L}$ is given as

$$\underset{\sim}{L} = \begin{bmatrix} \frac{c_1 + c_3}{3} & \frac{-c_3}{3} & \frac{-c_2}{3} & 0 \\ \frac{-c_3}{3} & \frac{c_3 + c_1}{3} & \frac{-c_2}{3} & 0 \\ \frac{-c_2}{3} & \frac{-c_2}{3} & \frac{c_1 + c_2}{3} & 0 \\ 0 & 0 & 0 & c_4 \end{bmatrix}$$

A coordinate transformation relates the material frame to the principle directions of $\underset{\sim}{s}$ is used to obtain the α_k coefficients consistent with the rotated principle axes:

$$\alpha_k = \alpha_x p_{1k}^2 + \alpha_y p_{2k}^2 + \alpha_z p_{3k}^2$$

$$\alpha_z = \alpha_{z0} \cos^2 2\beta + \alpha_{z1} \sin^2 2\beta$$

where p_{ij} are components of the transformation matrix. The angle β defines a measure of the rotation between the frame of the principal value of $\underset{\sim}{s}$ and the principal anisotropy axes.

*MAT_FABRIC

This is Material Type 34. This material is especially developed for airbag materials. The fabric model is a variation on the layered orthotropic composite model of material 22 and is valid for 3 and 4 node membrane elements only. In addition to being a constitutive model, this model also invokes a special membrane element formulation which is more suited to the deformation experienced by fabrics under large deformation. For thin fabrics, buckling can result in an inability to support compressive stresses; thus a flag is included for this option. A linearly elastic liner is also included which can be used to reduce the tendency for these elements to be crushed when the no-compression option is invoked. In LS-DYNA versions after 931 the isotropic elastic option is available.

Card 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	FVOPT	TSRFAC
Type	F	F	F	F	F	F	F	F
Remarks		3	3		4	0	0	0

Card 4

Variable				A1	A2	A3		
Type				F	F	F		

Card 5

Variable	V1	V2	V3	D1	D2	D3	BETA	
Type	F	F	F	F	F	F	F	

Define if and only if FORM=4.

Card 6

Variable	LCA	LCB	LCAB	LCUA	LCUB	LCUAB		
Type	I	I	I	I	I	I		

<u>VARIABLE</u>	<u>DESCRIPTION</u>
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	ν_{ba} , Poisson's ratio ba direction.
PRCA	ν_{ca} , Poisson's ratio ca direction, set to zero for isotropic elastic material.

VARIABLE	DESCRIPTION
PRCB	ν_{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.
CSE	Compressive stress elimination option (default 0.0): EQ.0.0: don't eliminate compressive stresses, EQ.1.0: eliminate compressive stresses (This option does not apply to the liner).
EL	Young's modulus for elastic liner (optional).
PRL	Poisson's ratio for elastic liner (optional).
LRATIO	Ratio of liner thickness to total fabric thickness.
DAMP	Rayleigh damping coefficient. A 0.05 coefficient is recommended corresponding to 5% of critical damping. Sometimes larger values are necessary.
AOPT	Material axes option (see MAT_OPTION TROPIC_ELASTIC for a more complete description): EQ. 0.0: locally orthotropic with material axes determined by element nodes 1, 2, and 4, as with *DEFINE_COORDINATE_NODES. EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR. EQ. 3.0: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle, BETA, from a line in the plane of the element defined by the cross product of the vector \mathbf{v} with the element normal.
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.

VARIABLE	DESCRIPTION
LNRC	Flag to turn off compression in liner until the reference geometry is reached, i.e., the fabric element becomes tensile. EQ.0.0: off. EQ.1.0: on.
FORM	Flag to modify membrane formulation for fabric material: EQ.0.0:default. Least costly and very reliable. EQ.1.0:invariant local membrane coordinate system EQ.2.0:Green-Lagrange strain formulation EQ.3.0:large strain with nonorthogonal material angles. See Remark 5. EQ.4.0:large strain with nonorthogonal material angles and nonlinear stress strain behavior. Define optional load curve IDs on optional card.
FVOPT	Fabric venting option. EQ. 1: Wang-Nefske formulas for venting through an orifice are used. Blockage is not considered. EQ. 2: Wang-Nefske formulas for venting through an orifice are used. Blockage of venting area due to contact is considered. EQ. 3: Leakage formulas of Graefe, Krummheuer, and Siejak [1990] are used. Blockage is not considered. EQ. 4: Leakage formulas of Graefe, Krummheuer, and Siejak [1990] are used. Blockage of venting area due to contact is considered. EQ. 5: Leakage formulas based on flow through a porous media are used. Blockage is not considered. EQ. 6: Leakage formulas based on flow through a porous media are used. Blockage of venting area due to contact is considered.
TSRFAC	Tensile stress cutoff reduction factor LT.0: TSRFAC is the load curve ID of the curve defining TSRFAC versus time.
A1 A2 A3	Components of vector a for AOPT = 2.
V1 V2 V3	Components of vector v for AOPT = 3.
D1 D2 D3	Components of vector d for AOPT = 2.
BETA	Material angle in degrees for AOPT = 3, may be overridden on the element card, see *ELEMENT_SHELL_BETA.
LCA	Load curve ID for stress versus strain along the a-axis fiber; available for FORM=4 only. If zero, EA is used.
LCB	Load curve ID for stress versus strain along the b-axis fiber; available for FORM=4 only. If zero, EB is used.
LCAB	Load curve ID for stress versus strain in the ab-plane; available for FORM=4 only.. If zero, GAB is used.

<u>VARIABLE</u>	<u>DESCRIPTION</u>
LCUA	Unload/reload curve ID for stress versus strain along the a-axis fiber; available for FORM=4 only. If zero, LCA is used.
LCUB	Unload/reload curve ID for stress versus strain along the b-axis fiber; available for FORM=4 only. If zero, LCB is used.
LCUAB	Unload/reload curve ID for stress versus strain in the ab-plane; available for FORM=4 only.. If zero, LCAB is used.

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.
5. For FORM=0, 1, and 2, the a-axis and b-axis fiber directions are assumed to be orthogonal and are completely defined by the material axes option, AOPT=0, 2, or 3. For FORM=3 or 4, the fiber directions are not assumed orthogonal and must be specified using the ICOMP=1 option on *SECTION_SHELL. Offset angles should be input into the B1 and B2 fields used normally for integration points 1 and 2. The a-axis and b-axis directions will then be offset from the a-axis direction as determined by the material axis option, AOPT=0, 2, or 3.
6. For FORM=4, nonlinear true stress versus true strain load curves may be defined for a-axis, b-axis, and shear stresses for loading and also for unloading and reloading. All curves should start at the origin and be defined for positive strains only. The a-axis and b-axis stress follows the curves for tension only. For compression, stress is calculated from the constant values, EA or EB. Shear stress/strain behavior is assumed symmetric. If a load curve is omitted, the stress is calculated from the appropriate constant modulus, EA, EB, or GAB.
7. When both loading and unloading curves are defined, the initial yield strain is assumed to be equal to the strain at the first point in the load curve with stress greater than zero. When strain

exceeds the yield strain, the stress continues to follow the load curve and the yield strain is updated to the current strain. When unloading occurs, the unload/reload curve is shifted along the x-axis until it intersects the load curve at the current yield strain. If the curve shift is to the right, unloading and reloading will follow the shifted unload/reload curve. If the curve shift is zero or to the left, unloading and reloading will occur along the load curve.

- 8 The FVOPT flag allows an airbag fabric venting equation to be assigned to a material. The anticipated use for this option is to allow a vent to be defined using FVOPT=1 or 2 for one material and fabric leakage to be defined for using FVOPT=3, 4, 5, or 6 for other materials. In order to use FVOPT, a venting option must first be defined for the airbag using the OPT parameter on *AIRBAG_WANG_NEFSKE or *AIRBAG_HYBRID. If OPT=0, then FVOPT is ignored. If OPT is defined and FVOPT is omitted, then FVOPT is set equal to OPT.
- 9 The TSRFAC factor is used to assure that airbags that have a reference geometry will open to the correct geometry. Airbags that use a reference geometry might have an initial geometry that results in initial tensile strains. To prevent such strains from prematurely opening an airbag, these tensile strains are eliminated by default. A side effect of this behavior is that airbags that use a reference geometry and that are initially stretched will never achieve the correct shape. The TSRFAC factor is used to restore the tensile strains over time such that the correct geometry is achieved. It is recommended that a load curve be used to define TSRFAC as function of time. Initially the load curve ordinate value should be 1.0 which will allow the bag to remain unstressed. At a time when the bag is partially open, the value of TSRFAC can ramp down to 0.99 or 0.999 which will cause the initially stretched elements to shrink. Permissible values for TSRFAC is 0.9 to 1.0.

*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			

<u>VARIABLE</u>	<u>DESCRIPTION</u>
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	ITER
Type	I	F	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				A1	A2	A3		
Type				F	F	F		

Card 5

Variable	V1	V2	V3	D1	D2	D3	BETA	
Type	F	F	F	F	F	F	F	

VARIABLE**DESCRIPTION**

MID	Material identification. A unique number has to be chosen.
RO	Mass density.
E	Young's modulus, E
PR	Poisson's ratio, ν
HR	Hardening rule: EQ.1.0: linear (default), EQ.2.0: exponential. EQ.3.0: load curve
P1	Material parameter: HR.EQ.1.0: Tangent modulus, HR.EQ.2.0: k, strength coefficient for exponential hardening
P2	Material parameter: HR.EQ.1.0: Yield stress HR.EQ.2.0: n, exponent
ITER	Iteration flag for speed: ITER.EQ.0.0: fully iterative ITER.EQ.1.0: fixed at three iterations Generally, ITER=0 is recommended. However, ITER=1 is somewhat faster and may give acceptable results in most problems.
M	m, exponent in Barlat's yield surface
R00	R ₀₀ , Lankford parameter determined from experiments
R45	R ₄₅ , Lankford parameter determined from experiments
R90	R ₉₀ , Lankford parameter determined from experiments
LCID	load curve ID for the load curve hardening rule
E0	ϵ_0 for determining initial yield stress for exponential hardening. (Default=0.0)

<u>VARIABLE</u>	<u>DESCRIPTION</u>
SPI	<p><i>spi</i>, if ϵ_0 is zero above. (Default=0.0)0</p> <p>EQ.0.0: $\epsilon_0 = (E / k) * [1 / (n - 1)]$</p> <p>LE..02: $\epsilon_0 = spi$</p> <p>GT..02: $\epsilon_0 = (spi / k) * [1 / n]$</p>
AOPT	<p>Material axes option (see MAT_OPTION TROPIC_ELASTIC for a more complete description):</p> <p>EQ. 0.0: locally orthotropic with material axes determined by element nodes 1, 2, and 4, as with *DEFINE_COORDINATE_NODES.</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: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle, BETA, from a line in the plane of the element defined by the cross product of the vector v with the element normal.</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.
BETA	Material angle in degrees for AOPT = 3, may be overridden on the element card, see *ELEMENT_SHELL_BETA.

Remarks:

The anisotropic yield criterion Φ 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 σ_Y is the yield stress and $K_{i=1,2}$ are given by:

$$K_1 = \frac{\sigma_x + h\sigma_y}{2}$$

$$K_2 = \sqrt{\left(\frac{\sigma_x - h\sigma_y}{2}\right)^2 + p^2\tau_{xy}^2}$$

The anisotropic material constants a, c, h, and p are obtained through R_{00} , R_{45} , and R_{90} :

$$a = 2 - 2\sqrt{\frac{R_{00}}{1 + R_{00}} \frac{R_{90}}{1 + R_{90}}} \qquad c = 2 - a$$

$$h = \sqrt{\frac{R_{00}}{1 + R_{00}} \frac{1 + R_{90}}{R_{90}}}$$

The anisotropy parameter p is calculated implicitly. According to Barlat and Lian the R value, width to thickness strain ratio, for any angle ϕ can be calculated from:

$$R_\phi = \frac{2m\sigma_Y^m}{\left(\frac{\partial\Phi}{\partial\sigma_x} + \frac{\partial\Phi}{\partial\sigma_y}\right)\sigma_\phi} - 1$$

where σ_ϕ is the uniaxial tension in the ϕ direction. This expression can be used to iteratively calculate the value of p . Let $\phi=45$ 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 ε_{yp} is the elastic strain to yield and $\bar{\varepsilon}^p$ is the effective plastic strain (logarithmic). If SIGY is set to zero, the strain to yield 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_037 *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

<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	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 σ_{y1} , σ_{y2} , and σ_{y3} , are the tensile yield stresses and σ_{y12} , σ_{y23} , and σ_{y31} are the shear yield stresses. The constants F, G, H, L, M, and N are related to the yield stress by

$$2L = \frac{1}{\sigma_{23}^2}$$

$$2M = \frac{1}{\sigma_{y31}^2}$$

$$2N = \frac{1}{\sigma_{y12}^2}$$

$$2F = \frac{1}{\sigma_{y2}^2} + \frac{1}{\sigma_{y3}^2} - \frac{1}{\sigma_{y1}^2}$$

$$2G = \frac{1}{\sigma_{y3}^2} + \frac{1}{\sigma_{y1}^2} - \frac{1}{\sigma_{y2}^2}$$

$$2H = \frac{1}{\sigma_{y1}^2} + \frac{1}{\sigma_{y2}^2} - \frac{1}{\sigma_{y3}^2}$$

The isotropic case of von Mises plasticity can be recovered by setting $F = G = H = \frac{1}{2\sigma_y^2}$

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

$$W = \frac{G}{2} \left(\frac{\text{II}}{\text{III}} + 2\sqrt{\text{III}} - 5 \right)$$

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

$$S^{ij} = G \left[\left(I\delta_{ij} - C_{ij} \right) \frac{1}{\text{III}} + \left(\sqrt{\text{III}} - \frac{\text{II}}{\text{III}} \right) C_{ij}^{-1} \right]$$

where C_{ij} is the right Cauchy-Green strain tensor. This stress measure is transformed to the Cauchy stress, σ_{ij} , according to the relationship

$$\sigma^{ij} = \text{III}^{-1/2} F_{ik} F_{jl} S_{lk}$$

where F_{ij} is the deformation gradient tensor.

***MAT_FLD_TRANSVERSELY_ANISOTROPIC**

This is Material Type 39. This model is for simulating sheet forming processes with anisotropic material. Only transverse anisotropy can be considered. Optionally, an arbitrary dependency of stress and effective plastic strain can be defined via a load curve. A Forming Limit Diagram (FLD) can be defined using a curve and is used to compute the maximum strain ratio which can be plotted in LS-POST. 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							

<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	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 Forming Limit Diagram. Minor strains in percent are defined as abscissa values and Major strains in percent are defined as ordinate values. The forming limit diagram is shown in Figure 20.15. 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 $\epsilon_{minor_{workpiece}}$

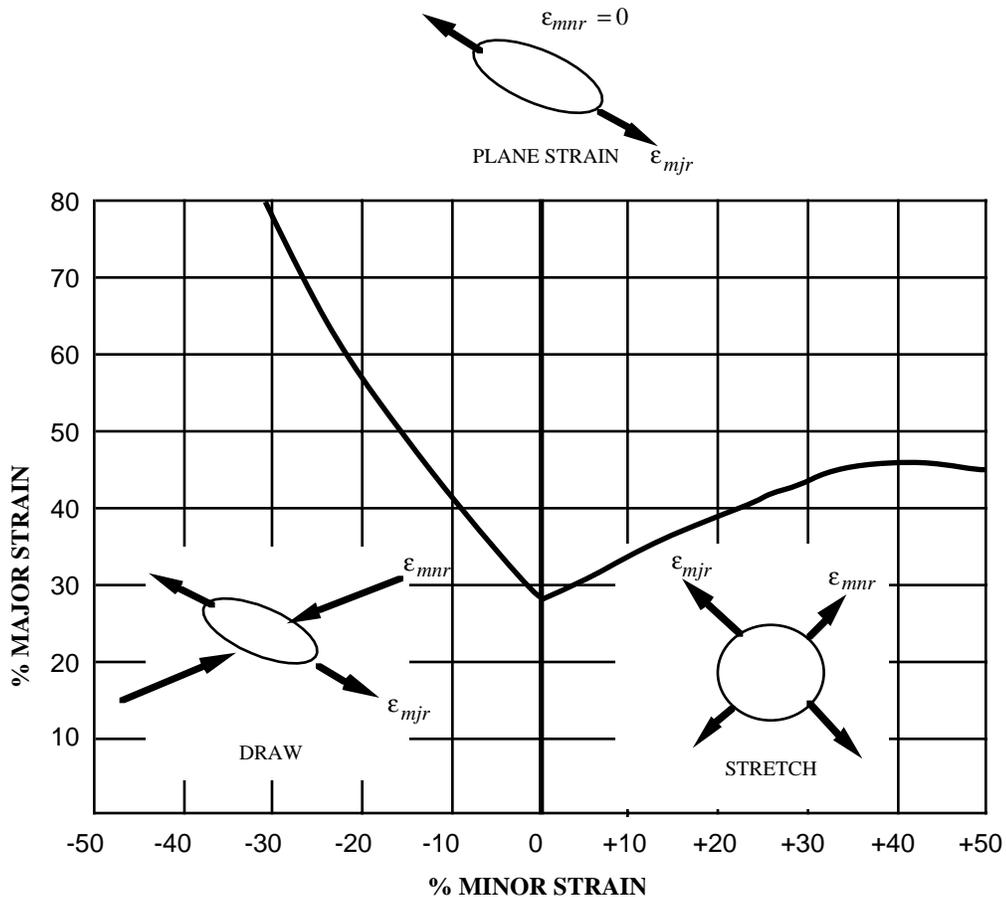


Figure 20.15. Forming Limit Diagram.

***MAT_NONLINEAR_ORTHOTROPIC**

This is Material Type 40. This model allows the definition of an orthotropic nonlinear elastic material based on a finite strain formulation with the initial geometry as the reference. Failure is optional with two failure criteria available. Optionally, stiffness proportional damping can be defined. In the stress initialization phase, temperatures can be varied to impose the initial stresses. This model is only available for shell and solid elements. We do not recommend using this model at this time since it can be unstable especially if the stress-strain curves increase in stiffness with increasing strain.

Card 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							

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				A1	A2	A3		
Type				F	F	F		

Card 5 1 2 3 4 5 6 7 8

Variable	V1	V2	V3	D1	D2	D3	BETA	
Type	F	F	F	F	F	F	F	

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 ν_{ba} , Poisson's ratio ba.
- PRCA ν_{ca} , Poisson's ratio ca.
- PRCB ν_{cb} , Poisson's ratio cb.
- GAB G_{ab} , shear modulus ab.

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 λ_a is the stretch ratio along the a axis.
LCIDB	Optional load curve ID defining the nominal stress versus strain along b-axis. Strain is defined as $\lambda_b - 1$ where λ_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 (see MAT_OPTION TROPIC_ELASTIC for more complete description): EQ. 0.0: locally orthotropic with material axes determined by element nodes 1, 2, and 4, as with *DEFINE_COORDINATE_NODES . EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR . EQ. 3.0: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle, BETA, from a line in the plane of the element defined by the cross product of the vector v with the element normal.
A1,A2,A3	$a_1 a_2 a_3$, define components of vector a for AOPT = 2.
D1,D2,D3	$d_1 d_2 d_3$, define components of vector d for AOPT = 2.
V1,V2,V3	$v_1 v_2 v_3$, define components of vector v for AOPT = 3
BETA	Material angle in degrees for AOPT = 3, may be overridden on the element card, see *ELEMENT_SHELL_BETA .

The following input is optional and applies to SOLID ELEMENTS only.

<u>VARIABLE</u>	<u>DESCRIPTION</u>
LCIDC	Load curve ID defining the nominal stress versus strain along c-axis. Strain is defined as $\lambda_c - 1$ where λ_c is the stretch ratio along the c axis.
LCIDAB	Load curve ID defining the nominal ab shear stress versus ab-strain in the ab-plane. Strain is defined as the $\sin(\gamma_{ab})$ where γ_{ab} is the shear angle.
LCIDBC	Load curve ID defining the nominal ab shear stress versus ab-strain in the bc-plane. Strain is defined as the $\sin(\gamma_{bc})$ where γ_{bc} is the shear angle.
LCIDCA	Load curve ID defining the nominal ab shear stress versus ab-strain in the ca-plane. Strain is defined as the $\sin(\gamma_{ca})$ where γ_{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. Isotropic 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	ITHERM	IHYPER	IEOS			
Type	I	I	I	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	BETA	
Type	F	F	F	F	F	F	F	

Define LMC material parameters using 8 parameters per card.

Card 1 2 3 4 5 6 7 8

Variable	P1	P2	P3	P4	P5	P6	P7	P8
Type	F	F	F	F	F	F	F	F

<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. (LMC ≤ 40 if IORTHO=1)
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 shell elements due to a material failure criterion.
ITHERM	Temperature flag (on=1). Compute element temperature.
AOPT	Material axes option (see MAT_OPTION TROPIC_ELASTIC for a more complete description): EQ. 0.0: locally orthotropic with material axes determined by element nodes 1, 2, and 4, as with *DEFINE_COORDINATE_NODES. EQ. 1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center; this is the a-direction. This option is for solid elements only. EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR. EQ. 3.0: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle, BETA, from a line in the plane of the element defined by the cross product of the vector v with the element normal.

<u>VARIABLE</u>	<u>DESCRIPTION</u>
	EQ. 4.0: locally orthotropic in cylindrical coordinate system with the material axes determined by a vector \mathbf{v} , and an originating point, P, which define the centerline axis. This option is for solid elements only.
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.
BETA	Material angle in degrees for AOPT = 3, may be overridden on the element card, see *ELEMENT_SHELL_BETA.
P1	First material parameter.
P2	Second material parameter.
P3	Third material parameter.
P4	Fourth material parameter.
.	.
.	.
.	.
PLMC	LMCth material parameter.

*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

VARIABLE	DESCRIPTION
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

VARIABLE	DESCRIPTION
A1	α_1 , initial value of internal state variable 1
A2	α_2 , initial value of internal state variable 2
A3	α_4 , initial value of internal state variable 3
A4	α_5 , initial value of internal state variable 4
A5	α_6 , initial value of internal state variable 5
A6	κ , 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
v		—	—
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 σ 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, κ is the scalar hardening variable, and ξ' is the difference between the deviatoric Cauchy stress σ' and the tensor variable α' ,

$$\xi' = \sigma' - \alpha'$$

and $f(T)$, $Y(T)$, $V(T)$ are scalar functions whose specific dependence upon the temperature is given below. Assuming isotropic thermal expansion and introducing the expansion coefficient \dot{A} , the thermal stretching can be written,

$$D^{th} = \dot{A}T1.$$

The evolution of the internal variables α and κ 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 ρ is the density of the material and C_v the specific heat.

In terms of the input parameters the functions defined above become:

$V(T) = C1 \exp(-C2/T)$	$h(T) = C9 \exp(C10/T)$
$Y(T) = C3 \exp(C4/T)$	$r_s(T) = C11 \exp(-C12/T)$
$f(T) = C5 \exp(-C6/T)$	$RD(T) = C13 \exp(-C14/T)$
$rd(T) = C7 \exp(-C8/T)$	$H(T) = C15 \exp(C16/T)$
	$RS(T) = C17 \exp(-C18/T)$

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 (°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

<u>VARIABLE</u>	<u>DESCRIPTION</u>
C13	1/psi
C14	°R
C15	psi
C16	°R
C17	1/psi-s
C18	°R
A1	α_1 , initial value of internal state variable 1
A2	α_2 , initial value of internal state variable 2
A3	α_3 , initial value of internal state variable 3
A4	α_4 , initial value of internal state variable 4
A5	α_5 , initial value of internal state variable 5
A6	α_6 , initial value of internal state variable 6
N	Exponent in damage evolution
D0	Initial damage (porosity)
FS	Failure strain for erosion.

Remarks:

The evolution of the damage parameter, ϕ , is defined by [Bammann, et al. 1990]

$$\dot{\phi} = \beta \left[\frac{1}{(1 - \phi)^N} - (1 - \phi) \right]^{|D^p|}$$

in which

$$\beta = \sinh \left[\frac{2(2N - 1)p}{(2N - 1)\bar{\sigma}} \right]$$

where p is the pressure and $\bar{\sigma}$ is the effective stress.

*MAT_CLOSED_CELL_FOAM

This is Material Type 53. This allows the modeling of low density, closed cell polyurethane foam. It is for simulating impact limiters in automotive applications. The effect of the confined air pressure is included with the air being treated as an ideal gas. The general behavior is isotropic with uncoupled components of the stress tensor.

Card 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	LCID						
Type	F	I						

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_0
PHI	Ratio of foam to polymer density, ϕ
GAMA0	Initial volumetric strain, γ_0 . The default is zero.
LCID	Optional load curve defining the von Mises yield stress versus $-\gamma$. If the load curve ID is given, the yield stress is taken from the curve and the constants a, b, and c are not needed. The load curve is defined in the positive quadrant, i.e., positive values of γ are defined as negative values on the abscissa.

Remarks:

A rigid, low density, closed cell, polyurethane foam model developed at Sandia Laboratories [Neilsen et al. 1987] has been recently implemented for modeling impact limiters in automotive applications. A number of such foams were tested at Sandia and reasonable fits to the experimental data were obtained.

In some respects this model is similar to the crushable honeycomb model type 26 in that the components of the stress tensor are uncoupled until full volumetric compaction is achieved. However, unlike the honeycomb model this material possesses no directionality but includes the effects of confined air pressure in its overall response characteristics..

$$\sigma_{ij} = \sigma_{ij}^{sk} - \delta_{ij} \sigma^{air}$$

where σ_{ij}^{sk} is the skeletal stress and σ^{air} is the air pressure computed from the equation:

$$\sigma^{air} = -\frac{p_0 \gamma}{1 + \gamma - \phi}$$

where p_0 is the initial foam pressure, usually taken as the atmospheric pressure, and γ defines the volumetric strain

$$\gamma = V - 1 + \gamma_0$$

where V is the relative volume, defined as the ratio of the current volume to the initial volume, and γ_0 is the initial volumetric strain, which is typically zero. The yield condition is applied to the principal skeletal stresses, which are updated independently of the air pressure. We first obtain the skeletal stresses:

$$\sigma_{ij}^{sk} = \sigma_{ij} + \sigma_{ij} \sigma^{air}$$

and compute the trial stress, σ^{skt}

$$\sigma_{ij}^{skt} = \sigma_{ij}^{sk} + E \dot{\epsilon}_{ij} \Delta t$$

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, σ_i^{skt} , exceeds the yield stress, σ_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 γ is the volumetric strain as defined above. After scaling the principal stresses they are transformed back into the global system and the final stress state is computed

$$\sigma_{ij} = \sigma_{ij}^{sk} - \delta_{ij} \sigma^{air}$$

*MAT_ENHANCED_COMPOSITE_DAMAGE

These are Material Types 54-55 which are enhanced versions of the composite model material type 22. Arbitrary 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. By using the user defined integration rule, see *INTEGRATION_SHELL, the constitutive constants can vary through the shell thickness. For all shells, except the DKT formulation, laminated shell theory can be activated to properly model the transverse shear deformation. Lamination theory is applied to correct for the assumption of a uniform constant shear strain through the thickness of the shell. For sandwich shells where the outer layers are much stiffer than the inner layers, the response will tend to be too stiff unless lamination theory is used. To turn on lamination theory see *CONTROL_SHELL.

Card 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				A1	A2	A3	MANGLE	
Type				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	

<u>VARIABLE</u>	<u>DESCRIPTION</u>
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	ν_{ba} , Poisson's ratio ba
(PRCA)	ν_{ca} , Poisson's ratio ca (not used)
(PRCB)	ν_{cb} , Poisson's ratio cb (not used)
GAB	G_{ab} , shear modulus ab
GBC	G_{bc} , shear modulus bc

<u>VARIABLE</u>	<u>DESCRIPTION</u>
GCA	G_{ca} , shear modulus ca
(KF)	Bulk modulus of failed material (not used)
AOPT	Material axes option (see MAT_OPTION TROPIC_ELASTIC for a more complete description): EQ. 0.0: locally orthotropic with material axes determined by element nodes 1, 2, and 4, as with *DEFINE_COORDINATE_NODES. EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR. EQ. 3.0: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle (MANGLE) from a line in the plane of the element defined by the cross product of the vector v with the element normal.
A1 A2 A3	Define components of vector a for AOPT = 2.
V1 V2 V3	Define components of vector v for AOPT = 3.
D1 D2 D3	Define components of vector d for AOPT = 2.
MANGLE	Material angle in degrees for AOPT = 3, may be overridden on the element card, see *ELEMENT_SHELL_BETA or *ELEMENT_SOLID_ORTHO.
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: ≤ 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.

VARIABLE	DESCRIPTION
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 (mat_54) 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 (mat_55) 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. <i>ef(i)</i>	<i>tensile fiber mode</i>	<i>1 - elastic</i> <i>0 - failed</i>	81
2. <i>ec(i)</i>	<i>compressive fiber mode</i>		82
3. <i>em(i)</i>	<i>tensile matrix mode</i>		83
4. <i>ed(i)</i>	<i>compressive matrix mode</i>		84
5. <i>efail</i>	<i>max[ef(ip)]</i>		85
6. <i>dam</i>	<i>damage parameter</i>	<i>-1 - element intact</i> <i>10⁻⁸ - element in crashfront</i> <i>+1 - element failed</i>	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
$\frac{1}{nip} \sum_{i=1}^{nip} em(i)$	3

***MAT_LOW_DENSITY_FOAM**

This is Material Type 57 for modeling highly compressible low density foams. Its main applications are for seat cushions and padding on the Side Impact Dummies (SID). Optionally, a tension cut-off failure can be defined. Also, see the notes below.

Card 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.		0.05
Remarks	---	---	---	---	---	3	1	---

Card 2

Variable	SHAPE	FAIL	BVFLAG	ED	BETA1	KCON	REF	
Type	F	F	F	F	F	F	F	
Default	1.0	0.0	0.0	0.0	0.0	0.0	0.0	
Remarks	3	---	2	5	5	6		

VARIABLE

DESCRIPTION

- MID Material identification. A unique number 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

<u>VARIABLE</u>	<u>DESCRIPTION</u>
HU	Hysteretic unloading factor between 0 and 1 (default=1, i.e., no energy dissipation), see also Figure 20.16.
BETA	β , decay constant to model creep in unloading
DAMP	Viscous coefficient (.05< recommended value <.50) to model damping effects. LT.0.0: DAMP is the load curve ID, which defines the damping constant as a function of the maximum strain in compression defined as: $\epsilon_{\max} = \max(1 - \lambda_1, 1 - \lambda_2, 1 - \lambda_3).$ In tension, the damping constant is set to the value corresponding to the strain at 0. The abscissa should be defined from 0 to 1.
SHAPE	Shape factor for unloading. Active for nonzero values of the hysteretic unloading factor. Values less than one reduces the energy dissipation and greater than one increases dissipation, see also Figure 20.16.
FAIL	Failure option after cutoff stress is reached: EQ.0.0: tensile stress remains at cut-off value, EQ.1.0: tensile stress is reset to zero.
BVFLAG	Bulk viscosity activation flag, see remark below: EQ.0.0: no bulk viscosity (recommended), EQ.1.0: bulk viscosity active.
ED	Optional Young's relaxation modulus, E_d , for rate effects. See comments below.
BETA1	Optional decay constant, β_1 .
KCON	Stiffness coefficient for contact interface stiffness. If undefined the maximum slope in stress vs. strain curve is used. When the maximum slope is taken for the contact, the time step size for this material is reduced for stability. In some cases Δt may be significantly smaller, and defining a reasonable stiffness is recommended.
REF	Use reference geometry to initialize the stress tensor. The reference 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 compressive behavior is illustrated in Figure 20.16 where hysteresis on unloading is shown. This behavior under uniaxial loading is assumed not to significantly couple in the transverse directions. In tension the material behaves in a linear fashion until tearing occurs. Although our implementation may be somewhat unusual, it was motivated by Storakers [1986].

The model uses tabulated input data for the loading curve where the nominal stresses are defined as a function of the elongations, ϵ_i , which are defined in terms of the principal stretches, λ_i , as:

$$\epsilon_i = \lambda_i - 1$$

The stretch ratios are found by solving for the eigenvalues of the left stretch tensor, V_{ij} , which is obtained via a polar decomposition of the deformation gradient matrix, F_{ij} . Recall that,

$$F_{ij} = R_{ik}U_{kj} = V_{ik}R_{kj}$$

The update of V_{ij} follows the numerically stable approach of [Taylor and Flanagan 1989]. After solving for the principal stretches, we compute the elongations and, if the elongations are compressive, the corresponding values of the nominal stresses, τ_i , are interpolated. If the elongations are tensile, the nominal stresses are given by

$$\tau_i = E\epsilon_i$$

and the Cauchy stresses in the principal system become

$$\sigma_i = \frac{\tau_i}{\lambda_j \lambda_k}$$

The stresses can now be transformed back into the global system for the nodal force calculations.

Additional Remarks:

1. When hysteretic unloading is used the reloading will follow the unloading curve if the decay constant, β , is set to zero. If β is nonzero the decay to the original loading curve is governed by the expression:

$$1.-e^{-\beta t}$$

2. The bulk viscosity, which generates a rate dependent pressure, may cause an unexpected volumetric response and, consequently, it is optional with this model.
3. The hysteretic unloading factor results in the unloading curve to lie beneath the loading curve as shown in Figure 20.16. This unloading provide energy dissipation which is reasonable in certain kinds of foam.
4. Note that since this material has no effective plastic strain, the internal energy per initial volume is written into the output databases.
5. Rate effects are accounted for through linear viscoelasticity by a convolution integral of the form

$$\sigma_{ij}^r = \int_0^t g_{ijkl}(t - \tau) \frac{\partial \epsilon_{kl}}{\partial \tau} d\tau$$

where $g_{ijkl}(t - \tau)$ is the relaxation function. The stress tensor, σ_{ij}^r , augments the stresses determined from the foam, σ_{ij}^f ; consequently, the final stress, σ_{ij} , is taken as the summation of the two contributions:

$$\sigma_{ij} = \sigma_{ij}^f + \sigma_{ij}^r.$$

Since we wish to include only simple rate effects, the relaxation function is represented by one term from the Prony series:

$$g(t) = \alpha_0 + \sum_{m=1}^N \alpha_m e^{-\beta_m t}$$

given by,

$$g(t) = E_d e^{-\beta_1 t}$$

This model is effectively a Maxwell fluid which consists of a damper and spring in series. We characterize this in the input by a Young's modulus, E_d , and decay constant, β_1 . The formulation is performed in the local system of principal stretches where only the principal values of stress are computed and triaxial coupling is avoided. Consequently, the one-dimensional nature of this foam material is unaffected by this addition of rate effects. The addition of rate effects necessitates twelve additional history variables per integration point. The cost and memory overhead of this model comes primarily from the need to "remember" the local system of principal stretches.

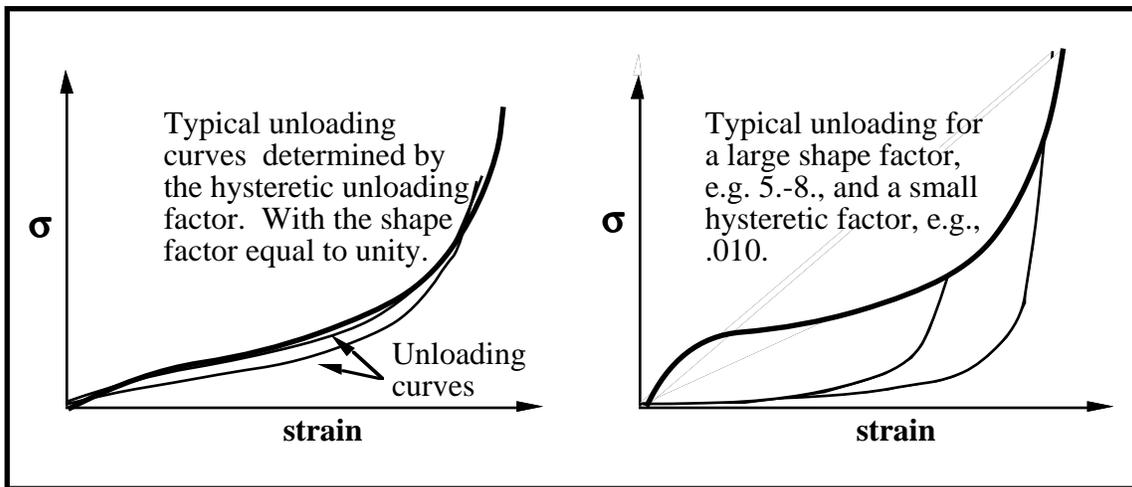


Figure 20.16. Behavior of the low density urethane foam model.

6. The time step size is based on the current density and the maximum of the instantaneous loading slope, E, and ECON. If ECON is undefined the maximum slope in the loading curve is used instead.

*MAT_LAMINATED_COMPOSITE_FABRIC

This is Material Type 58. Depending on the type of failure surface, this model may be used to model composite materials with unidirectional layers, complete laminates, and woven fabrics. This model is implemented for shell 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	SLIMT1	SLIMC1	SLIMT2	SLIMC2	SLIMS
Type	F	F	F	F	F	F	F	F

Card 3

Variable	AOPT	TSIZE	ERODS	SOFT	FS			
Type	F	F	F	F	F			

Card 4

Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card 5

Variable	V1	V2	V3	D1	D2	D3	BETA	
Type	F	F	F	F	F	F	F	

Card 6

Variable	E11C	E11T	E22C	E22T	GMS			
Type	F	F	F	F	F			

Card 7

Variable	XC	XT	YC	YT	SC			
Type	F	F	F	F	F			

<u>VARIABLE</u>	<u>DESCRIPTION</u>
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	ν_{ba} , Poisson's ratio ba
TAU1	τ_1 , stress limit of the first slightly nonlinear part of the shear stress versus shear strain curve. The values τ_1 and γ_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	γ_1 , strain limit of the first slightly nonlinear part of the shear stress versus shear strain curve.

<u>VARIABLE</u>	<u>DESCRIPTION</u>
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 determine the minimum stress limit after stress maximum (shear).
AOPT	Material axes option (see MAT_OPTION TROPIC_ELASTIC for a more complete description): EQ. 0.0: locally orthotropic with material axes determined by element nodes 1, 2, and 4, as with *DEFINE_COORDINATE_NODES. EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR. EQ. 3.0: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle (BETA) from a line in the plane of the element defined by the cross product of the vector \mathbf{v} with the element normal.
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.

VARIABLE	DESCRIPTION
	EQ.-1.:faceted failure surface. When the strength values are reached then damage evolves in tension and compression for both the fiber and transverse direction. Shear behavior is also considered. This option can be used with complete laminates and fabrics.
XP YP ZP	Define coordinates of point p for AOPT = 1.
A1 A2 A3	Define components of vector a for AOPT = 2.
V1 V2 V3	Define components of vector v for AOPT = 3.
D1 D2 D3	Define components of vector d for AOPT = 2.
BETA	Material angle in degrees for AOPT = 3, may be overridden on the element card, see *ELEMENT_SHELL_BETA.
E11C	Strain at longitudinal compressive strength, a-axis.
E11T	Strain at longitudinal tensile strength, a-axis.
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 20.17 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.

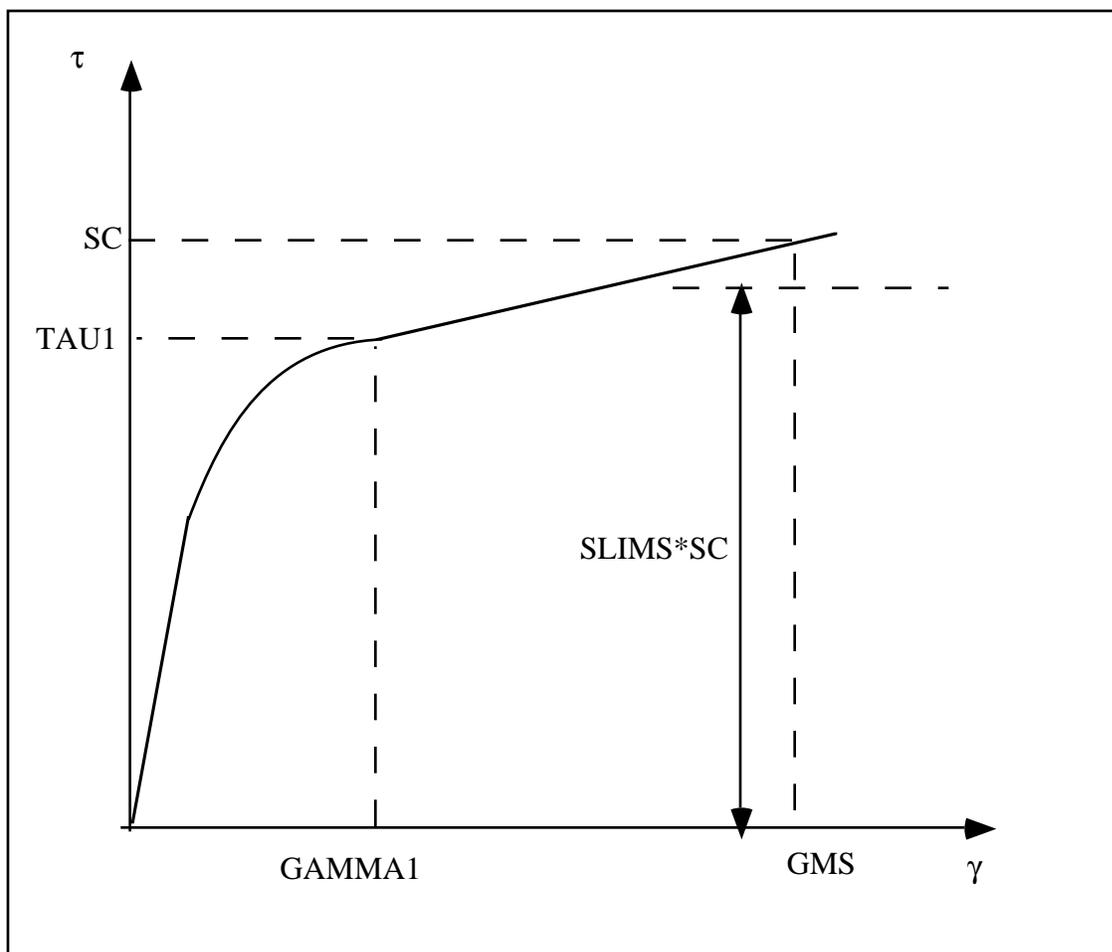


Figure 20.17. Stress-strain diagram for shear.

*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	BETA	
Type	F	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					

VARIABLE	DESCRIPTION
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	ν_{ba} Poisson's ratio ba
PRCA	ν_{ca} Poisson's ratio ca
PRCB	ν_{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 (see MAT_OPTION TROPIC_ELASTIC for a more complete description): EQ. 0.0: locally orthotropic with material axes determined by element nodes 1, 2, and 4, as with *DEFINE_COORDINATE_NODES. EQ. 1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center; this is the a-direction. This option is for solid elements only. EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR. EQ. 3.0: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle, BETA, from a line in the plane of the element defined by the cross product of the vector \mathbf{v} with the element normal. EQ. 4.0: locally orthotropic in cylindrical coordinate system with the material axes determined by a vector \mathbf{v} , and an originating point, P, which define the centerline axis. This option is for solid elements only.
XP YP ZP	Define coordinates of point p for AOPT = 1 and 4.
A1 A2 A3	Define components of vector a for AOPT = 2.
V1 V2 V3	Define components of vector v for AOPT = 3 and 4.
D1 D2 D3	Define components of vector d for AOPT = 2:
BETA	Material angle in degrees for AOPT = 3, may be overridden on the element card, see *ELEMENT_SHELL_BETA or *ELEMENT_SOLID_ORTHO.

VARIABLE	DESCRIPTION
MAFLAG	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.
BETA	Material angle in degrees for AOPT = 3, may be overridden on the element card, see *ELEMENT_SHELL_BETA.
TSIZE	Time step for automatic element deletion
ALP	Nonlinear shear stress parameter
SOFT	Softening reduction factor for strength in crush
FBRT	Softening of fiber tensile strength
SR	s_r , reduction factor(default=0.447)
SF	s_f , softening factor(default=0.0)
XC	Longitudinal compressive strength, a-axis
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

VARIABLE	DESCRIPTION
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 temperature, see type (iii). (Optional).
T1, T2,...TN	Temperatures, define up to 8 values
PR1, PR2,...PRN	Poisson's ratios for the temperatures T_i
V1, V2,...VN	Corresponding viscosity coefficients (define only one if not varying with temperature).
E1, E2,...EN	Corresponding Young's moduli coefficients (define only one if not varying with temperature).
ALPHA....	Corresponding thermal expansion coefficients

Remarks:

Volumetric behavior is treated as linear elastic. The deviatoric strain rate is considered to be the sum of elastic and viscous strain rates:

$$\dot{\tilde{\epsilon}}'_{total} = \dot{\tilde{\epsilon}}'_{elastic} + \dot{\tilde{\epsilon}}'_{viscous} = \frac{\dot{\tilde{\sigma}}'}{2G} + \frac{\dot{\tilde{\sigma}}'}{2\nu}$$

where G is the elastic shear modulus, ν is the viscosity coefficient, and \sim indicates a tensor. The stress increment over one timestep dt is

$$d\tilde{\sigma}' = 2G \dot{\tilde{\epsilon}}'_{total} dt - \frac{G}{\nu} dt \tilde{\sigma}'$$

The stress before the update is used for $\tilde{\sigma}'$. For shell elements the through-thickness strain rate is calculated as follows.

$$d\sigma_{33} = 0 = K \left(\dot{\epsilon}_{11} + \dot{\epsilon}_{22} + \dot{\epsilon}_{33} \right) dt + 2G \dot{\epsilon}'_{33} dt - \frac{G}{\nu} dt \sigma'_{33}$$

where the subscript $ij = 33$ denotes the through-thickness direction and K is the elastic bulk modulus. This leads to:

$$\dot{\epsilon}_{33} = -a \left(\dot{\epsilon}_{11} + \dot{\epsilon}_{22} \right) + bp$$

$$a = \frac{\left(K - \frac{2}{3}G \right)}{\left(K + \frac{4}{3}G \right)}$$

$$b = \frac{Gdt}{\nu \left(K + \frac{4}{3}G \right)}$$

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. This material is a classical Kelvin-Maxwell model for modelling viscoelastic bodies, e.g., foams. This model is valid for solid elements only. 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

<u>VARIABLE</u>	<u>DESCRIPTION</u>
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_∞
DC	Maxwell decay constant, β [FO=0.0] or Kelvin relaxation constant, τ [FO=1.0]
FO	Formulation option: EQ.0.0: Maxwell, EQ.1.0: Kelvin.
SO	Strain (logarithmic) output option to 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:

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

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

<u>VARIABLE</u>	<u>DESCRIPTION</u>
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, ν

Remarks:

The model consists of a nonlinear elastic stiffness in parallel with a viscous damper. The elastic stiffness is intended to limit total crush while the viscosity absorbs energy. The stiffness E_2 exists to prevent timestep problems. It is used for time step calculations as long as E_1' 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:

$$E_1' = E_1(V^{-n_1})$$

$$V_2' = V_2(abs(1 - V))^{n_2}$$

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 elastic-perfectly-plastic at the tension cut-off value. A modified version of this model, *MAT_MODIFIED_CRUSHABLE_FOAM includes strain rate effects.

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	
Default	none	none	none	none	none	0.0	0.10	

<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
LCID	Load curve ID defining yield stress versus volumetric strain, γ , see Figure 20.18.
TSC	Tensile stress cutoff. A nonzero, positive value is strongly recommended for realistic behavior.
DAMP	Rate sensitivity via damping coefficient (.05<recommended value<.50).

Remarks:

The volumetric strain is defined in terms of the relative volume, V, as:

$$\gamma = 1.-V$$

The relative volume is defined as the ratio of the current to the initial volume. In place of the effective plastic strain in the D3PLOT database, the integrated volumetric strain is output.

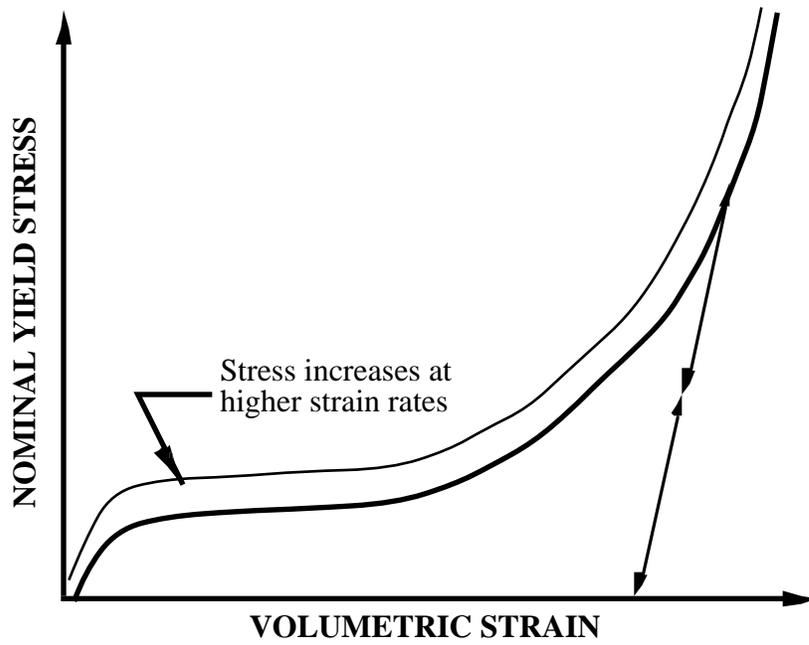


Figure 20.18. Behavior of strain rate sensitive crushable foam. Unloading is elastic to the tension cutoff. Subsequent reloading follows the unloading curve.

***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 σ is the yield stress, ε is the effective plastic strain, $\dot{\varepsilon}$ is the normalized effective plastic strain rate, and the constants k , m , and n can be expressed as functions of effective plastic strain or can be constant with respect to the plastic strain. The case of no strain hardening can be obtained by setting the exponent of the plastic strain equal to a very small positive value, i.e. 0.0001.

This model can be combined with the superplastic forming input to control the magnitude of the pressure in the pressure boundary conditions in order to limit the effective plastic strain rate so that it does not exceed a maximum value at any integration point within the model.

A fully viscoplastic formulation is optional. An additional cost is incurred but the improvement 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	

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MID	Material identification. A unique number has to be chosen.
RO	Mass density
G	Shear modulus
E0	ϵ_0 , factor to normalize strain rate
N	n, exponent for bcc metal
TROOM	T_r , room temperature

<u>VARIABLE</u>	<u>DESCRIPTION</u>
PC	pc , 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 ₁ , coefficients for flow stress, see notes below.
C2	C ₂ , coefficients for flow stress, see notes below.
C3	C ₃ , coefficients for flow stress, see notes below.
C4	C ₄ , coefficients for flow stress, see notes below.
C5	C ₅ , coefficients for flow stress, see notes below.
C6	C ₆ , coefficients for flow stress, see notes below.
EFAIL	Failure strain for erosion
VP	Formulation for rate effects: EQ.0.0: Scale yield stress (default) EQ.1.0: Viscoplastic formulation
B1	B ₁ , coefficients for polynomial to represent temperature dependency of flow stress yield.
B2	B ₂
B3	B ₃
G1	G ₁ , coefficients for defining heat capacity and temperature dependency of heat capacity.
G2	G ₂
G3	G ₃
G4	G ₄
BULK	Bulk modulus defined for shell elements only. Do not input for solid elements.

Remarks:

The Armstrong-Zerilli Material Model expresses the flow stress as follows.

For fcc metals ($n=0$),

$$\sigma = C_1 + \left\{ C_2 (\epsilon^p)^{1/2} \left[e^{(-C_3 + C_4 \ln(\dot{\epsilon}^*))T} \right] + C_5 \right\} \left(\frac{\mu(T)}{\mu(293)} \right)$$

ϵ^p = effective plastic strain

$\dot{\epsilon}^* = \frac{\dot{\epsilon}}{\epsilon_0}$ effective plastic strain rate where $\epsilon_0 = 1, 1e-3, 1e-6$ for time units of seconds, milliseconds, and microseconds, respectively.

For bcc metals ($n > 0$),

$$\sigma = C_1 + C_2 e^{(-C_3 + C_4 \ln(\dot{\epsilon}^*))T} + \left[C_5 (\epsilon^p)^n + C_6 \right] \left(\frac{\mu(T)}{\mu(293)} \right)$$

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. For finite length discrete beams the absolute value of the variable SCOOR in the SECTION_BEAM input should be set to a value of 2.0, which causes the local r-axis to be aligned along the two nodes of the beam to give physically correct behavior. The distance between the nodes of a beam should not affect the behavior of this model. A triad is used to orient the beam for the directional springs. Translational/rotational stiffness and viscous damping effects are considered for a local cartesian system, see notes below. Applications for this element include the modeling of joint stiffnesses.

Card 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. For finite length discrete beams the absolute value of the variable SCOOR in the SECTION_BEAM input should be set to a value of 2.0, which causes the local r-axis to be aligned along the two nodes of the beam to give physically correct behavior. The distance between the nodes of a beam should not affect the behavior of this material model. A triad is used to orient the beam for the directional springs. Arbitrary curves to model transitional/ rotational stiffness and damping effects are allowed. See notes below.

Card 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		

<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 definition.
LCIDTR	Load curve ID defining translational force resultant along local r-axis versus relative translational displacement, see Remarks and Figure 20.19.
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 20.19 depicts a typical load curve for a force resultant. Load curves used for determining the damping forces and moment resultants always act identically in tension and compression, since only the positive quadrant values are considered, i.e., start the load curve at the origin [0,0].

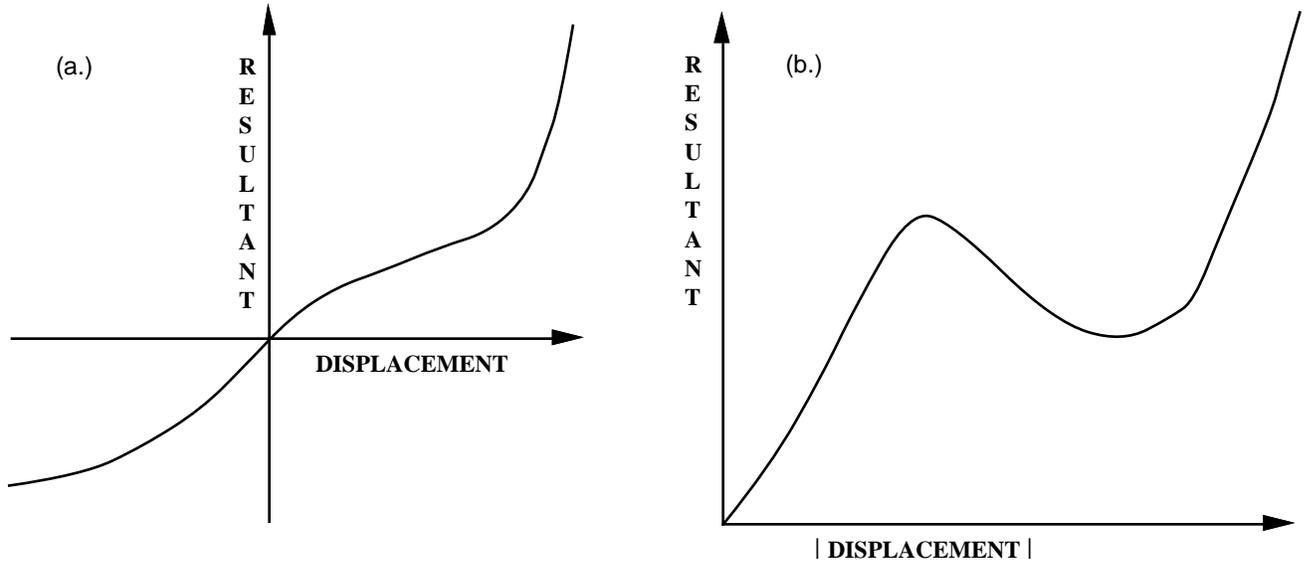


Figure 20.19. The resultant forces and moments are determined by a table lookup. If the origin of the load curve is at [0,0] as in (b.) and tension and compression responses are symmetric.

***MAT_NONLINEAR_PLASTIC_DISCRETE_BEAM**

This is Material Type 68. This material model is defined for simulating the effects of nonlinear elastoplastic, linear viscous behavior of beams by using six springs each acting about one of the six local degrees-of-freedom. The two nodes defining a beam may be coincident to give a zero length beam, or offset to give a finite length beam. For finite length discrete beams the absolute value of the variable SCOOR in the SECTION_BEAM input should be set to a value of 2.0, which causes the local r-axis to be aligned along the two nodes of the beam to give physically correct behavior. The distance between the nodes of a beam should not affect the behavior of this material model. A triad is used to orient the beam for the directional springs. Translational/rotational stiffness and damping effects can be considered. The plastic behavior is 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							

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

VARIABLE	DESCRIPTION
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.

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MFAILS	Optional failure parameter. If zero, the corresponding moment, M_s , is not considered in the failure calculation.
MFAILT	Optional failure parameter. If zero, the corresponding moment, M_t , is not considered in the failure calculation.
UFAILR	Optional failure parameter. If zero, the corresponding displacement, u_r , is not considered in the failure calculation.
UFAILS	Optional failure parameter. If zero, the corresponding displacement, u_s , is not considered in the failure calculation.
UFAILT	Optional failure parameter. If zero, the corresponding displacement, u_t , is not considered in the failure calculation.
TFAILR	Optional failure parameter. If zero, the corresponding rotation, θ_r , is not considered in the failure calculation.
TFAILS	Optional failure parameter. If zero, the corresponding rotation, θ_s , is not considered in the failure calculation.
TFAILT	Optional failure parameter. If zero, the corresponding rotation, θ_t , is not considered in the failure calculation.

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.

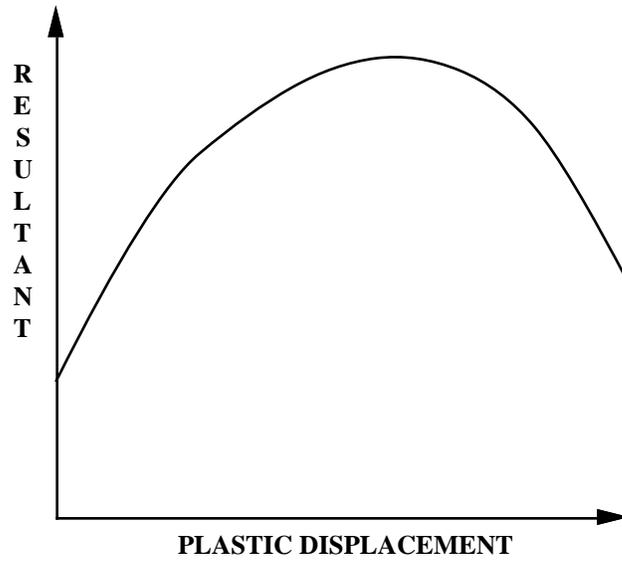


Figure 20.20. The resultant forces and moments are limited by the yield definition. The initial yield point corresponds to a plastic displacement of zero.

*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 20.21 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	ρ_{fluid} , fluid density
C1	C_1 , coefficient for linear velocity term
C2	C_2 , coefficient for quadratic velocity term
LCIDF	Load curve number ID defining force versus piston displacement, s, i.e., term $f(s + s_0)$. Compressive behavior is defined in the positive quadrant of the force displacement curve. Displacements falling outside of the defined force displacement curve are extrapolated. Care must be taken to ensure that extrapolated values are reasonable.
LCIDD	Load curve number ID defining damping coefficient versus piston displacement, s, i.e., $g(s + s_0)$. Displacements falling outside the defined curve are extrapolated. Care must be taken to ensure that extrapolated values are reasonable.
S0	Initial displacement s_0 , typically set to zero. A positive displacement corresponds to compressive behavior.
ORFLOC	d_i , orifice location of ith orifice relative to the fixed end.
ORFRAD	r_i , orifice radius of ith orifice, if zero the default radius is used.

VARIABLE	DESCRIPTION
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 20.21 the piston gradually blocks off and effectively closes the orifices. The number of orifices and the size of their opening control the damper resistance and performance. The damping force is computed from,

$$F = SF \left\{ KA_p V_p \left\{ \frac{C_1}{A_0^t} + C_2 |V_p| \rho_{fluid} \left[\left(\frac{A_p}{CA_0^t} \right)^2 - 1 \right] \right\} - f(s + s_0) + V_p g(s + s_0) \right\}$$

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 , ρ_{fluid} is the fluid density, C_1 is the coefficient for the linear term, and C_2 is the coefficient for the quadratic term.

In the implementation, the orifices are assumed to be circular with partial covering by the orifice controller. As the piston closes, the closure of the orifice is gradual. This gradual closure is properly taken into account to insure a smooth response. If the piston stroke is exceeded, the stiffness value, k , limits further movement, i.e., if the damper bottoms out in tension or compression the damper forces are calculated by replacing the damper by a bottoming out spring and damper, k and c , respectively. The piston stroke must exceed the initial length of the beam element. The time step calculation is based in part on the stiffness value of the bottoming out spring. A typical force versus displacement curve at constant relative velocity is shown in Figure 20.22.

The factor, SF , which scales the force defaults to 1.0 and is analogous to the adjusting ring on the damper.

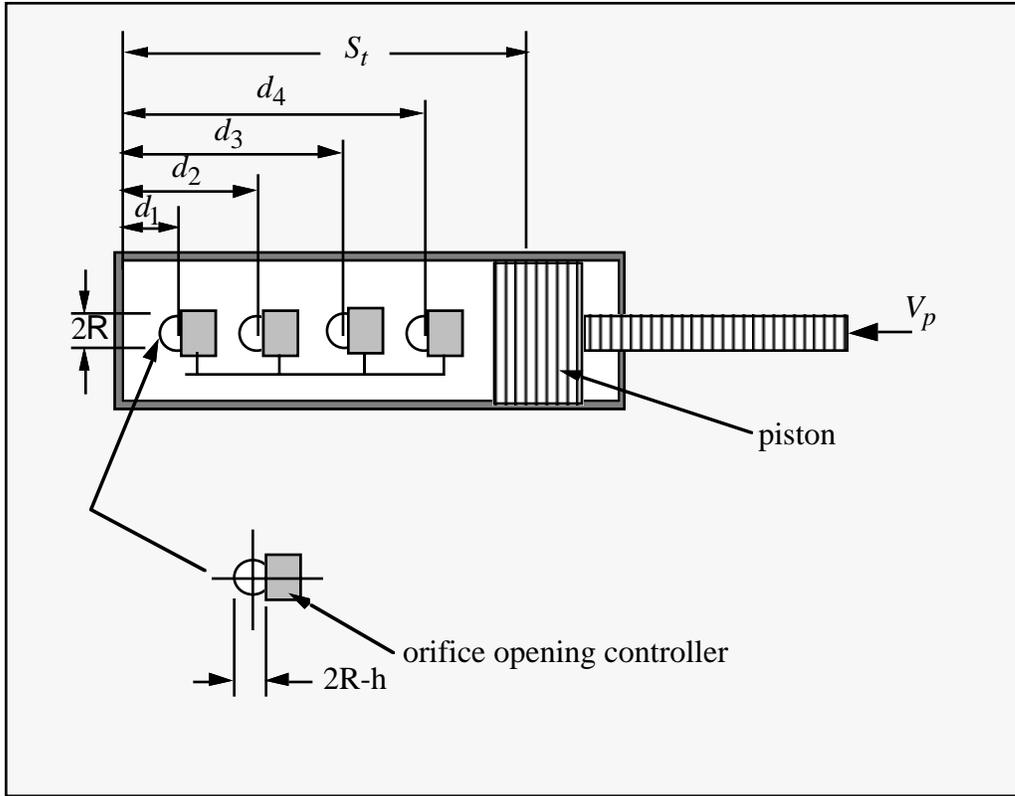


Figure 20.21. Mathematical model for the Side Impact Dummy damper.

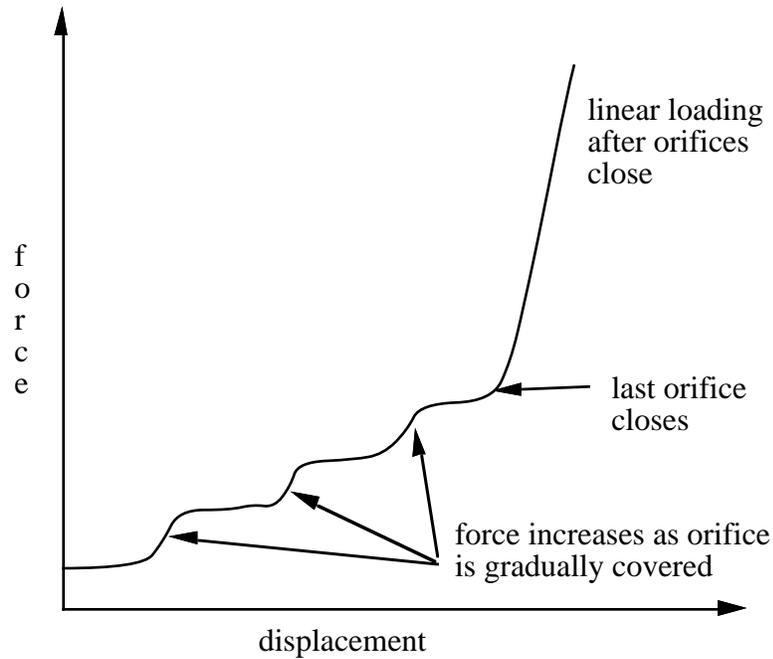


Figure 20.22. Force versus displacement as orifices are covered at a constant relative velocity. Only the linear velocity term is active.

*MAT_HYDRAULIC_GAS_DAMPER_DISCRETE_BEAM

This is Material Type 70. This special purpose element represents a combined hydraulic and gas-filled damper which has a variable orifice coefficient. A schematic of the damper is shown in Figure 20.23. Dampers of this type are sometimes used on buffers at the end of railroad tracks and as aircraft undercarriage shock absorbers. This material can be used only as a discrete beam element. 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

<u>VARIABLE</u>	<u>DESCRIPTION</u>
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).

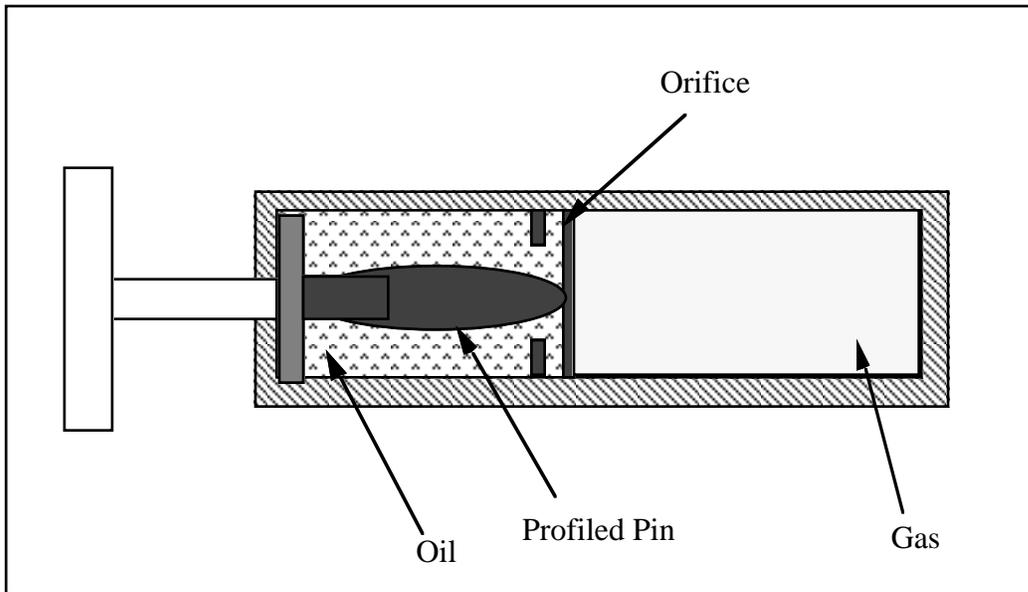


Figure 20.23. Schematic of Hydraulic/Gas damper.

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			

<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 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 ΔL 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	λ_1	λ_2	λ_3	λ_4	λ_5	λ_6	λ_7	λ_8
Type	F	F	F	F	F	F	F	F
Default	none							

Card 6

Variable	λ_9	λ_{10}	λ_{11}	λ_{12}	λ_{13}			
Type	F	F	F	F	F			
Default	none	none	none	none	none			

Card 7

1 2 3 4 5 6 7 8

Variable	η_1	η_2	η_3	η_4	η_5	η_6	η_7	η_8
Type	F	F	F	F	F	F	F	F
Default	none							

Card 8

Variable	η_9	η_{10}	η_{11}	η_{12}	η_{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.
λ_1 – λ_{13}	Tabulated damage function
η_1 – η_{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 for Modeling Low Density Urethane Foam with high compressibility and with rate sensitivity. which can be characterized by a relaxation curve. Its main applications are for seat cushions, padding on the Side Impact Dummies (SID), bumpers, and interior foams. Optionally, a tension cut-off failure can be defined. Also, see the notes below and the description of material 57: *MAT_LOW_DENSITY_FOAM.

Card 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 define the following viscoelastic constants. Up to 6 cards may be input. A keyword card (with a “*” in column 1) terminates this input if less than 6 cards are used. If **LCID2** is nonzero skip this input. The variable **REF** is taken from the first card of this sequence.

Optional Cards	1	2	3	4	5	6	7	8
Variable	GI	BETAI	REF					
Type	F	F	F					

<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 20.16.
BETA	β , decay constant to model creep in unloading. EQ:0 No relaxation.
DAMP	Viscous coefficient (.05< recommended value <.50) to model damping effects. LT.0.0: DAMP is the load curve ID, which defines the damping constant as a function of the maximum strain in compression defined as: $\epsilon_{\max} = \max(1 - \lambda_1, 1 - \lambda_2, 1 - \lambda_3).$ In tension, the damping constant is set to the value corresponding to the strain at 0. The abscissa should be defined from 0 to 1.
SHAPE	Shape factor for unloading. Active for nonzero values of the hysteretic unloading factor. Values less than one reduces the energy dissipation and greater than one increases dissipation, see also Figure 20.16.
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.

VARIABLE	DESCRIPTION
BVFLAG	Bulk viscosity activation flag, see remark below: EQ.0.0: no bulk viscosity (recommended), EQ.1.0: bulk viscosity active.
KCON	Stiffness coefficient for contact interface stiffness. Maximum slope in stress vs. strain curve is used. When the maximum slope is taken for the contact, the time step size for this material is reduced for stability. In some cases Δt may be significantly smaller, and defining a reasonable stiffness is recommended.
LCID2	Load curve ID of relaxation curve. If constants β_i are determined via a least squares fit. This relaxation curve is shown in Figure 20.25. This model ignores the constant stress.
BSTART	Fit parameter. In the fit, β_1 is set to zero, β_2 is set to BSTART, β_3 is 10 times β_2 , β_4 is 100 times greater than β_3 , and so on. If zero, BSTART=.01.
TRAMP	Optional ramp time for loading.
NV	Number of terms in fit. If zero, the default is 6. Currently, the maximum number is set to 6. Values of 2 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, σ_{ij}^r , augments the stresses determined from the foam, σ_{ij}^f ; consequently, the final stress, σ_{ij} , is taken as the summation of the two contributions:

$$\sigma_{ij} = \sigma_{ij}^f + \sigma_{ij}^r.$$

Since we wish to include only simple rate effects, the relaxation function is represented by up to six terms of the Prony series:

$$g(t) = \alpha_0 + \sum_{m=1}^N \alpha_m e^{-\beta t}$$

This model is effectively a Maxwell fluid which consists of a dampers and springs in series. The formulation is performed in the local system of principal stretches where only the principal values of stress are computed and triaxial coupling is avoided. Consequently, the one-dimensional nature of this foam material is unaffected by this addition of rate effects. The addition of rate effects necessitates 42 additional history variables per integration point. The cost and memory overhead of this model comes primarily from the need to “remember” the local system of principal stretches and the evaluation of the viscous stress components.

Additional Remarks:

1. When hysteretic unloading is used the reloading will follow the unloading curve if the decay constant, β , is set to zero. If β is nonzero the decay to the original loading curve is governed by the expression:

$$1 - e^{-\beta t}$$

2. The bulk viscosity, which generates a rate dependent pressure, may cause an unexpected volumetric response and, consequently, it is optional with this model.
3. The hysteretic unloading factor results in the unloading curve to lie beneath the loading curve as shown in Figure 20.16. This unloading provide energy dissipation which is reasonable in certain kinds of foam.

*MAT_ELASTIC_SPRING_DISCRETE_BEAM

This is Material Type 74. This model permits elastic springs with damping to be combined and represented with a discrete beam element type 6. Linear stiffness and damping coefficients can be defined, and, for nonlinear behavior, a force versus deflection and force versus rate curves can be used. Displacement based failure and an initial force are optional

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	MID	RO	K	F0	D	CDF	TDF	
Type	I	F	F	F	F	F	F	

Card 2 1 2 3 4 5 6 7 8

Variable	FLCID	HLCID	C1	C2	DLE	GLCID		
Type	F	F	F	F	F	I		

VARIABLE

DESCRIPTION

MID	Material identification. A unique number has to be chosen.
RO	Mass density, see also volume in *SECTION_BEAM definition.
K	Stiffness coefficient.
F0	Optional initial force. This option is inactive if this material is referenced in a part referenced by material type *MAT_ELASTIC_6DOF_SPRING
D	Viscous damping coefficient.
CDF	Compressive displacement at failure. Input as a positive number. After failure, no forces are carried. This option does not apply to zero length springs. EQ.0.0: inactive.
TDF	Tensile displacement at failure. After failure, no forces are carried.
FLCID	Load curve ID, see *DEFINE_CURVE, defining force versus deflection for nonlinear behavior.

VARIABLE	DESCRIPTION
HLCID	Load curve ID, see *DEFINE_CURVE, defining force versus relative velocity for nonlinear behavior (optional). If the origin of the curve is at (0,0) the force magnitude is identical for a given magnitude of the relative velocity, i.e., only the sign changes.
C1	Damping coefficient for nonlinear behavior (optional).
C2	Damping coefficient for nonlinear behavior (optional).
DLE	Factor to scale time units. The default is unity.
GLCID	Optional load curve ID, see *DEFINE_CURVE, defining a scale factor versus deflection for load curve ID, HLCID. If zero, a scale factor of unity is assumed.

Remarks:

If the linear spring stiffness is used, the force, F , is given by:

$$F = F_0 + K\Delta L + D\Delta\dot{L}$$

but if the load curve ID is specified, the force is then given by:

$$F = F_0 + K f(\Delta L) \left[1 + C1 \cdot \Delta\dot{L} + C2 \cdot \text{sgn}(\Delta\dot{L}) \ln \left(\max \left\{ 1, \frac{|\Delta\dot{L}|}{DLE} \right\} \right) \right] + D\Delta\dot{L} + g(\Delta L)h(\Delta\dot{L})$$

In these equations, ΔL is the change in length

$$\Delta L = \text{current length} - \text{initial length}$$

The cross sectional area is defined on the section card for the discrete beam elements, See *SECTION_BEAM. The square root of this area is used as the contact thickness offset if these elements are included in the contact treatment.

*MAT_BILKHU/DUBOIS_FOAM

This is Material Type 75. This model is for the simulation of isotropic crushable forms. Uniaxial and triaxial test data 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	PC	VPC
Type	I	F	F	F	F	F	F	F

Card 2

Variable	TSC	VTSC	LCRATE	PR				
Type	I	F	F	F				

<u>VARIABLE</u>	<u>DESCRIPTION</u>
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 20.24.
LCUYS	Load curve ID giving uniaxial yield stress versus volumetric strain, see Figure 20.24.
VC	Viscous damping coefficient (.05<recommended value<.50).
PC	Pressure cutoff. If zero, the default is set to one-tenth of p_0 , the yield pressure corresponding to a volumetric strain of zero.
VPC	Variable pressure cutoff as a fraction of pressure yield value. If non-zero this will override the pressure cutoff value PC.
TC	Tension cutoff for uniaxial tensile stress. Default is zero.
VTC	Variable tension cutoff as a fraction of the uniaxial compressive yield strength, if non-zero this will override the tension cutoff value TC.

VARIABLE	DESCRIPTION
LCRATE	Load curve ID giving a scale factor for the previous yield curves, dependent upon the volumetric plastic strain.
PR	Poisson coefficient, which applies to both elastic and plastic deformations

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.

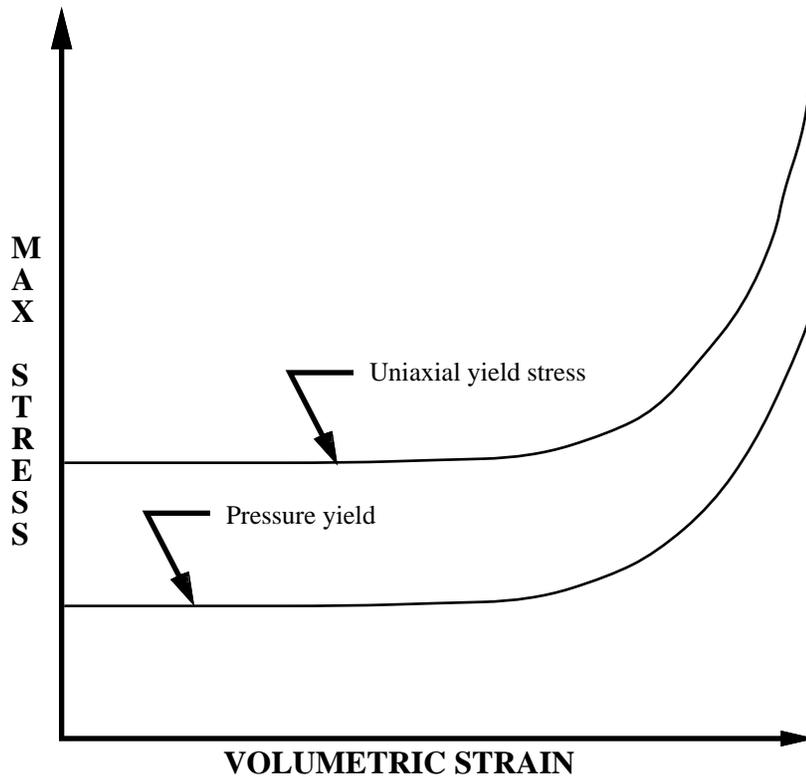


Figure 20.24. Behavior of crushable foam. Unloading is elastic.

The yield surface is defined as an ellipse in the equivalent pressure and von Mises stress plane.

*MAT_GENERAL_VISCOELASTIC

This is Material Type 76. This material model provides a general viscoelastic Maxwell model having up to 6 terms in the prony series expansion and is useful for modeling dense continuum rubbers and solid explosives. Either the coefficients of the prony series expansion or a relaxation curve may be specified to define the viscoelastic deviatoric and bulk behavior.

The material model can also be used with laminated shell. Either an elastic or viscoelastic layer can be defined with the laminated formulation. To activate laminated shell you need the laminated formulation flag on *CONTROL_SHELL. With the laminated option a userdefined integration rule is needed.

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	MID	RO	BULK	PCF	EF			
Type	I	F	F	F	F			

Insert a blank card here if constants are defined on cards 3,4,... below.

If an elastic layer is defined in a laminated shell this card must be blank.

Card 2 1 2 3 4 5 6 7 8

Variable	LCID	NT	BSTART	TRAMP	LCIDK	NTK	BSTARTK	TRAMPK
Type	F	I	F	F	F	I	F	F

Card Format for viscoelastic constants. Up to 6 cards may be input. A keyword card (with a "*" in column 1) terminates this input if less than 6 cards are used. These cards are not needed if relaxation data is defined. The number of terms for the shear behavior may differ from that for the bulk behavior: simply insert zero if a term is not included.

If an elastic layer is defined you only need to defined GI and KI (note in an elastic layer only one card is needed)

Optional Cards 1 2 3 4 5 6 7 8

Variable	GI	BETAI	KI	BETAKI				
Type	F	F	F	F				

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
BULK	Elastic bulk modulus.
PCF	Tensile pressure elimination flag for solid elements only. If set to unity tensile pressures are set to zero.
EF	Elastic flag (if equal 1, the layer is elastic. If 0 the layer is viscoelastic).
LCID	Load curve ID for deviatoric behavior if constants, G_i , and β_i are determined via a least squares fit. This relaxation curve is shown below.
NT	Number of terms in shear fit. If zero the default is 6. Fewer than NT terms will be used if the fit produces one or more negative shear moduli. Currently, the maximum number is set to 6.
BSTART	In the fit, β_1 is set to zero, β_2 is set to BSTART, β_3 is 10 times β_2 , β_4 is 100 times greater than β_3 , and so on. If zero, BSTART is determined by an iterative trial and error scheme.
TRAMP	Optional ramp time for loading.
LCIDK	Load curve ID for bulk behavior if constants, K_i , and $\beta\kappa_i$ are determined via a least squares fit. This relaxation curve is shown below.
NTK	Number of terms desired in bulk fit. If zero the default is 6. Currently, the maximum number is set to 6.
BSTARTK	In the fit, $\beta\kappa_1$ is set to zero, $\beta\kappa_2$ is set to BSTARTK, $\beta\kappa_3$ is 10 times $\beta\kappa_2$, $\beta\kappa_4$ is 100 times greater than $\beta\kappa_3$, and so on. If zero, BSTARTK is determined by an iterative trial and error scheme.
TRAMPK	Optional ramp time for bulk loading.
GI	Optional shear relaxation modulus for the ith term
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, β_i . An arbitrary number of terms, up to 6, may be used when applying the viscoelastic model.

For volumetric relaxation, the relaxation function is also represented by the Prony series in terms of bulk moduli:

$$k(t) = \sum_{m=1}^N K_m e^{-\beta_{k_m} t}$$

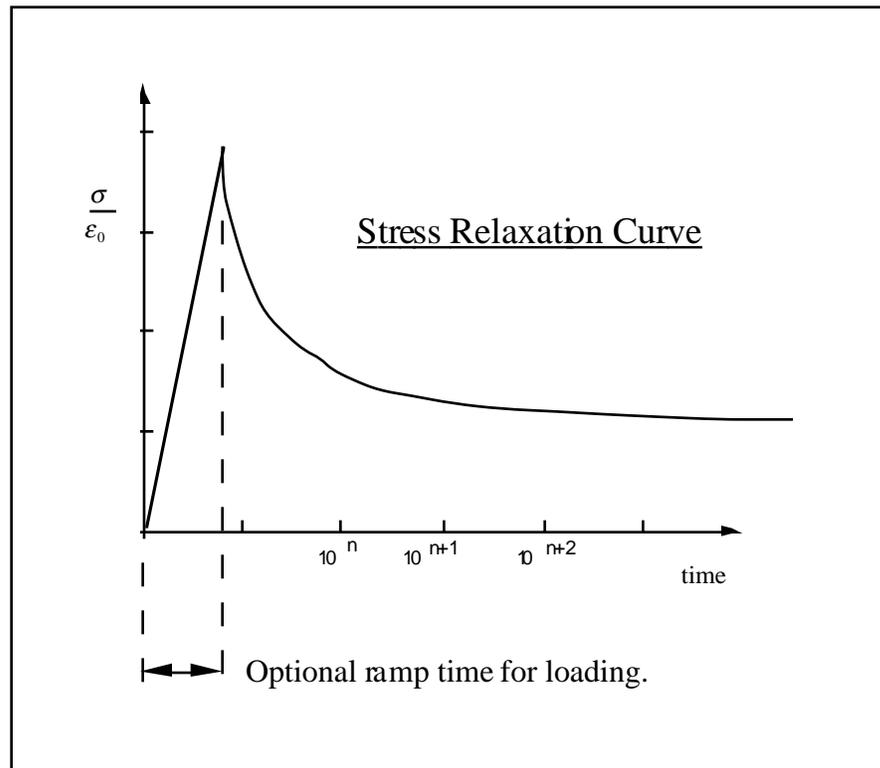


Figure 20.25. Relaxation curve. This curve defines stress versus time where time is defined on a logarithmic scale. For best results, the points defined in the load curve should be equally spaced on the logarithmic scale. Furthermore, the load curve should be smooth and defined in the positive quadrant. If nonphysical values are determined by least squares fit, LS-DYNA will terminate with an error message after the initialization phase is completed. If the ramp time for loading is included, then the relaxation which occurs during the loading phase is taken into account. This effect may or may not be important.

***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	G	SIGF	
Type	I	F	F	I	I	F	F	

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	BSTART	TRAMP
Type	F	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		

VARIABLE	DESCRIPTION
DATA	Type of experimental data. EQ.0.0: uniaxial data (Only option for this model)
LCID2	Load curve ID of relaxation curve If constants β_i are determined via a least squares fit. This relaxation curve is shown in Figure 20.25. This model ignores the constant stress.
BSTART	In the fit, β_1 is set to zero, β_2 is set to BSTART, β_3 is 10 times β_2 , β_4 is 100 times greater than β_3 , and so on. If zero, BSTART is determined by an iterative trial and error scheme.
TRAMP	Optional ramp time for loading.

If N=0, the following constants have to be defined:

C10	C10
C01	C01
C11	C11
C20	C20
C02	C02
C30	C30
GI	Optional shear relaxation modulus for the ith term
BETAI	Optional decay constant if ith term

Remarks:

Rubber is generally considered to be fully incompressible since the bulk modulus greatly exceeds the shear modulus in magnitude. To model the rubber as an unconstrained material a hydrostatic work term, $W_H(J)$, is included in the strain energy functional which is function of the relative volume, J , [Ogden, 1984]:

$$W(J_1, J_2, J) = \sum_{p,q=0}^n C_{pq} (J_1 - 3)^p (J_2 - 3)^q + W_H(J)$$

$$J_1 = I_1 J^{-1/3}$$

$$J_2 = I_2 J^{-2/3}$$

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:

$$g(t) = \alpha_0 + \sum_{m=1}^N \alpha_m e^{-\beta t}$$

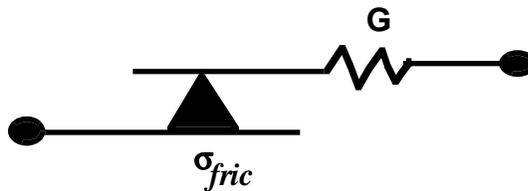
given by,

$$g(t) = \sum_{i=1}^n G_i e^{-\beta_i t}$$

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, β_i . The viscoelastic behavior is optional and an arbitrary number of terms may be used.

The Mooney-Rivlin rubber model (model 27) is obtained by specifying $n=1$. In spite of the differences in formulations with Model 27, we find that the results obtained with this model are nearly identical with those of Material 27 as long as large values of Poisson's ratio are used.

The frequency independent damping is obtained by the having a spring and slider in series as shown in the following sketch:



***MAT_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	G	SIGF	
Type	I	F	F	I	I	F	F	

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	BSTART	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 1 2 3 4 5 6 7 8
Cards

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.
- G Shear modulus for frequency independent damping. Frequency independent damping is based of a spring and slider in series. The critical stress for the slider mechanism is SIGF defined below. For the best results, the value of G should be 250-1000 times greater than SIGF.
- SIGF Limit stress for frequency independent, frictional, damping.

<u>VARIABLE</u>	<u>DESCRIPTION</u>
if N>0 test information from a uniaxial test are used:	
SGL	Specimen gauge length
SW	Specimen width
ST	Specimen thickness
LCID1	Load curve ID giving the force versus actual change in the gauge length
DATA	Type of experimental data. EQ.1.0: uniaxial data (default) EQ.2.0: biaxial data
LCID2	Load curve ID of relaxation curve If constants β_l are determined via a least squares fit. This relaxation curve is shown in Figure 20.25. This model ignores the constant stress.
BSTART	In the fit, β_1 is set to zero, β_2 is set to BSTART, β_3 is 10 times β_2 , β_4 is 100 times greater than β_3 , and so on. If zero, BSTART is determined by an iterative trial and error scheme.
TRAMP	Optional ramp time for loading. If N=0, the constants μ_i and α_i have to be defined:
μ_i	μ_i , the <i>i</i> th shear modulus, <i>i</i> varies up to 8. See discussion below.
α_i	α_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
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 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 be eliminated from the principal stretches, λ_j^* . The number of terms, *n*, is 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_{ij} , and Green's strain tensor, E_{ij} ,

$$S_{ij} = \int_0^t G_{ijkl}(t-\tau) \frac{\partial E_{kl}}{\partial \tau} d\tau$$

where $g_{ijkl}(t-\tau)$ and $G_{ijkl}(t-\tau)$ are the relaxation functions for the different stress measures. This stress is added to the stress tensor determined from the strain energy functional.

If we wish to include only simple rate effects, the relaxation function is represented by six terms from the Prony series:

$$g(t) = \alpha_0 + \sum_{m=1}^N \alpha_m e^{-\beta t}$$

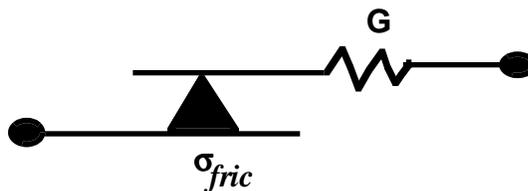
given by,

$$g(t) = \sum_{i=1}^n G_i e^{-\beta_i t}$$

This model is effectively a Maxwell fluid which consists of a dampers and springs in series. We characterize this in the input by shear moduli, G_i , and decay constants, β_i . The viscoelastic behavior is optional and an arbitrary number of terms may be used.

The Mooney-Rivlin rubber model (model 27) is obtained by specifying $n=1$. In spite of the differences in formulations with Model 27, we find that the results obtained with this model are nearly identical with those of Material 27 as long as large values of Poisson's ratio are used.

The frequency independent damping is obtained by the having a spring and slider in series as shown in the following sketch:



***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 20.26.
FAIL	Flag for failure: EQ.0: no failure, EQ.1: When pressure reaches failure pressure element is eroded, EQ.2: When pressure reaches failure pressure element loses it ability to carry tension.

Remarks:

Pressure is positive in compression. Volumetric strain is defined as the natural log of the relative volume and is *positive* in compression where the relative volume, V, is the ratio of the current volume to the initial volume. The tabulated data should be given in order of increasing compression. If the pressure drops below the cutoff value specified, it is reset to that value and the deviatoric stress state is eliminated.

If the load curve ID (LCYP) is provided as a positive number, the deviatoric, perfectly plastic, pressure dependent, yield function ϕ , is given as

$$\phi = \sqrt{3J_2} - F(p) = \sigma_y - F(p)$$

where , $F(p)$ is a tabulated function of yield stress versus pressure, and the second invariant, J_2 , is defined in terms of the deviatoric stress tensor as:

$$J_2 = \frac{1}{2} S_{ij} S_{ij}$$

assuming that . If the ID is given as negative then the yield function becomes:

$$\phi = J_2 - F(p)$$

being the deviatoric stress tensor.

If cracking is invoked by setting the residual strength factor, B, on card 2 to a value between 0.0 and 1.0, the yield stress is multiplied by a factor f which reduces with plastic strain according to a trilinear law as shown in Figure 20.26.

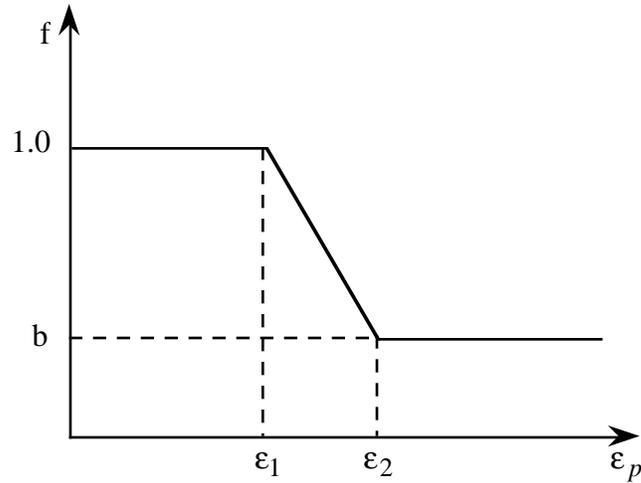


Figure 20.26. Strength reduction factor.

b = residual strength factor

ϵ_1 = plastic stain at which cracking begins.

ϵ_2 = plastic stain at which residual strength is reached.

ϵ_1 and ϵ_2 are tabulated function of pressure that are defined by load curves, see Figure 20.27. The values on the curves are pressure versus strain and should be entered in order of increasing pressure. The strain values should always increase monotonically with pressure.

By properly defining the load curves, it is possible to obtain the desired strength and ductility over a range of pressures, see Figure 20.28.

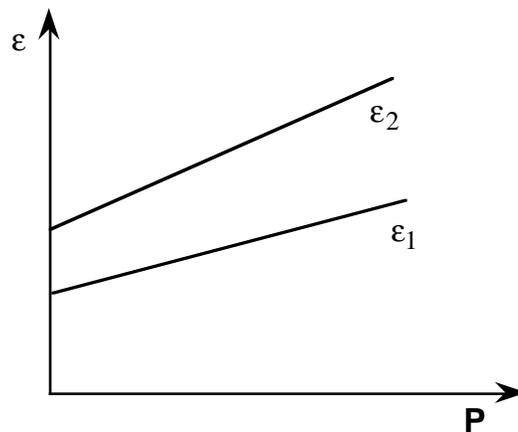


Figure 20.27. Cracking strain versus pressure.

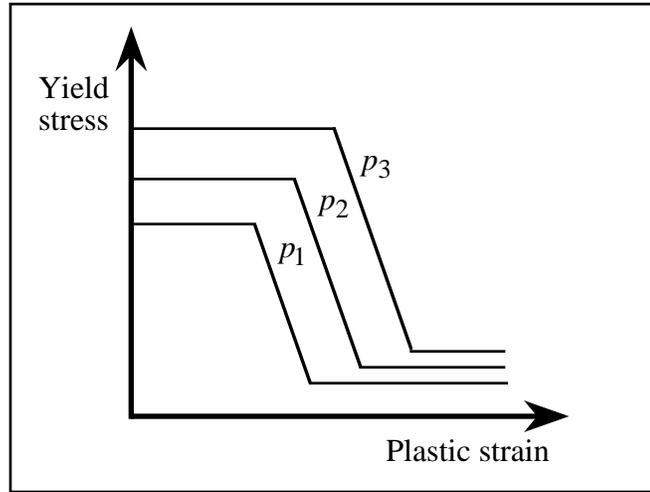


Figure 20.28.

***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	γ_1 , shear strain (ignored if LCID is non zero).
GAM2	γ_2 , shear strain (ignored if LCID is non zero).
GAM3	γ_3 , shear strain (ignored if LCID is non zero).
GAM4	γ_4 , shear strain (ignored if LCID is non zero).
GAM5	γ_5 , shear strain (ignored if LCID is non zero).

<u>VARIABLE</u>	<u>DESCRIPTION</u>
TAU1	τ_1 , shear stress at γ_1 (ignored if LCID is non zero).
TAU2	τ_2 , shear stress at γ_2 (ignored if LCID is non zero).
TAU3	τ_3 , shear stress at γ_3 (ignored if LCID is non zero).
TAU4	τ_4 , shear stress at γ_4 (ignored if LCID is non zero).
TAU5	τ_5 , shear stress at γ_5 (ignored if LCID is non zero).

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 γ . 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 (γ_y)
TAUY	Reference shear stress (τ_y)
ALPHA	Stress coefficient (α)
R	Stress exponent (r)
BULK	Elastic bulk modulus

Remarks:

The Ramberg-Osgood equation is an empirical constitutive relation to represent the one-dimensional elastic-plastic behavior of many materials, including soils. This model allows a simple rate independent representation of the hysteretic energy dissipation observed in soils subjected to cyclic shear deformation. For monotonic loading, the stress-strain relationship is given by:

$$\frac{\gamma}{\gamma_y} = \frac{\tau}{\tau_y} + \alpha \left| \frac{\tau}{\tau_y} \right|^r \quad \text{if } \gamma \geq 0$$

$$\frac{\gamma}{\gamma_y} = \frac{\tau}{\tau_y} - \alpha \left| \frac{\tau}{\tau_y} \right|^r \quad \text{if } \gamma < 0$$

where γ is the shear and τ 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 γ_0 and τ_0 represent the values of strain and stress at the point of load reversal. Subsequent load reversals are detected by $(\gamma - \gamma_0)\dot{\gamma} < 0$.

The Ramberg-Osgood equations are inherently one-dimensional and are assumed to apply to shear components. To generalize this theory to the multidimensional case, it is assumed that each component of the deviatoric stress and deviatoric tensorial strain is independently related by the one-dimensional stress-strain equations. A projection is used to map the result back into deviatoric stress space if required. The volumetric behavior is elastic, and, therefore, the pressure p is found by

$$p = -K\varepsilon_v$$

where ε_v is the volumetric strain.

***MAT_PLASTICITY_WITH_DAMAGE_{OPTION}**

This is Material Types 81 and 82. An elasto-visco-plastic material with an arbitrary stress versus strain curve and arbitrary strain rate dependency can be defined. Damage is considered before rupture occurs. Also, failure based on a plastic strain or a minimum time step size can be defined.

Options include:

<BLANK>

ORTHO

Including ORTHO invokes an orthotropic damage model. It is implemented only for shell elements with multiple through thickness integration points and is an extension to include orthotropic damage as a means of treating failure in aluminum panels. Directional damage begins after a defined failure strain is reached in tension and continues to evolve until a tensile rupture strain is reached in either one of the two orthogonal directions. After rupture is detected at all integration points, the element is deleted.

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	NUMINT
Type	F	F	F	F	F	F	F	I
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

<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.
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.

<u>VARIABLE</u>	<u>DESCRIPTION</u>
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;
LCSR	Load curve ID defining strain rate scaling effect on yield stress.
ES1-ES8	Corresponding yield stress values to EPS1 - EPS8.
EPPFR	Plastic strain at which material ruptures (logrithmic).
VP	Formulation for rate effects: EQ.0.0: Scale yield stress (default), EQ.1.0: Viscoplastic formulation.
LCDM	Load curve ID defining nonlinear damage curve.
NUMINT	Number of through thickness integration points which must fail before the element is deleted. (If zero, all points must fail.) The default of all integration points is not recommended since elements undergoing large strain are often not deleted due to nodal fiber rotations which limit strains at active integration points after most points have failed. Better results are obtained if NUMINT is set to 1 or a number less than one half of the number of through thickness points. For example, if four through thickness points are used, NUMINT should not exceed 2, even for fully integrated shells which have 16 integration points.
EPS1-EPS8	Effective plastic strain values (optional if SIGY is defined). At least 2 points should be defined.

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, ω , which varies from zero if no damage has occurred to unity for complete rupture. For uniaxial loading, the nominal stress in the damaged material is given by

$$\sigma_{nominal} = \frac{P}{A}$$

where P is the applied load and A is the surface area. The true stress is given by:

$$\sigma_{true} = \frac{P}{A - A_{loss}}$$

where A_{loss} is the void area. The damage variable can then be defined:

$$\omega = \frac{A_{loss}}{A} \quad 0 \leq \omega \leq 1$$

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.

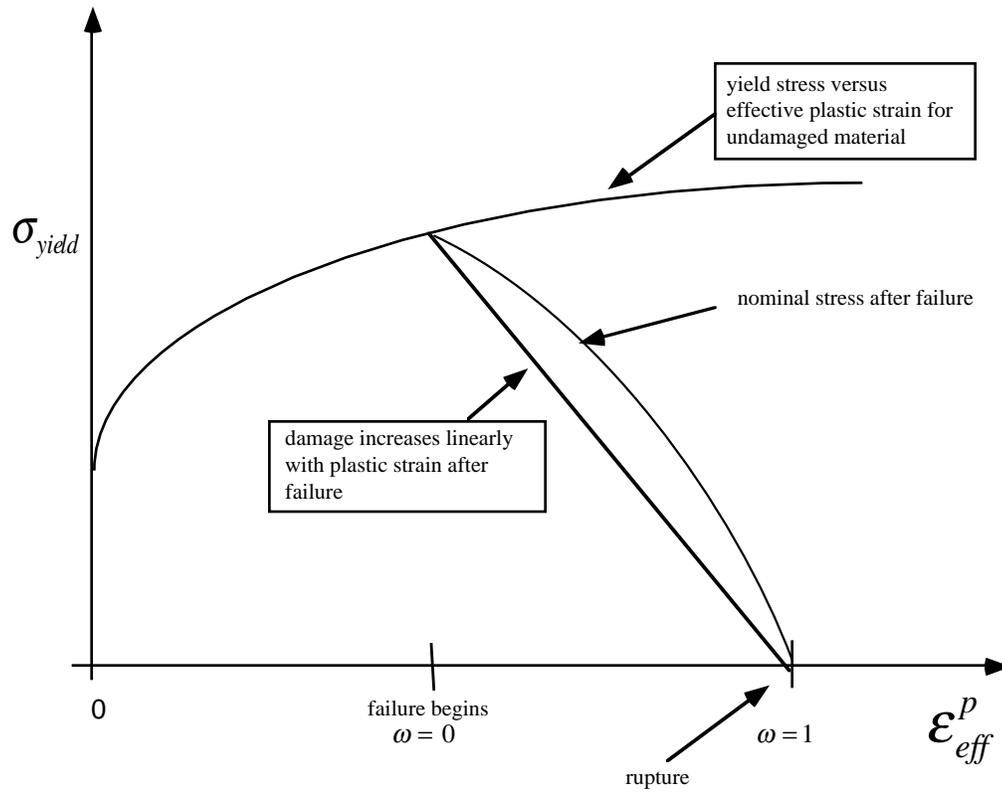


Figure 20.29. Stress strain behavior when damage is included.

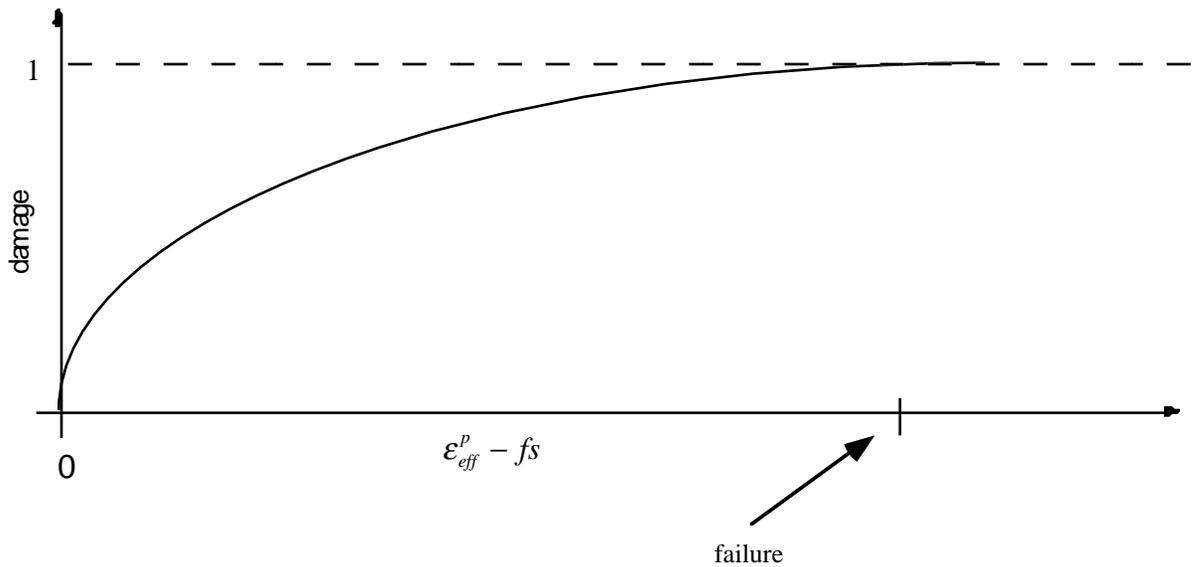


Figure 20.30. A nonlinear damage curve is optional. Note that the origin of the curve is at (0,0). It is permissible to input the failure strain, f_s , as zero for this option. The nonlinear damage curve is useful for controlling the softening behavior after the failure strain is reached.

***MAT_FU_CHANG_FOAM**

This is Material Type 83. Rate effects can be modelled in low and medium density foams, see Figure 20.31. 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	SRAF		
Type	F	F	F	F	F	F		
Default	1.0	0.0	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	

<u>VARIABLE</u>	<u>DESCRIPTION</u>
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 Δt may be significantly smaller, and defining a reasonable stiffness is recommended.
TC	Tension cut-off stress
FAIL	Failure option after cutoff stress is reached: EQ.0.0: tensile stress remains at cut-off value, EQ.1.0: tensile stress is reset to zero.
DAMP	Viscous coefficient (.05< recommended value <.50) to model damping effects.

<u>VARIABLE</u>	<u>DESCRIPTION</u>
TBID	Table ID, see *DEFINE_TABLE, for nominal stress vs. strain data as a function of strain rate. If the table ID is provided, cards 3 and 4 may be left blank and the fit will be done internally.
BVFLAG	Bulk viscosity activation flag, see remark below: EQ.0.0: no bulk viscosity (recommended), EQ.1.0: bulk viscosity active.
SFLAG	Strain rate flag (see remark 2 below): EQ.0.0: true constant strain rate, EQ.1.0: engineering strain rate.
RFLAG	Strain rate evaluation flag: EQ.0.0: first principal direction, EQ.1.0: principal strain rates for each principal direction, EQ.2.0: volumetric strain rate.
TFLAG	Tensile stress evaluation: EQ.0.0: linear in tension. EQ.1.0: input via load curves with the tensile response corresponds to negative values of stress and strain.
PVID	Optional load curve ID defining pressure versus volumetric strain.
SRAF	Strain rate averaging flag. EQ.0.0: use weighted running average. EQ.1.0: average the last twelve values.
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_{ij}</i> ,	material constant, see equations below.

<u>VARIABLE</u>	<u>DESCRIPTION</u>
<i>SIJ</i>	material constant, see equations below.
MINR	Ratemin, minimum strain rate of interest.
MAXR	Ratemax, maximum strain rate of interest.

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 τ in $T_t : 0 \leq \tau \leq \infty$ and

$$E_t^N(\tau) = E^N(t - \tau)$$

where τ 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 σ^* 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:

$$\dot{E}^N = \frac{\sigma}{\|\sigma\|} D_0 \exp \left[-c_0 \left(\frac{tr(\sigma S)}{(\|\sigma\|)^2} \right)^{2n_0} \right]$$

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.

$$\dot{S}_{ij} = \left[c_1 (a_{ij} R - c_2 S_{ij}) P + c_3 W^{n_1} (\|\dot{E}^N\|)^{n_2} I_{ij} \right] R$$

$$R = 1 + c_4 \left(\frac{\|\dot{E}^N\|}{c_5} - 1 \right)^{n_3}$$

$$P = tr(\sigma \dot{E}^N)$$

$$W = \int tr(\sigma dE)$$

where $c_1, c_2, c_3, c_4, c_5, n_1, n_2, n_3$, and a_{ij} are material constants and:

$$\|\sigma\| = (\sigma_{ij} \sigma_{ij})^{\frac{1}{2}}$$

$$\|\dot{E}\| = (\dot{E}_{ij} \dot{E}_{ij})^{\frac{1}{2}}$$

$$\|\dot{E}^N\| = (\dot{E}_{ij}^N \dot{E}_{ij}^N)^{\frac{1}{2}}$$

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.

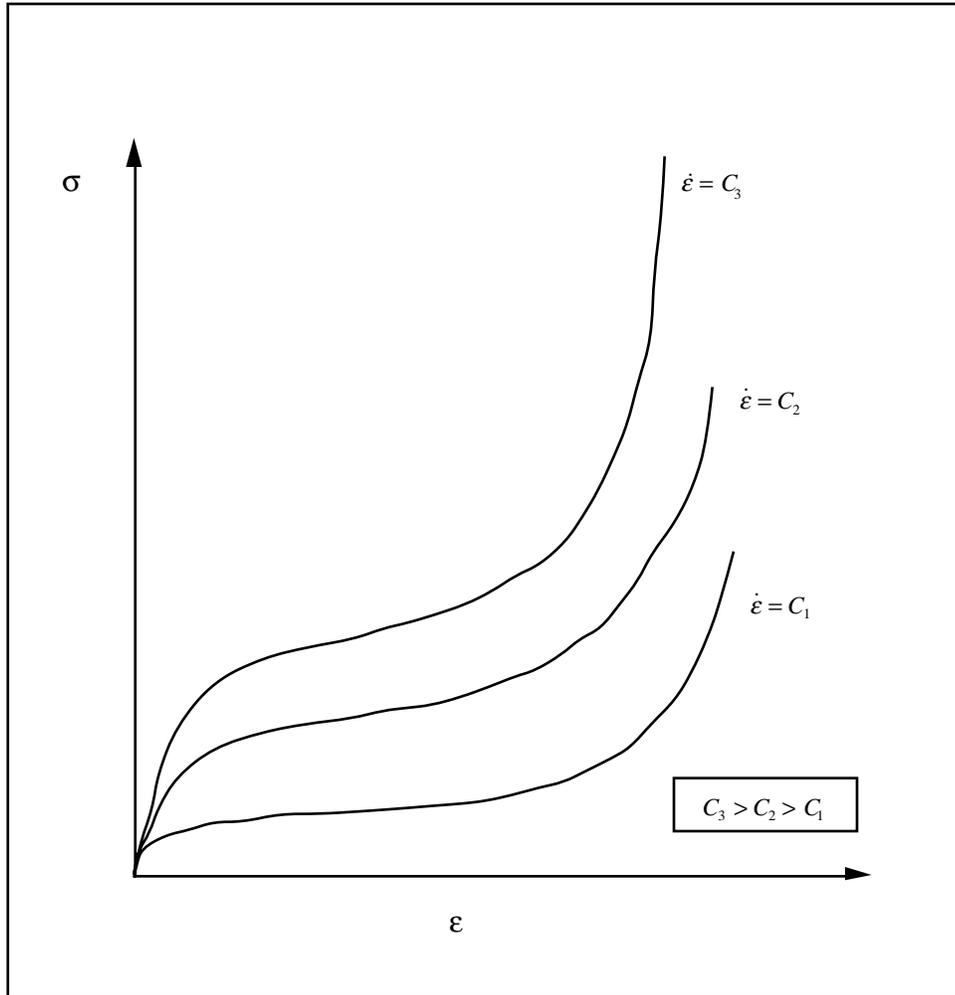


Figure 20.31. Nominal stress versus engineering strain curves, which are used to model rate effects in Fu Chang's foam model.

Additional Remarks:

1. The bulk viscosity, which generates a rate dependent pressure, may cause an unexpected volumetric response and, consequently, it is optional with this model.
2. Dynamic compression tests at the strain rates of interest in vehicle crash are usually performed with a drop tower. In this test the loading velocity is nearly constant but the true strain rate, which depends on the instantaneous specimen thickness, is not. Therefore, the engineering strain rate input is optional so that the stress strain curves obtained at constant velocity loading can be used directly.
3. To further improve the response under multiaxial loading, the strain rate parameter can either be based on the principal strain rates or the volumetric strain rate.
4. Correlation under triaxial loading is achieved by directly inputting the results of hydrostatic testing in addition to the uniaxial data. Without this additional information which is fully optional, triaxial response tends to be underestimated.

***MAT_WINFRITH_CONCRETE**

This is Material Type 84 and Material Type 85, only the former of which includes rate effects. The Winfrith concrete model is a smeared crack (sometimes known as pseudo crack), smeared rebar model, implemented in the 8-node single integration point continuum element. This model was developed by Broadhouse and Neilson [1987], and Broadhouse [1995] over many years and has been validated against experiments. The input documentation given here is taken directly from the report by Broadhouse. The Fortran subroutines and quality assurance test problems were also provided to LSTC by the Winfrith Technology Center. The rebar is defined in the section: *MAT_WINFRITH_CONCRETE_REINFORCEMENT which follows.

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	MID	RO	TM	PR	UCS	UTS	FE	ASIZE
Type	I	F	F	F	F	F	F	F

Card 2

Variable	E	YS	EH	UELONG	RATE	CONM	CONL	CONT
Type	F	F	F	F	F	F	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	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.
TM	Tangent modulus (concrete).
PR	Poisson's ratio.
UCS	Uniaxial compressive strength.
UTS	Uniaxial tensile strength.
FE	Depends on value of RATE below. RATE.EQ.0.: Fracture energy (energy per unit area dissipated in opening crack). RATE.EQ 1.: Crack width at which crack-normal tensile stress goes to zero.
ASIZE	Aggregate size (radius).
E	Young's modulus of rebar.
YS	Yield stress of rebar.
EH	Hardening modulus of rebar
UELONG	Ultimate elongation before rebar fails.
RATE	Rate effects: EQ.0.0: strain rate effects are included (mat 84 - recommended). EQ.1.0: strain rate effects are turned off (mat 85).
CONM	GT.0: Factor to convert model mass units to kg. EQ.-1.: Mass, length, time units in model are lbf*sec ² /in, inch, sec. EQ.-2.: Mass, length, time units in model are g, cm, microsec. EQ.-3.: Mass, length, time units in model are g, mm, msec. EQ.-4.: Mass, length, time units in model are metric ton, mm, sec. EQ.-5.: Mass, length, time units in model are kg, mm, msec.
CONL	If CONM.GT.0, factor to convert model length units to meters; otherwise CONL is ignored.
CONT	If CONM.GT.0, factor to convert model time units to seconds; otherwise CONT is ignored.
EPS1,.....	Volumetric strain values (natural logarithmic values), see comments below. A maximum of 8 values are allowed. The tabulated values must completely cover the expected values in the analysis. If the first value is not for a volumetric strain value of zero then the point (0.0,0.0) will be automatically generated and up to a further nine additional values may be defined.

VARIABLE	DESCRIPTION
P1, P2,..PN	Pressures corresponding to volumetric strain values.

Remarks:

Pressure is positive in compression; volumetric strain is given by the natural log of the relative volume and is negative in compression. The tabulated data are given in order of increasing compression, with no initial zero point.

If the volume compaction curve is omitted, the following scaled curve is automatically used where p_1 is the pressure at uniaxial compressive failure from:

$$p_1 = \frac{\sigma_c}{3}$$

and K is the bulk unloading modulus computed from

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

where E_s is one-half the input tangent modulus for concrete and ν is Poisson's ratio.

Volumetric Strain Pressure (MPa)

$-p_1/K$	$1.00xp_1$
-0.002	$1.50xp_1$
-0.004	$3.00xp_1$
-0.010	$4.80xp_1$
-0.020	$6.00xp_1$
-0.030	$7.50xp_1$
-0.041	$9.45xp_1$
-0.051	$11.55xp_1$
-0.062	$14.25xp_1$
-0.094	$25.05xp_1$

Table 20.1. Default pressure versus volumetric strain curve for concrete if the curve is not defined.

The Winfrith concrete model generates an additional binary output file containing information on crack locations, directions, and widths. In order to invoke the model and generate this file, the execution line is modified by adding:

q=crf where crf is the name of a crack file (e.g., q=DYNCRCK).

LS-TAURUS can read the crack file and display the cracks on the deformed mesh plots. The execution line is again modified by adding:

q=crf where crf is the name of a crack file (e.g., q=DYNCRCK).

Then the command:

cracks w

entered after a "time" or "state" command will cause the crack information to be read at that time and all subsequent "view" or "draw" commands will cause cracks to be superimposed on the mesh plot. The parameter "w" causes all cracks greater than width=*w* to be plotted; thereby, permitting selective crack plotting and estimation of crack sizes. If *w*=1 then all cracks are plotted regardless of size.

LS-POST and LS-PREPOST can also read the crack file and display cracks.

***MAT_WINFRITH_CONCRETE_REINFORCEMENT**

This is Material Types 84 rebar reinforcement. Reinforcement may be defined in specific groups of elements, but it is usually more convenient to define a two-dimensional mat in a specified layer of a specified material. Reinforcement quantity is defined as the ratio of the cross-sectional area of steel relative to the cross-sectional area of concrete in the element (or layer). These cards may follow either one of two formats below and may also be defined in any order.

Option 1 (Reinforcement quantities in element groups).

Card 1 2 3 4 5 6 7 8

Variable	EID1	EID2	INC	XR	YR	ZR		
Type	I	I	I	F	F	F		

Option 2 (Two dimensional layers by part ID).

Card 1 2 3 4 5 6 7 8

Variable		PID	AXIS	COOR	RQA	RQB		
Type	blank	I	I	F	F	F		

<u>VARIABLE</u>	<u>DESCRIPTION</u>
EID1	First element ID in group.
EID2	Last element ID in group
INC	Element increment for generation.
XR	X-reinforcement quantity (for bars running parallel to global x-axis).
YR	Y-reinforcement quantity (for bars running parallel to global y-axis).
ZR	Z-reinforcement quantity (for bars running parallel to global z-axis).

<u>VARIABLE</u>	<u>DESCRIPTION</u>
PID	Part ID of reinforced elements.
AXIS	Axis normal to layer. EQ.1: A and B are parallel to global Y and Z, respectively. EQ.2: A and B are parallel to global Z and X, respectively. EQ.3: A and B are parallel to global X and Y, respectively.
COOR	Coordinate location of layer (X-coordinate if AXIS.EQ.1; Y-coordinate if AXIS.EQ.2; Z-coordinate if AXIS.EQ.3).
RQA	Reinforcement quantity (A).
RQB	Reinforcement quantity (B).

Remarks:

1. Reinforcement quantity is the ratio of area of reinforcement in an element to the element's total cross-sectional area in a given direction. This definition is true for both Options 1 and 2. Where the options differ is in the manner in which it is decided which elements are reinforced. In Option 1, the reinforced element IDs are spelled out. In Option 2, elements of part ID PID which are cut by a plane (layer) defined by AXIS and COOR are reinforced.

***MAT_ORTHOTROPIC_VISCOELASTIC**

This is Material Type 86. It allows the definition of an orthotropic material with a viscoelastic part. This model applies to shell elements.

Card 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				A1	A2	A3		
Type				F	F	F		

Card 5

Variable	V1	V2	V3	D1	D2	D3		
Type	F	F	F	F	F	F		

VARIABLE**DESCRIPTION**

MID	Material identification. A unique number 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_∞ , long-time shear modulus
BETA	β , decay constant
PRBA	Poisson's ratio, ν_{ba}
PRCA	Poisson's ratio, ν_{ca}
PRCB	Poisson's ratio, ν_{cb}
GAB	Shear modulus, G_{ab}
GBC	Shear modulus, G_{bc}
GCA	Shear modulus, G_{ca}

<u>VARIABLE</u>	<u>DESCRIPTION</u>
AOPT	Material axes option (see MAT_OPTION TROPIC_ELASTIC for a more complete description): EQ. 0.0: locally orthotropic with material axes determined by element nodes 1, 2, and 4, as with *DEFINE_COORDINATE_NODES. EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR. EQ. 3.0: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle, MANGLE, from a line in the plane of the element defined by the cross product of the vector \mathbf{v} with the element normal.
MANGLE	Material angle in degrees for AOPT = 3, may be overridden on the element card, see *ELEMENT_SHELL_BETA.
A1 A2 A3	Define components of vector a for AOPT = 2.
V1 V2 V3	Define components of vector v for AOPT = 3.
D1 D2 D3	Define components of vector d for AOPT = 2.

Remarks:

For the orthotropic definition it is referred to Material Type 2 and 21.

***MAT_CELLULAR_RUBBER**

This is Material Type 87. This material model provides a cellular rubber model with confined air pressure combined with linear viscoelasticity as outlined by Christensen [1980]. See Figure 20.32.

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 ΔL in the gauge length.

Define, if $N = 0$:

C10 Coefficient, C_{10}

C01 Coefficient, C_{01}

C11 Coefficient, C_{11}

C20 Coefficient, C_{20}

C02 Coefficient, C_{02}

VARIABLE	DESCRIPTION
P0	Initial air pressure, P_0
PHI	Ratio of cellular rubber to rubber density, Φ
IVS	Initial volumetric strain, γ_0
G	Optional shear relaxation modulus, G , for rate effects (viscosity)
BETA	Optional decay constant, β_1

Remarks:

Rubber is generally considered to be fully incompressible since the bulk modulus greatly exceeds the shear modulus in magnitude. To model the rubber as an unconstrained material a hydrostatic work term, $W_H(J)$, is included in the strain energy functional which is function of the relative volume, J , [Ogden, 1984]:

$$W(J_1, J_2, J) = \sum_{p,q=0}^n C_{pq} (J_1 - 3)^p (J_2 - 3)^q + W_H(J)$$

$$J_1 = I_1 I_3^{-1/3}$$

$$J_2 = I_2 I_3^{-2/3}$$

In order to prevent volumetric work from contributing to the hydrostatic work the first and second invariants are modified as shown. This procedure is described in more detail by Sussman and Bathe [1987].

The effects of confined air pressure in its overall response characteristics is included by augmenting the stress state within the element by the air pressure.

$$\sigma_{ij} = \sigma_{ij}^{sk} - \delta_{ij} \sigma^{air}$$

where σ_{ij}^{sk} is the bulk skeletal stress and σ^{air} is the air pressure computed from the equation:

$$\sigma^{air} = -\frac{P_0 \gamma}{1 + \gamma - \phi}$$

where p_0 is the initial foam pressure usually taken as the atmospheric pressure and γ defines the volumetric strain

$$\gamma = V - 1 + \gamma_0$$

where V is the relative volume of the voids and γ_0 is the initial volumetric strain which is typically zero. The rubber skeletal material is assumed to be incompressible.

Rate effects are taken into account through linear viscoelasticity by a convolution integral of the form:

$$\sigma_{ij} = \int_0^t g_{ijkl}(t - \tau) \frac{\partial \epsilon_{kl}}{\partial \tau} d\tau$$

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, β_1 .

The Mooney-Rivlin rubber model (model 27) is obtained by specifying $n=1$ without air pressure and viscosity. In spite of the differences in formulations with Model 27, we find that the results obtained with this model are nearly identical with those of material type 27 as long as large values of Poisson's ratio are used.

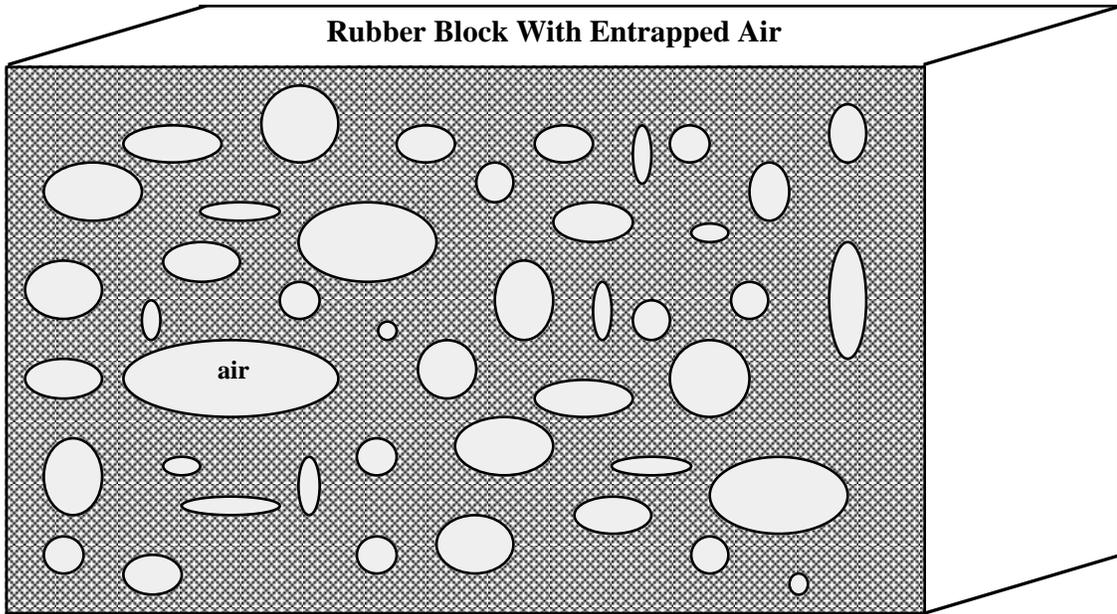


Figure 20.32. Cellular rubber with entrapped air. By setting the initial air pressure to zero, an open cell, cellular rubber can be simulated.

***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 valure for the mechanical threshold stress, the flow stress is determined using either a thermal-activation-controlled or a drag-controlled kinetics relationship.. An equation-of-state is required for solid elements and a bulk modulus must be defined below for shell elements.

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	GOI	PINVI	QINVI	EDOTS	GOS	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						

VARIABLE

DESCRIPTION

- 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 $\hat{\sigma}_{\text{eso}}$, saturation threshold stress at 0° K (force/area).
- BULK Bulk modulus defined for shell elements only. Do not input for solid elements.

<u>VARIABLE</u>	<u>DESCRIPTION</u>
EDOTS0	$\dot{\epsilon}_{\epsilon s o}$, reference strain-rate (time ⁻¹).
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 ⁻¹).
G0	g_0 , normalized activation energy for a .dislocation/dislocation interaction.
PINV	$\frac{1}{p}$, material constant.
QINV	$\frac{1}{q}$, material constant.
EDOTI	$\dot{\epsilon}_{o,i}$, reference strain-rate (time ⁻¹).
G0I	$g_{0,i}$, normalized activation energy for a dislocation/interstitial interaction.
PINVI	$\frac{1}{p_i}$, material constant.
QINVI	$\frac{1}{q_i}$, material constant.
EDOTS	$\dot{\epsilon}_{o,s}$, reference strain-rate (time ⁻¹).
G0S	$g_{0,s}$ normalized activation energy for a dislocation/solute interaction.
PINVS	$\frac{1}{p_s}$, material constant.
QINVS	$\frac{1}{q_s}$, material constant.

VARIABLE	DESCRIPTION
RHOCP	ρc_p , product of density and specific heat.
TEMPRF	T_{ref} , initial element temperature in degrees K.
ALPHA	α , material constant (typical value is between 0 and 2).
EPS0	ϵ_o , factor to normalize strain rate in the calculation of Θ_o . (Use 1., 10^{-3} , or 10^{-6} for time units of seconds, milliseconds, or microseconds, respectively.)

Remarks:

The flow stress σ is given by:

$$\sigma = \hat{\sigma}_a + \frac{G}{G_0} \left[s_{th} \hat{\sigma} + s_{th,i} \hat{\sigma}_i + s_{th,s} \hat{\sigma}_s \right]$$

The first product in the equation for σ contains a micro-structure evolution variable, i.e., $\hat{\sigma}$, called the *Mechanical Threshold Stress* (MTS), that is multiplied by a constant-structure deformation variable s_{th} : s_{th} is a function of absolute temperature T and the plastic strain-rates $\dot{\epsilon}^p$. The evolution equation for $\hat{\sigma}$ is a differential hardening law representing dislocation-dislocation interactions:

$$\frac{\partial \hat{\sigma}}{\partial \epsilon^p} \equiv \Theta_o \left[1 - \frac{\tanh\left(\alpha \frac{\hat{\sigma}}{\hat{\sigma}_{\epsilon s}}\right)}{\tanh(\alpha)} \right]$$

The term, $\frac{\partial \hat{\sigma}}{\partial \epsilon^p}$, represents the hardening due to dislocation generation and the stress ratio, $\frac{\hat{\sigma}}{\hat{\sigma}_{\epsilon s}}$, represents softening due to dislocation recovery. The threshold stress at zero strain-hardening $\hat{\sigma}_{\epsilon s}$ is called the saturation threshold stress. Relationships for Θ_o , $\hat{\sigma}_{\epsilon s}$ 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, $\hat{\sigma}_{\epsilon s}$, is given as:

$$\hat{\sigma}_{\epsilon s} = \hat{\sigma}_{\epsilon s0} \left(\frac{\dot{\epsilon}^p}{\dot{\epsilon}_{\epsilon s0}} \right)^{kT/Gb^3A}$$

which contains the input constants: $\hat{\sigma}_{\dot{\epsilon}_{so}}$, $\dot{\epsilon}_{\dot{\epsilon}_{so}}$, 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, (Cowper Symonds).
P	Strain rate parameter, P, (Cowper Symonds).
LCSS	Load curve ID defining effective stress versus total effective strain.
LCSR	Load curve ID defining strain rate scaling effect on yield stress.
EFTX	Failure flag. EQ.0.0: failure determined by maximum tensile strain (default), EQ.1.0: failure determined only by tensile strain in local x direction, EQ.2.0: failure determined only by tensile strain in local y direction.
DAMP	Stiffness-proportional damping ratio. Typical values are 1e-3 or 1e-4. If set too high instabilities can result.
RATEFAC	Filtering factor for strain rate effects. Must be between 0 (no filtering) and 1 (infinite filtering) The filter is a simple low pass filter to remove high frequency oscillation from the strain rates before they are used in rate effect calculations. The cut off frequency of the filter is $[(1 - \text{RATEFAC}) / \text{timestep}]$ 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

<u>VARIABLE</u>	<u>DESCRIPTION</u>
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_SOFT_TISSUE_{OPTION}

Options include:

<BLANK>

VISCO

This is Material Type 91 (OPTION=<BLANK>) or Material Type 92 (OPTION=VISCO). This material is a transversely isotropic hyperelastic model for representing biological soft tissues such as ligaments, tendons, and fascia. The representation provides an isotropic Mooney-Rivlin matrix reinforced by fibers having a strain energy contribution with the qualitative material behavior of collagen. The model has a viscoelasticity option which activates a six-term Prony series kernel for the relaxation function. In this case, the hyperelastic strain energy represents the elastic (long-time) response. See Weiss et al. [1996] and Puso and Weiss [1998] for additional details. The material is available for use with brick and shell elements. When used with shell elements, the Belytschko-Tsay formulation (#2) must be selected.

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	MID	RO	C1	C2	C3	C4	C5	
Type	I	F	F	F	F	F	F	

Card 2 1 2 3 4 5 6 7 8

Variable	XK	XLAM	FANG	XLAM0				
Type	F	F	F	F				

Card Format (continued)

Card 3 1 2 3 4 5 6 7 8

Variable	AOPT	AX	AY	AZ	BX	BY	BZ	
Type	F	F	F	F	F	F	F	

Card 4 1 2 3 4 5 6 7 8

Variable	LA1	LA2	LA3					
Type	F	F	F					

Define the following two cards only for the **VISCO** option:

Card 5 1 2 3 4 5 6 7 8

Variable	S1	S2	S3	S4	S5	S6		
Type	F	F	F	F	F	F		

Card 6 1 2 3 4 5 6 7 8

Variable	T1	T2	T3	T4	T5	T6		
Type	F	F	F	F	F	F		

VARIABLE

DESCRIPTION

MID Material identification. A unique number has to be chosen.

RO Mass density

C1 - C5 Hyperelastic coefficients (see equations below)

<u>VARIABLE</u>	<u>DESCRIPTION</u>
XK	Bulk Modulus
XLAM	Stretch ratio at which fibers are straightened
FANG	Fiber angle in local shell coordinate system (shells only)
XLAM0	Initial fiber stretch (optional)
AOPT	Material axes option, see Figure 20.1 (bricks only): 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.
AX, AY, AZ	First material axis point or vector (bricks only)
BX, BY, BZ	Second material axis point or vector (bricks only)
LAX, LAY, LAZ	Local fiber orientation vector (bricks only)
S1 - S6	Spectral strengths for Prony series relaxation kernel (<i>OPTION=VISCO</i>)
T1 - T6	Characteristic times for Prony series relaxation kernel (<i>OPTION=VISCO</i>)

Remarks on Formulation:

The overall strain energy W is "uncoupled" and includes two isotropic deviatoric matrix terms, a fiber term F , and a bulk term:

$$W = C_1 \left(\tilde{I}_1 - 3 \right) + C_2 \left(\tilde{I}_2 - 3 \right) + F(\lambda) + \frac{1}{2} K [\ln(J)]^2$$

Here, \tilde{I}_1 and \tilde{I}_2 are the deviatoric invariants of the right Cauchy deformation tensor, λ is the deviatoric part of the stretch along the current fiber direction, and $J = \det \mathbf{F}$ is the volume ratio. The material coefficients C_1 and C_2 are the Mooney-Rivlin coefficients, while K is the effective bulk modulus of the material (input parameter XK).

The derivatives of the fiber term F are defined to capture the behavior of crimped collagen. The fibers are assumed to be unable to resist compressive loading - thus the model is isotropic when $\lambda < 1$. An exponential function describes the straightening of the fibers, while a linear function describes the behavior of the fibers once they are straightened past a critical fiber stretch level $\lambda \geq \lambda^*$ (input parameter XLAM):

$$\frac{\partial F}{\partial \lambda} = \left\{ \begin{array}{ll} 0 & \lambda < 1 \\ \frac{C_3}{\lambda} [\exp(C_4(\lambda - 1)) - 1] & \lambda < \lambda^* \\ \frac{1}{\lambda} (C_5 \lambda + C_6) & \lambda \geq \lambda^* \end{array} \right\}$$

Coefficients C_3 , C_4 , and C_5 must be defined by the user. C_6 is determined by LS-DYNA to ensure stress continuity at $\lambda = \lambda^*$. Sample values for the material coefficients $C_1 - C_5$ and λ^* for ligament tissue can be found in Quapp and Weiss [1998]. The bulk modulus K should be at least 3 orders of magnitude larger than C_1 to ensure near-incompressible material behavior.

Viscoelasticity is included via a convolution integral representation for the time-dependent second Piola-Kirchhoff stress $\mathbf{S}(\mathbf{C}, t)$:

$$\mathbf{S}(\mathbf{C}, t) = \mathbf{S}^e(\mathbf{C}) + \int_0^t 2G(t-s) \frac{\partial W}{\partial \mathbf{C}(s)} ds$$

Here, \mathbf{S}^e is the elastic part of the second PK stress as derived from the strain energy, and $G(t-s)$ is the reduced relaxation function, represented by a Prony series:

$$G(t) = \sum_{i=1}^6 S_i \exp\left(-\frac{t}{T_i}\right)$$

Puso and Weiss [1998] describe a graphical method to fit the Prony series coefficients to relaxation data that approximates the behavior of the continuous relaxation function proposed by Y-C. Fung as quasilinear viscoelasticity.

Remarks on Input Parameters:

Cards 1 through 4 must be included for both shell and brick elements, although for shells cards 3 and 4 are ignored and may be blank lines.

For shell elements, the fiber direction lies in the plane of the element. The local axis is defined by a vector between nodes n1 and n2, and the fiber direction may be offset from this axis by an angle FANG.

For brick elements, the local coordinate system is defined using the convention described previously for *MAT_ORTHOTROPIC_ELASTIC. The fiber direction is oriented in the local system using input parameters LAX, LAY, and LAZ. By default, (LAX,LAY,LAZ) = (1,0,0) and the fiber is aligned with the local x-direction.

An optional initial fiber stretch can be specified using XLAM0. The initial stretch is applied during the first time step. This creates preload in the model as soft tissue contracts and equilibrium is established. For example, a ligament tissue "uncrimping strain" of 3% can be represented with initial stretch value of 1.03.

If the **VISCO** option is selected, at least one Prony series term (S1,T1) must be defined.

*MAT_ELASTIC_6DOF_SPRING_DISCRETE_BEAM

This is Material Type 93. This material model is defined for simulating the effects of nonlinear elastic and nonlinear viscous beams by using six springs each acting about one of the six local degrees-of-freedom. The input consists of part ID's that reference material type, *MAT_ELASTIC_SPRING_DISCRETE_BEAM above (type 74 above). Generally, these referenced parts are used only for the definition of this material model and are not referenced by any elements. The two nodes defining a beam may be coincident to give a zero length beam, or offset to give a finite length beam. For finite length discrete beams the absolute value of the variable SCOOR in the SECTION_BEAM input should be set to a value of 2.0, which causes the local r-axis to be aligned along the two nodes of the beam to give physically correct behavior. The distance between the nodes of a beam should not affect the behavior of this material model. A triad is used to orient the beam for the directional springs.

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	MID	RO	TPIDR	TPIDS	TPIDT	RPIDR	RPIDS	RPIDT
Type	I	F	I	I	I	I	I	I

<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 definition.
TPIDR	Translational motion in the local r-direction is governed by part ID TPIDR. If zero, no force is computed in this direction.
TPIDS	Translational motion in the local s-direction is governed by part ID TPIDS. If zero, no force is computed in this direction.
TPIDT	Translational motion in the local t-direction is governed by part ID TPIDT. If zero, no force is computed in this direction.
RPIDR	Rotational motion about the local r-axis is governed by part ID RPIDR. If zero, no moment is computed about this axis.
RPIDS	Rotational motion about the local s-axis is governed by part ID RPIDS. If zero, no moment is computed about this axis.
RPIDT	Rotational motion about the local t-axis is governed by part ID RPIDT. If zero, no moment is computed about this axis.

***MAT_INELASTIC_SPRING_DISCRETE_BEAM**

This is Material Type 94. This model permits elastoplastic springs with damping to be represented with a discrete beam element type 6. A yield force versus deflection curve is used which can vary in tension and compression..

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	MID	RO	K	F0	D	CDF	TDF	
Type	I	F	F	F	F	F	F	

Card 2 1 2 3 4 5 6 7 8

Variable	FLCID	HLCID	C1	C2	DLE	GLCID		
Type	F	F	F	F	F	I		

<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 definition.
K	Elastic loading/unloading stiffness. This is required input.
F0	Optional initial force. This option is inactive if this material is referenced in a part referenced by material type *MAT_INELASTIC_6DOF_SPRING
D	Optional viscous damping coefficient.
CDF	Compressive displacement at failure. Input as a positive number. After failure, no forces are carried. This option does not apply to zero length springs. EQ.0.0: inactive.
TDF	Tensile displacement at failure. After failure, no forces are carried. EQ.0.0: inactive.

VARIABLE	DESCRIPTION
FLCID	Load curve ID, see *DEFINE_CURVE, defining the yield force versus plastic deflection. If the origin of the curve is at (0,0) the force magnitude is identical in tension and compression, i.e., only the sign changes. If not, the yield stress in the compression is used when the spring force is negative. The plastic displacement increases monotonically in this implementation. The load curve is required input.
HLCID	Load curve ID, see *DEFINE_CURVE, defining force versus relative velocity (Optional). If the origin of the curve is at (0,0) the force magnitude is identical for a given magnitude of the relative velocity, i.e., only the sign changes.
C1	Damping coefficient.
C2	Damping coefficient
DLE	Factor to scale time units.
GLCID	Optional load curve ID, see *DEFINE_CURVE, defining a scale factor versus deflection for load curve ID, HLCID. If zero, a scale factor of unity is assumed.

Remarks:

The yield force is taken from the load curve:

$$F^Y = F_y(\Delta L^{plastic})$$

where $L^{plastic}$ is the plastic deflection. A trial force is computed as:

$$F^T = F^n + K\Delta\dot{L}(\Delta t)$$

and is checked against the yield force to determine F :

$$F = \begin{cases} F^Y & \text{if } F^T > F^Y \\ F^T & \text{if } F^T \leq F^Y \end{cases}$$

The final force, which includes rate effects and damping, is given by:

$$F^{n+1} = F \cdot \left[1 + C1 \cdot \Delta\dot{L} + C2 \cdot \text{sgn}(\Delta\dot{L}) \ln \left(\max \left\{ 1, \frac{|\Delta\dot{L}|}{DLE} \right\} \right) \right] + D\Delta\dot{L} + g(\Delta L)h(\Delta\dot{L})$$

Unless the origin of the curve starts at (0,0), the negative part of the curve is used when the spring force is negative where the negative of the plastic displacement is used to interpolate, F_y . The

positive part of the curve is used whenever the force is positive. In these equations, ΔL is the change in length

$$\Delta L = \text{current length} - \text{initial length}$$

The cross sectional area is defined on the section card for the discrete beam elements, See *SECTION_BEAM. The square root of this area is used as the contact thickness offset if these elements are included in the contact treatment.

*MAT_INELASTIC_6DOF_SPRING_DISCRETE_BEAM

This is Material Type 95. This material model is defined for simulating the effects of nonlinear inelastic and nonlinear viscous beams by using six springs each acting about one of the six local degrees-of-freedom. The input consists of part ID's that reference material type, *MAT_INELASTIC_SPRING_DISCRETE_BEAM above (type 94). Generally, these referenced parts are used only for the definition of this material model and are not referenced by any elements. The two nodes defining a beam may be coincident to give a zero length beam, or offset to give a finite length beam. For finite length discrete beams the absolute value of the variable SCOOR in the SECTION_BEAM input should be set to a value of 2.0, which causes the local r-axis to be aligned along the two nodes of the beam to give physically correct behavior. The distance between the nodes of a beam should not affect the behavior of this material model. A triad must be used to orient the beam for zero length beams.

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	MID	RO	TPIDR	TPIDS	TPIDT	RPIDR	RPIDS	RPIDT
Type	I	F	I	I	I	I	I	I

<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 definition.
TPIDR	Translational motion in the local r-direction is governed by part ID TPIDR. If zero, no force is computed in this direction.
TPIDS	Translational motion in the local s-direction is governed by part ID TPIDS. If zero, no force is computed in this direction.
TPIDT	Translational motion in the local t-direction is governed by part ID TPIDT. If zero, no force is computed in this direction.
RPIDR	Rotational motion about the local r-axis is governed by part ID RPIDR. If zero, no moment is computed about this axis.
RPIDS	Rotational motion about the local s-axis is governed by part ID RPIDS. If zero, no moment is computed about this axis.
RPIDT	Rotational motion about the local t-axis is governed by part ID RPIDT. If zero, no moment is computed about this axis.

***MAT_BRITTLE_DAMAGE**

This is Material Type 96.

Card 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.

VARIABLE	DESCRIPTION
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. ν 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, ε is the small constant, H is the softening modulus, and α is an internal variable. H is set automatically by the program; see g_c below. α measures the crack field intensity and is output in the equivalent plastic strain field, $\bar{\varepsilon}^p$, in a normalized fashion.

The evolution of alpha is governed by a maximum dissipation argument. When the normalized value reaches unity it means that the material's strength has been reduced to 2% of its original value in the normal and parallel directions to the smeared crack. Note that for plotting purposes it is never output greater than 5.

4. f_s is the initial shear traction that may be transmitted across a smeared crack plane. The shear traction is limited to be less than or equal to $f_s(1 - \beta)(1 - \exp[-H\alpha])$, through the use of two orthogonal shear damage surfaces. Note that the shear degradation is coupled to the tensile degradation through the internal variable alpha which measures the intensity of the crack field. β is the shear retention factor defined below. The shear degradation is taken care of by reducing the material's shear stiffness parallel to the smeared crack plane.
5. g_c is the fracture toughness of the material. It should be entered as fracture energy per unit area crack advance. Once entered the softening modulus is automatically calculated based on element and crack geometries.
6. β is the shear retention factor. As the damage progresses the shear tractions allowed across the smeared crack plane asymptote to the product βf_s .
7. η represents the viscosity of the material. Viscous behavior is implemented as a simple Perzyna regularization method. This allows for the inclusion of first order rate effects. The use of some viscosity is recommended as it serves as regularizing parameter that increases the stability of calculations.
8. σ_y is a uniaxial compressive yield stress. A check on compressive stresses is made using the J2 yield function $\mathbf{s}:\mathbf{s} - \sqrt{\frac{2}{3}}\sigma_y \leq 0$, 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 η between 104 to 106 psi/sec are recommended. Our limited experience thus far has shown that many problems require nonzero values of η to run to avoid error terminations.

Remark: Various other internal variables such as crack orientations and degraded stiffness tensors are internally calculated but currently not available for output.

*MAT_GENERAL_JOINT_DISCRETE_BEAM

This is Material Type 97. This model is used to define a general joint constraining any combination of degrees of freedom between two nodes. The nodes may belong to rigid or deformable bodies. In most applications the end nodes of the beam are coincident and the local coordinate system (r,s,t axes) is defined by CID (see *SECTION_BEAM).

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	MID	RO	TR	TS	TT	RR	RS	RT
Type	I	F	I	I	I	I	I	
Remarks	1							

Card 2 1 2 3 4 5 6 7 8

Variable	RPST	RPSR						
Type	F	F						
Remarks	2							

<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 definition.
TR	Translational constraint code along the r-axis (0 => free, 1 => constrained)
TS	Translational constraint code along the s-axis (0 => free, 1 => constrained)
TT	Translational constraint code along the t-axis (0 => free, 1 => constrained)
RR	Rotational constraint code about the r-axis (0 => free, 1 => constrained)
RS	Rotational constraint code about the s-axis (0 => free, 1 => constrained)
RT	Rotational constraint code about the t-axis (0 => free, 1 => constrained)
RPST	Penalty stiffness scale factor for translational constraints.

<u>VARIABLE</u>	<u>DESCRIPTION</u>
RPSR	Penalty stiffness scale factor for rotational constraints.

Remarks:

1. For explicit calculations, the additional stiffness due to this joint may require addition mass and inertia for stability. Mass and rotary inertia for this beam element is based on the defined mass density , the volume, and the mass moment of inertia defined in the *SECTION_BEAM input.
2. The penalty stiffness applies to explicit calculations. For implicit calculations, constraint equations are generated and imposed on the system equations; therefore, these constants, RPST and RPSR, are not used.

***MAT_SIMPLIFIED_JOHNSON_COOK**

This is Material Type 98. The Johnson/Cook strain sensitive plasticity is used for problems where the strain rates vary over a large range. In this simplified model, thermal effects and damage are ignored, and the maximum stress is directly limited since thermal softening which is very significant in reducing the yield stress under adiabatic loading is not available. An iterative plane stress update is used for the shell elements, but due to the simplifications related to thermal softening and damage, this model is 50% faster than the full Johnson/Cook implementation. To compensate for the lack of thermal softening, limiting stress values are used to keep the stresses within reasonable limits. A resultant formulation for the Belytschko-Tsay, the C0 Triangle, and the fully integrated type 16 shell elements is activated by specifying either zero or one through thickness integration point on the *SHELL_SECTION card. This latter option is less accurate than through thickness integration but is somewhat faster. Since the stresses are not computed in the resultant formulation, the stress output to the databases for the resultant elements are zero. This model is also available for the Hughes-Liu beam, the Belytschko-Schwer beam, and the truss element. For the resultant beam formulation, the rate effects are approximated by the axial rate since the thickness of the beam about it bending axes is unknown. The linear bulk modulus is used to determine the pressure in the elements, since the use of this model is primarily for structural analysis.

Card 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	A	B	N	C	PSFAIL	SIGMAX	SIGSAT	EPSO
Type	F	F	F	F	F	F	F	F
Default	none	0.0	0.0	0.0	1.0E+17	SIGSAT	1.0E+28	1.0

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MID	Material identification. A unique number has to be chosen.
RO	Mass density
E	Young's modulus

VARIABLE	DESCRIPTION
PR	Poisson's ratio
VP	Formulation for rate effects: EQ.0.0: Scale yield stress (default), EQ.1.0: Viscoplastic formulation. This option applies only to the 4-node shell and 8-node thick shell if and only if through thickness integration is used.
A	See equations below.
B	See equations below.
N	See equations below.
C	See equations below.
PSFAIL	Effective plastic strain at failure. If zero failure is not considered.
SIGMAX	Maximum stress obtainable from work hardening before rate effects are added (optional). This option is ignored if VP=1.0. This option is ignored if VP=1.0.
SIGSAT	Saturation stress which limits the maximum value of effective stress which can develop after rate effects are added (optional).
EPSO	Effective plastic strain rate. This value depends on the time units. Typically, input 1 for units of seconds, 0.001 for units of milliseconds, 0.000001 for microseconds, etc.

Remarks:

Johnson and Cook express the flow stress as

$$\sigma_y = \left(A + B \bar{\epsilon}^{p^n} \right) \left(1 + c \ln \dot{\epsilon}^* \right)$$

where

A, B, C and n are input constants

$\bar{\epsilon}^p$ effective plastic strain

$\dot{\epsilon}^* = \frac{\dot{\bar{\epsilon}}}{\epsilon_0}$ effective strain rate for $\epsilon_0 = 1 s^{-1}$

The maximum stress is limited by *sigmax* and *sigsat* by:

$$\sigma_y = \min \left\{ \min \left[A + B \bar{\epsilon}^{p^n}, sigmax \right] \left(1 + c \ln \dot{\epsilon}^* \right), sigsat \right\}$$

Failure occurs when the effective plastic strain exceeds *psfail*.

If the viscoplastic option is active, VP=1.0, the parameters SIGMAX and SIGSAT are ignored since these parameters make convergence of the viscoplastic strain iteration loop difficult to achieve. The viscoplastic option replaces the plastic strain in the forgoing equations by the viscoplastic strain and the strainrate by the viscoplastic strain rate. Numerical noise is substantially reduced by the viscoplastic formulation.

***MAT_SIMPLIFIED_JOHNSON_COOK_ORTHOTROPIC_DAMAGE**

This is Material Type 99. This model, which is implemented only for shell elements with multiple through thickness integration points, is an extension of model 98 to include orthotropic damage as a means of treating failure in aluminum panels. Directional damage begins after a defined failure strain is reached in tension and continues to evolve until a tensile rupture strain is reached in either one of the two orthogonal directions. After rupture is detected at all integration points, the element is deleted.

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	MID	RO	E	PR	VP	EPPFR	LCDM	NUMINT
Type	I	F	F	F	F	F	I	I
Default	none	none	none	none	0.0	1.e+16	optional	all intg. pt

Card 2

Variable	A	B	N	C	PSFAIL	SIGMAX	SIGSAT	EPSO
Type	F	F	F	F	F	F	F	F
Default	none	0.0	0.0	0.0	1.0E+17	SIGSAT	1.0E+28	1.0

<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
VP	Formulation for rate effects: EQ.0.0: Scale yield stress (default), EQ.1.0: Viscoplastic formulation. This option applies only to the 4-node shell and 8-node thick shell if and only if through thickness integration is used.

<u>VARIABLE</u>	<u>DESCRIPTION</u>
EPPFR	Plastic strain at which material ruptures (logrithmic).
LCDM	Load curve ID defining nonlinear damage curve. See Figure 20.30.
NUMINT	Number of through thickness integration points which must fail before the element is deleted. (If zero, all points must fail.) The default of all integration points is not recommended since elements undergoing large strain are often not deleted due to nodal fiber rotations which limit strains at active integration points after most points have failed. Better results are obtained if NUMINT is set to 1 or a number less than one half of the number of through thickness points. For example, if four through thickness points are used, NUMINT should not exceed 2, even for fully integrated shells which have 16 integration points.
A	See equations below.
B	See equations below.
N	See equations below.
C	See equations below.
PSFAIL	Effective plalstic strain at failure. If zero failure is not considerd.
SIGMAX	Maximum stress obtainable from work hardening before rate effects are added (optional). This option is ignored if VP=1.0. This option is ignored if VP=1.0.
SIGSAT	Saturation stress which limits the maximum value of effective stress which can develop after rate effects are added (optional).
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.

Remarks:

See the description for the SIMPLIFIED_JOHNSON_COOK model above.

***MAT_SPOTWELD_{OPTION}**

This is Material Type 100. The material model applies to beam element type 9 and to solid element type 1. The failure models apply to both beam and solid elements.

The beam elements, based on the Hughes-Liu beam formulation, may be placed between any two deformable shell surfaces and tied with constraint contact, *CONTACT_SPOTWELD, which eliminates the need to have adjacent nodes at 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.

In flat topologies the shell elements have an unconstrained drilling degree-of-freedom which prevents torsional forces from being transmitted. If the torsional forces are deemed to be important, brick elements should be used to model the spot welds..

Beam and solid element force resultants for MAT_SPOTWELD are written to the spot weld force file, SWFORC, and the file for element stresses and resultants for designated elements, ELOUT.

It is advisable to include all spotwelds, which provide the slave nodes, and spot welded materials, which define the master segments, within a single *CONTACT_SPOTWELD interface. As a constraint method these interfaces are treated independently which can lead to significant problems if such interfaces share common nodal points. An added benefit is that memory usage can be substantially less with a single interface.

Options include:

<BLANK>

DAMAGE-FAILURE

The DAMAGE-FAILURE option causes one additional line to be read with the damage parameter and a flag that determines how failure is computed from the resultants. On this line the parameter, DMG, if nonzero, invokes damage mechanics combined with the plasticity model to achieve a smooth drop off of the resultant forces prior to the removal of the spotweld. The parameter FOPT determines the method used in computing resultant based failure, which is unrelated to damage.

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

Define this card as Card 2 when the DAMAGE-FAILURE option is inactive. Card 3 is not defined

Card 2 1 2 3 4 5 6 7 8

Variable	EFAIL	NRR	NRS	NRT	MRR	MSS	MTT	NF
Type	F	F	F	F	F	F	F	F

Define cards 2 and 3 below if the DAMAGE-FAILURE option is active.
OPT=0.0, Resultant based failure.

Card 2 1 2 3 4 5 6 7 8

Variable	EFAIL	NRR	NRS	NRT	MRR	MSS	MTT	NF
Type	F	F	F	F	F	F	F	F

OPT=1.0, Stress based failure.

Card 2 1 2 3 4 5 6 7 8

Variable	EFAIL	SIGAX	SIGTAU					NF
Type	F	F	F					F

OPT=1.0, Stress based failure if strain rate effects are included

Card 2 1 2 3 4 5 6 7 8

Variable	EFAIL	-LCAX	-LCTAU					NF
Type	F	F	F					F

OPT=2.0, User subroutine for failure

Card 2 1 2 3 4 5 6 7 8

Variable	EFAIL	USERV1	USERV2	USERV3	USERV4	USERV5	USERV6	NF
Type	F	F	F	F	F	F	F	F

OPT=3.0, 4.0, and 5.0

Card 2 1 2 3 4 5 6 7 8

Variable	EFAIL	ZD	ZT	ZALP1	ZALP2	ZALP3	ZRRAD	NF
Type	F	F	F	F	F	F	F	F

Card 3 is defined only for the DAMAGE-FAILURE option.

Card 3 1 2 3 4 5 6 7 8

Variable	RS	OPT	SIGKF					
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
PR	Poisson's ratio
SIGY	Initial yield stress
ET	Hardening modulus, E_t
DT	Time step size for mass scaling, Δt
TFAIL	Failure time if nonzero. If zero this option is ignored.

VARIABLE	DESCRIPTION
EFAIL	Effective plastic strain at failure. If the damage option is inactive, the spot weld element is deleted when the plastic strain at each integration point exceeds EFAIL. If the damage option is active, the plastic strain must exceed the rupture strain at each integration point before deletion occurs.
NRR	Axial force resultant N_{rr_F} or maximum axial stress σ_{rr}^F at failure depending on the value of OPT (see below). If zero, failure due to this component is not considered. If negative, NRR is the load curve ID defining the maximum axial stress at failure as a function of the effective strain rate.
NRS	Force resultant N_{rs_F} or maximum shear stress τ^F at failure depending on the value of OPT (see below). If zero, failure due to this component is not considered. If negative, NRS is the load curve ID defining the maximum shear stress at failure as a function of the effective strain rate.
NRT	Force resultant N_{rt_F} at failure. If zero, failure due to this component is not considered.
MRR	Torsional moment resultant M_{rr_F} at failure. If zero, failure due to this component is not considered.
MSS	Moment resultant M_{ss_F} at failure. If zero, failure due to this component is not considered.
MTT	Moment resultant M_{tt_F} at failure. If zero, failure due to this component is not considered.
NF	Number of force vectors stored for filtering. The default value is set to zero which is generally recommended unless oscillatory resultant forces are observed in the time history databases. Even though these welds should not oscillate significantly, this option was added for consistency with the other spot weld options. NF affects the storage since it is necessary to store the resultant forces as history variables. When NF is nonzero, the resultants in the output databases are filtered.
SIGAX	Maximum axial stress σ_{rr}^F at failure. If zero, failure due to this component is not considered.
SIGTAU	Maximum shear stress τ^F at failure. If zero, failure due to this component is not considered.
LCAX	Load curve ID defining the maximum axial stress at failure as a function of the effective strain rate. Input as a negative number.
LCTAU	Load curve ID defining the maximum shear stress at failure as a function of the effective strain rate. Input as a negative number.

VARIABLE	DESCRIPTION
USERVn	Failure constants for user failure subroutine, n=1,2,...6.
ZD	Notch diameter
ZT	Sheet thickness.
ZALP1	Correction factor alpha1
ZALP2	Correction factor alpha2
ZALP3	Correction factor alpha3
ZRHO	Notch root radius (OPT=3.0 only).
RS	Rupture strain. Define if and only if damage is active.
OPT	Failure option: EQ.0: resultant based failure EQ.1: stress based failure computed from resultants (Toyota) EQ.2: user subroutine to determine failure EQ.3: notch stress based failure (beam weld only) EQ.4: stress intensity factor at failure (beam weld only) EQ.5: structural stress at failure (beam weld only).
OPT	Failure parameter. If OPT: EQ.0: Not used. EQ.1: Not used. EQ.2: Not used. EQ.3: Notch stress value at failure. EQ.4: Stress intensity factor value at failure EQ.5: Structural stress value at failure.

Remarks:

The weld material is modeled with isotropic hardening plasticity coupled to four 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:

$$\left(\frac{\max(N_{rr},0)}{N_{rrF}}\right)^2 + \left(\frac{N_{rs}}{N_{rsF}}\right)^2 + \left(\frac{N_{rt}}{N_{rtF}}\right)^2 + \left(\frac{M_{rr}}{M_{rrF}}\right)^2 + \left(\frac{M_{ss}}{M_{ssF}}\right)^2 + \left(\frac{M_{tt}}{M_{ttF}}\right)^2 - 1 = 0$$

where the *numerators* in the equation are the resultants calculated in the local coordinates of the cross section, and the **denominators** are the values specified in the input. If NF is nonzero the resultants are filtered before failure is checked. The stress based failure model, which was developed by *Toyota Motor Corporation* and is based on the peak axial and transverse shear stresses, fails the entire weld if the stresses are outside of the failure surface defined by

$$\left(\frac{\sigma_{rr}}{\sigma_{rr}^F}\right)^2 + \left(\frac{\tau}{\tau^F}\right)^2 - 1 = 0$$

If strain rates are considered then the failure criteria becomes:

$$\left(\frac{\sigma_{rr}}{\sigma_{rr}^F(\dot{\epsilon}_{eff})}\right)^2 + \left(\frac{\tau}{\tau^F(\dot{\epsilon}_{eff})}\right)^2 - 1 = 0$$

where $\sigma_{rr}^F(\dot{\epsilon}_{eff})$ and $\tau^F(\dot{\epsilon}_{eff})$ are defined by load curves. The peak stresses are calculated from the resultants using simple beam theory.

$$\sigma_{rr} = \frac{N_{rr}}{A} + \frac{\sqrt{M_{rs}^2 + M_{rt}^2}}{Z} \quad \tau = \frac{M_{rr}}{2Z} + \frac{\sqrt{N_{rs}^2 + N_{rt}^2}}{A}$$

where the area and section modulus are given by:

$$A = \pi \frac{d^2}{4}$$

$$Z = \pi \frac{d^3}{32}$$

and d is either the diameter of the spot weld beam or the square root of the area divided by pi of the face of a solid element used as a spotweld. The last failure model is a user-written subroutine *uweldfail*, documented in Appendix N.

The failure based on notch stress, see Zhang [1999], occurs when the failure criterion:

$$\sigma_k - \sigma_{kF} \geq 0$$

is satisfied. The notch stress is give by the equation:

$$\sigma_k = \alpha_1 \frac{4F}{\pi dt} \left(1 + \frac{\sqrt{3} + \sqrt{19}}{8\sqrt{\pi}} \sqrt{\frac{t}{\rho}}\right) + \alpha_2 \frac{6M}{\pi dt^2} \left(1 + \frac{2}{\sqrt{3}\pi} \sqrt{\frac{t}{\rho}}\right) + \alpha_3 \frac{4F_{rr}}{\pi dt^2} \left(1 + \frac{5}{3\sqrt{2}\pi} \frac{d}{t} \sqrt{\frac{t}{\rho}}\right)$$

Here,

$$F = \sqrt{F_{rs}^2 + F_{rt}^2}$$

$$M = \sqrt{M_{ss}^2 + M_{tt}^2}$$

and α_i $i = 1, 2, 3$ are input corrections factors with default values of unity. If spot welds are between sheets of unequal thickness, the minimum thickness of the spot welded sheets may be introduced as a crude approximation.

The failure based on structural stress occurs, see Zhang [1999], when the failure criterion:

$$K_{eq} - K_{eqF} \geq 0$$

is satisfied where

$$K_{eq} = \sqrt{K_I^2 + K_{II}^2}$$

and

$$K_I = \alpha_1 \frac{\sqrt{3}F}{2\pi d\sqrt{t}} + \alpha_2 \frac{2\sqrt{3}M}{\pi dt\sqrt{t}} + \alpha_3 \frac{5\sqrt{2}F_{rr}}{3\pi d\sqrt{t}}$$

$$K_{II} = \alpha_1 \frac{2F}{\pi d\sqrt{t}}$$

Here, F and M are as defined above for the notch stress formulas and again, α_i $i = 1, 2, 3$ are input corrections factors with default values of unity. If spot welds are between sheets of unequal thickness, the mini

The maximum structural stress at the spot weld was utilized successfully for predictiong the fatigue failure of spot welds, see Rupp, *et. al.* [1994] and Sheppard [1993]. The corresponding results according to Rupp, *et. al.* are listed below where it is assumed that they may be suitable for crash conditions.

The failure criterion is given by:

$$\max(\sigma_{v1}, \sigma_{v2}, \sigma_{v3}) - \sigma_{sF} = 0$$

where σ_{sF} is the critical value of structural stress at failure. It is noted that the forces and moments in the equations below are referred to the beam nodes 1, 2, and to the mid point, respectively. The three stress values, $\sigma_{v1}, \sigma_{v2}, \sigma_{v3}$, are defined by :

$$\sigma_{v1}(\zeta) = \frac{F_{rs1}}{\pi dt_1} \cos \zeta + \frac{F_{rr1}}{\pi dt_1} \sin \zeta - \frac{1.046\beta_1 F_{rr1}}{t_1\sqrt{t_1}} - \frac{1.123M_{ss1}}{dt_1\sqrt{t_1}} \sin \zeta + \frac{1.123M_{tt1}}{dt_1\sqrt{t_1}} \cos \zeta \quad \text{with}$$

$$\beta_1 = 0 \quad \text{if } F_{rr1} \leq 0$$

$$\beta_1 = 1 \quad \text{if } F_{rr1} > 0$$

$$\sigma_{v2}(\zeta) = \frac{F_{rs2}}{\pi dt_2} \cos \zeta + \frac{F_{rr2}}{\pi dt_2} \sin \zeta - \frac{1.046\beta_2 F_{rr2}}{t_2\sqrt{t_2}} + \frac{1.123M_{ss2}}{dt_2\sqrt{t_2}} \sin \zeta - \frac{1.123M_{tt2}}{dt_2\sqrt{t_2}} \cos \zeta \quad \text{with}$$

$$\beta_2 = 0 \quad \text{if } F_{rr2} \leq 0$$

$$\beta_2 = 1 \quad \text{if } F_{rr2} > 0$$

$$\sigma_{v3}(\zeta) = 0.5\sigma(\zeta) + 0.5\sigma(\zeta)\cos(2\alpha) + 0.5\tau(\zeta)\sin(2\alpha)$$

where

$$\begin{aligned}\sigma(\zeta) &= \frac{4\beta_3 F_{rr}}{\pi d^2} + \frac{32M_{ss}}{\pi d^3} \sin \zeta - \frac{32M_{tt}}{\pi d^3} \cos \zeta \\ \tau(\zeta) &= \frac{16F_{rs}}{3\pi d^2} \sin^2 \zeta + \frac{16F_{rt}}{3\pi d^2} \cos^2 \zeta \\ \alpha &= \frac{1}{2} \tan^{-1} \frac{2\tau(\zeta)}{\sigma(\zeta)}\end{aligned}\quad \text{with} \quad \begin{aligned}\beta_3 &= 0 \text{ if } F_{rr} \leq 0 \\ \beta_3 &= 1 \text{ if } F_{rr} > 0\end{aligned}$$

The stresses are calculated for all directions, $0^\circ \leq \zeta \leq 90^\circ$, in order to find the maximum.

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} is nonzero, the failure surface is reduced to $|N_{rr}| = N_{rr}$. None, either, or both of the failure models may be active depending on the specified input values.

The inertias of the spot welds are scaled during the first time step so that their stable time step size is Δt . A strong compressive load on the spot weld at a later time may reduce the length of the spot weld so that stable time step size drops below Δt . If the value of Δt is zero, mass scaling is not performed, and the spot welds will probably limit the time step size. Under most circumstances, the inertias of the spot welds are small enough that scaling them will have a negligible effect on the structural response and the use of this option is encouraged.

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.

The constitutive properties for the damaged material are obtained from the undamaged material properties. The amount of damage evolved is represented by the constant, ω , which varies from zero if no damage has occurred to unity for complete rupture. For uniaxial loading, the nominal stress in the damaged material is given by

$$\sigma_{nominal} = \frac{P}{A}$$

where P is the applied load and A is the surface area. The true stress is given by:

$$\sigma_{true} = \frac{P}{A - A_{loss}}$$

where A_{loss} is the void area. The damage variable can then be defined:

$$\omega = \frac{A_{loss}}{A} \quad 0 \leq \omega \leq 1$$

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.

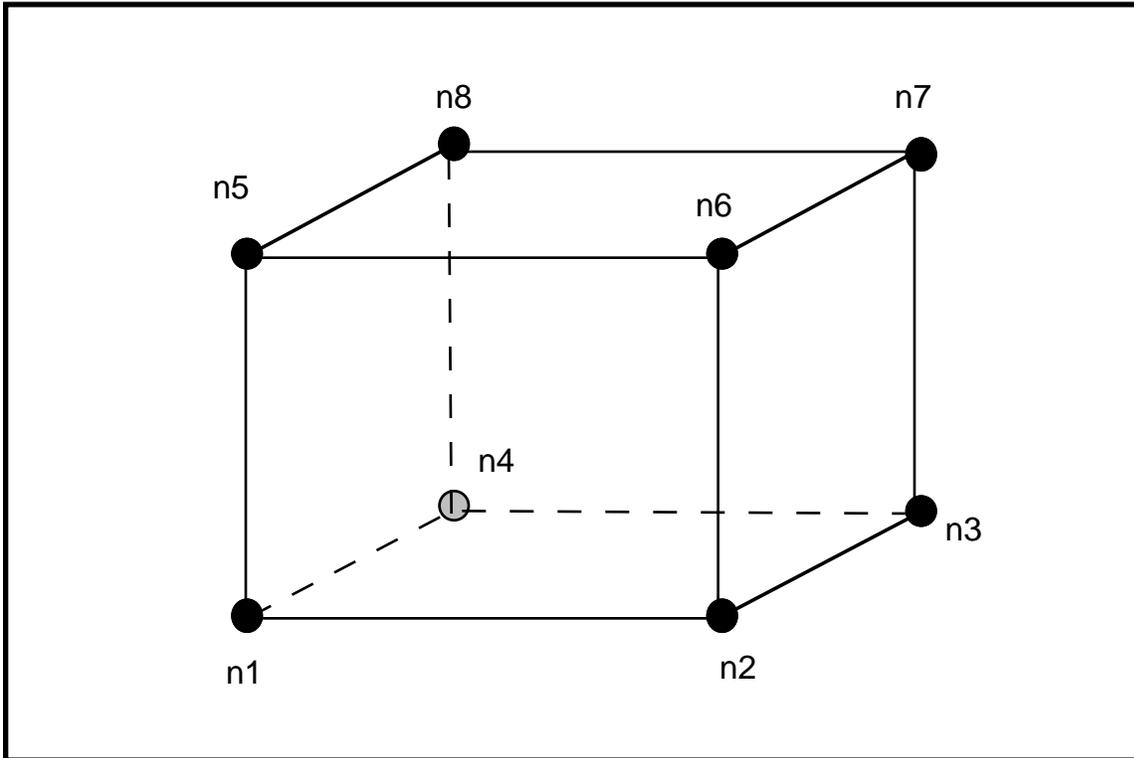


Figure 20.33. A solid element used as spot weld is shown. When resultant based failure is used orientation is very important. Nodes n1-n4 attach to the lower shell mid-surface and nodes n5-n8 attach to the upper shell mid-surface. The resultant forces and moments are computed based on the assumption that the brick element is properly oriented.

*MAT_GEPLASTIC_SRATE_2000a

This is Material Type 101. The GEPLASTIC_SRATE_2000a material model characterizes General Electric's commercially available engineering thermoplastics subjected to high strain rate events. This material model features the variation of yield stress as a function of strain rate, cavitation effects of rubber modified materials and automatic element deletion of either ductile or brittle materials.

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	MID	RO	E	PR	RATESF	EDOT0	ALPHA	
Type	I	F	F	F	F	F	F	

Card 2 1 2 3 4 5 6 7 8

Variable	LCSS	LCFEPS	LCFSIG	LCE				
Type	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.
RATESF	Constant in plastic strain rate equation.
EDOT0	Reference strain rate
ALPHA	Pressure sensitivity factor
LCSS	Load curve ID or Table ID that defines the post yield material behavior. The values of this stress-strain curve are the difference of the yield stress and strain respectively. This means the first values for both stress and strain should be zero. All subsequent values will define softening or hardening.
LCFEPS	Load curve ID that defines the plastic failure strain as a function of strain rate.

<u>VARIABLE</u>	<u>DESCRIPTION</u>
LCFSIG	Load curve ID that defines the Maximum principal failure stress as a function of strain rate.
LCE	Load curve ID that defines the Unloading moduli as a function of plastic strain.

Remarks:

The constitutive model for this approach is:

$$\dot{\epsilon}_p = \dot{\epsilon}_0 \exp(A\{\sigma - S(\epsilon_p)\}) \times \exp(-p\alpha A)$$

where $\dot{\epsilon}_0$ and A are rate dependent yield stress parameters, $S(\epsilon_p)$ internal resistance (strain hardening) and α is a pressure dependence parameter.

In this material the yield stress may vary throughout the finite element model as a function of strain rate and hydrostatic stress. Post yield stress behavior is captured in material softening and hardening values. Finally, ductile or brittle failure measured by plastic strain or maximum principal stress respectively is accounted for by automatic element deletion.

Although this may be applied to a variety of engineering thermoplastics, GE Plastics have constants available for use in a wide range of commercially available grades of their engineering thermoplastics.

<u>VARIABLE</u>	<u>DESCRIPTION</u>
A	See remarks.
Q	See remarks.
G	See remarks.

Remarks:

Resistance to deformation is both temperature and strain rate dependent. The flow stress equation is:

$$\sigma = \frac{1}{\alpha} \sinh^{-1} \left(\left[\frac{Z}{A} \right]^{\frac{1}{N}} \right)$$

where Z , the Zener-Holloman temperature compensated strain rate, is:

$$Z = \dot{\epsilon} \exp \left(\frac{Q}{GT} \right)$$

The units of the material constitutive constants are as follows: A (1/sec), N (dimensionless), α (1/MPa), the activation energy for flow, Q (J/mol), and the universal gas constant, G (J/mol K). The value of G will only vary with the unit system chosen. Typically it will be either 8.3144 J/mol ∞ K, or 40.8825 lb in/mol ∞ R.

The final equation necessary to complete our description of high strain rate deformation is one that allows us to compute the temperature change during the deformation. In the absence of a 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. Thus the heat generation coefficient is

$$HC \approx \frac{0.9}{\rho C_v}$$

where ρ is the density of the material and C_v is the specific heat.

*MAT_ANISOTROPIC_VISCOPLASTIC

This is Material Type 103. This anisotropic-viscoplastic material model applies to shell and brick elements. The material constants may be fit directly or, if desired, stress versus strain data may be input and a least squares fit will be performed by LS-DYNA to determine the constants. Kinematic or 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	

<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	Initial yield stress
FLAG	Flag EQ.0: Give all material parameters EQ.1: Material parameters are fit in LS-DYNA to Load curve or Table given below. The parameters Q_{r1} , C_{r1} , Q_{r2} , and C_{r2} for isotropic hardening are determined by the fit and those for kinematic hardening are found by scaling those for isotropic hardening by $(1 - \alpha)$ where α is defined below in columns 51-60. EQ.2: Use load curve directly, i.e., no fitting is required for the parameters Q_{r1} , C_{r1} , Q_{r2} , and C_{r2} .
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.

<u>VARIABLE</u>	<u>DESCRIPTION</u>
ALPHA	α distribution of hardening used in the curve-fitting. $\alpha = 0$ pure kinematic hardening and $\alpha = 1$ provides pure isotropic hardening
QR1	Isotropic hardening parameter Q_{r1}
CR1	Isotropic hardening parameter C_{r1}
QR2	Isotropic hardening parameter Q_{r2}
CR2	Isotropic hardening parameter C_{r2}
QX1	Kinematic hardening parameter $Q_{\chi1}$
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)

VARIABLE	DESCRIPTION
AOPT	<p>Material axes option (see MAT_OPTION TROPIC_ELASTIC for a more complete description):</p> <p>EQ. 0.0: locally orthotropic with material axes determined by element nodes 1, 2, and 4, as with *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. This option is for solid elements only.</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: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle, BETA, from a line in the plane of the element defined by the cross product of the vector v with the element normal.</p> <p>EQ. 4.0: locally orthotropic in cylindrical coordinate system with the material axes determined by a vector v, and an originating point, P, which define the centerline axis. This option is for solid elements only.</p>
XP,YP,ZP	x _p y _p z _p , define coordinates of point p for AOPT = 1 and 4.
A1,A2,A3	a ₁ a ₂ a ₃ , define components of vector a for AOPT = 2.
D1,D2,D3	d ₁ d ₂ d ₃ , define components of vector d for AOPT = 2.
V1,V2,V3	v ₁ v ₂ v ₃ , define components of vector v for AOPT = 3 and 4.
BETA	Material angle in degrees for AOPT = 3, may be overridden on the element card, see *ELEMENT_SHELL_BETA or *ELEMENT_SOLID_ORTHO.

Remarks:

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 ϵ_{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. The viscoplastic rate formulation is an option in other plasticity models in LS-DYNA, e.g., mat_3 and mat_24, invoked by setting the parameter VP to 1.

***MAT_ANISOTROPIC_PLASTIC**

This is Material Type 103. This anisotropic-plastic material model is a simplified version of the MAT_ANISOTROPIC_VISCOPLASTIC above. This material model applies only to shell elements.

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	MID	RO	E	PR	SIGY	LCSS		
Type	I	F	F	F	F	F		

Card 2 1 2 3 4 5 6 7 8

Variable	QR1	CR1	QR2	CR2				
Type	F	F	F	F				

Card 3 1 2 3 4 5 6 7 8

Variable	R00	R45	R90	S11	S22	S33	S12	
Type	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
LCSS	Load curve ID. The load curve ID defines effective stress versus effective plastic strain. Card 2 is ignored with this option.
QR1	Isotropic hardening parameter Q_{r1}
CR1	Isotropic hardening parameter C_{r1}
QR2	Isotropic hardening parameter Q_{r2}
CR2	Isotropic hardening parameter C_{r2}
R00	R_{00} for anisotropic hardening
R45	R_{45} for anisotropic hardening
R90	R_{90} for anisotropic hardening

<u>VARIABLE</u>	<u>DESCRIPTION</u>
S11	Yield stress in local x-direction. This input is ignored if $(R_{00}, R_{45}, R_{90}) > 0$.
S22	Yield stress in local y-direction. This input is ignored if $(R_{00}, R_{45}, R_{90}) > 0$.
S33	Yield stress in local z-direction. This input is ignored if $(R_{00}, R_{45}, R_{90}) > 0$.
S12	Yield stress in local xy-direction. This input is ignored if $(R_{00}, R_{45}, R_{90}) > 0$.
AOPT	Material axes option (see MAT_OPTION TROPIC_ELASTIC for a more complete description): EQ. 0.0: locally orthotropic with material axes determined by element nodes 1, 2, and 4, as with *DEFINE_COORDINATE_NODES. EQ. 1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center; this is the a-direction. This option is for solid elements only. EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR. EQ. 3.0: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle, BETA, from a line in the plane of the element defined by the cross product of the vector v with the element normal. EQ. 4.0: locally orthotropic in cylindrical coordinate system with the material axes determined by a vector v , and an originating point, P, which define the centerline axis. This option is for solid elements only.
XP,YP,ZP	$x_p y_p z_p$, define coordinates of point p for AOPT = 1 and 4.
A1,A2,A3	$a_1 a_2 a_3$, define components of vector a for AOPT = 2.
D1,D2,D3	$d_1 d_2 d_3$, define components of vector d for AOPT = 2.
V1,V2,V3	$v_1 v_2 v_3$, define components of vector v for AOPT = 3 and 4.
BETA	Material angle in degrees for AOPT = 3, may be overridden on the element card, see *ELEMENT_SHELL_BETA or *ELEMENT_SOLID_ORTHO.

Remarks:

If no load curve is defined for the effective stress versus effective plastic strain, the uniaxial stress-strain curve is given on the following form

$$\sigma(\epsilon_{eff}^p) = \sigma_0 + Q_{r1}(1 - \exp(-C_{r1}\epsilon_{eff}^p)) + Q_{r2}(1 - \exp(-C_{r2}\epsilon_{eff}^p))$$

where ϵ_{eff}^p is the effective plastic strain. For shells the anisotropic behavior is given by R_{00} , R_{45} and R_{90} , or the yield stress in the different direction. Default values are given by:

$$R_{00} = R_{45} = R_{90} = 1$$

if the variables R00, R45, R90, S11, S22, S33 and S12 are set to zero.

***MAT_DAMAGE_1**

This is Material Type 104. This is a continuum damage mechanics (CDM) model which includes anisotropy and viscoplasticity. The CDM model applies to shell, thick shell, and brick elements. A more detailed description of this model can be found in the paper by Berstad, Hopperstad, Lademo, and Malo[1999]. This material model can also model anisotropic damage behavior by setting the FLAG to -1 in Card 2.

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	MID	RO	E	PR	SIGY	LCSS	LCDS	
Type	I	F	F	F	F			

Card 2 1 2 3 4 5 6 7 8

Variable	Q1	C1	Q2	C2	EPSD	S or EPSR	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, σ_0 .
LCSS	Load curve ID. Load curve ID defining effective stress versus effective plastic strain. For FLAG = -1.
LCDS	Load curve ID defining nonlinear damage curve. For FLAG = -1.
Q1	Isotropic hardening parameter Q_1
C1	Isotropic hardening parameter C_1
Q2	Isotropic hardening parameter Q_2
C2	Isotropic hardening parameter C_2
EPSD	Damage threshold r_d Damage effective plastic strain when material softening begin. (Default=0.0)
S	Damage material constant S . (Default= $\frac{\sigma_0}{200}$). For FLAG ≥ 0 .

VARIABLE	DESCRIPTION
EPSR	Plastic strain at which material ruptures (logarithmic).
DC	Critical damage value D_c . When the damage value D reaches this value, the element is deleted from the calculation. (Default=0.5) For FLAG ≥ 0.
FLAG	Flag EQ.-1. Anisotropic damage EQ.0.No calculation of localization due to damage EQ.1.The model flags element where strain localization occur
VK	Viscous material parameter V_k
VM	Viscous material parameter V_m
R00	R_{00} for shell (Default=1.0)
R45	R_{45} for shell (Default=1.0)
R90	R_{90} for shell (Default=1.0)
F	F for brick (Default =1/2)
G	G for brick (Default =1/2)
H	H for brick (Default =1/2)
L	L for brick (Default =3/2)
M	M for brick (Default =3/2)
N	N for brick (Default =3/2)
AOPT	Material axes option (see MAT_OPTION TROPIC_ELASTIC for a more complete description): EQ. 0.0: locally orthotropic with material axes determined by element nodes 1, 2, and 4, as with *DEFINE_COORDINATE_NODES. EQ. 1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center; this is the a-direction. This option is for solid elements only. EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR. EQ. 3.0: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle, BETA, from a line in the plane of the element defined by the cross product of the vector \mathbf{v} with the element normal. EQ. 4.0: locally orthotropic in cylindrical coordinate system with the material axes determined by a vector \mathbf{v} , and an originating point, P, which define the centerline axis. This option is for solid elements only.

VARIABLE	DESCRIPTION
XP,YP,ZP	x_p y_p z_p , define coordinates of point p for AOPT = 1 and 4.
A1,A2,A3	a_1 a_2 a_3 , define components of vector a for AOPT = 2.
D1,D2,D3	d_1 d_2 d_3 , define components of vector d for AOPT = 2.
V1,V2,V3	v_1 v_2 v_3 , define components of vector v for AOPT = 3 and 4.
BETA	Material angle in degrees for AOPT = 3, may be overridden on the element card, see *ELEMENT_SHELL_BETA or *ELEMENT_SOLID_ORTHO.

Remarks:

Anisotropic Damage model (FLAG = -1). At each thickness integration points, an anisotropic damage law acts on the plane stress tensor in the directions of the principal total shell strains, ϵ_1 and ϵ_2 , as follows:

$$\begin{aligned}\sigma_{11} &= (1 - D_1(\epsilon_1))\sigma_{110} \\ \sigma_{22} &= (1 - D_2(\epsilon_2))\sigma_{220} \\ \sigma_{12} &= (1 - (D_1 + D_2)/2)\sigma_{120}\end{aligned}$$

The transverse plate shear stresses in the principal strain directions are assumed to be damaged as follows:

$$\begin{aligned}\sigma_{13} &= (1 - D_1/2)\sigma_{130} \\ \sigma_{23} &= (1 - D_2/2)\sigma_{230}\end{aligned}$$

In the anisotropic damage formulation, $D_1(\epsilon_1)$ and $D_2(\epsilon_2)$ are anisotropic damage functions for the loading directions 1 and 2, respectively. Stresses σ_{110} , σ_{220} , σ_{120} , σ_{130} and σ_{230} are stresses in the principal shell strain directions as calculated from the undamaged elastic-plastic material behavior. The strains ϵ_1 and ϵ_2 are the magnitude of the principal strains calculated upon reaching the damage thresholds. Damage can only develop for tensile stresses, and the damage functions $D_1(\epsilon_1)$ and $D_2(\epsilon_2)$ are identical to zero for negative strains ϵ_1 and ϵ_2 . The principal strain directions are fixed within an integration point as soon as either principal strain exceeds the initial threshold strain in tension. A more detailed description of the damage evolution for this material model is given in the description of material #81.

The Continuum Damage Mechanics (CDM) model (FLAG ≥ 0) is based on a CDM model proposed by Lemaitre [1992]. The effective stress $\tilde{\sigma}$, which is the stress calculated over the section that effectively resist the forces and reads.

$$\tilde{\sigma} = \frac{\sigma}{1 - D}$$

where D is the damage variable. The evolution equation for the damage variable is defined as

$$\dot{D} = \begin{cases} 0 & \text{for } r \leq r_D \\ \frac{Y}{S(1-D)} \dot{r} & \text{for } r > r_D \text{ and } \sigma_1 > 0 \end{cases}$$

where r_D is the damage threshold,

is a positive material constant, S is the so-called strain energy release rate and σ_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 σ_{vm} is the equivalent von Mises stress. The triaxiality function R_v is defined as

$$R_v = \frac{2}{3}(1+\nu) + 3(1-2\nu) \left(\frac{\sigma_H}{\sigma_{vm}} \right)^2$$

The uniaxial stress-strain curve is given in the following form

$$\begin{aligned} \sigma(r, \dot{\epsilon}_{eff}^p) = & \sigma_0 + Q_1(1 - \exp(-C_1 r)) + Q_2(1 - \exp(-C_2 r)) \\ & + V_k \dot{\epsilon}_{eff}^p V_m \end{aligned}$$

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

$$\begin{aligned} F(\tilde{\sigma}_{22} - \tilde{\sigma}_{33})^2 + G(\tilde{\sigma}_{33} - \tilde{\sigma}_{11})^2 + H(\tilde{\sigma}_{11} - \tilde{\sigma}_{22})^2 \\ + 2L\tilde{\sigma}_{23}^2 + 2M\tilde{\sigma}_{31}^2 + 2N\tilde{\sigma}_{12}^2 = \sigma(r, \dot{\epsilon}_{eff}^p) \end{aligned}$$

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.
C	Strain rate parameter, C, see formula below.
P	Strain rate parameter, P, see formula below.

VARIABLE	DESCRIPTION
LCSS	Load curve ID or Table ID. Load curve ID defining effective stress versus effective plastic strain. If defined EPS1-EPS8 and ES1-ES8 are ignored. The table ID defines for each strain rate value a load curve ID giving the stress versus effective plastic strain for that rate, See Figure 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_106

*MAT_ELASTIC_VISCOPLASTIC_THERMAL

*MAT_ELASTIC_VISCOPLASTIC_THERMAL

This is Material Type 106. This is an elastic viscoplastic material with thermal effects.

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	MID	RO	E	PR	SIGY	ALPHA	LCSS	
Type	I	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	C	P	LCE	LCPR	LCSIGY	LCR	LCX	LCALPH
Type	F	F	F	F	F	F	F	F

Card 4 1 2 3 4 5 6 7 8

Variable	LCC	LCP						
Type	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

VARIABLE	DESCRIPTION
SIGY	Initial yield stress
LCSS	Load curve ID. The load curve ID defines effective stress versus effective plastic strain. Card 2 is ignored with this option.
ALPHA	Coefficient of thermal expansion.
QR1	Isotropic hardening parameter Q_{r1}
CR1	Isotropic hardening parameter C_{r1}
QR2	Isotropic hardening parameter Q_{r2}
CR2	Isotropic hardening parameter C_{r2}
QX1	Kinematic hardening parameter $Q_{\chi1}$
CX1	Kinematic hardening parameter $C_{\chi1}$
QX2	Kinematic hardening parameter $Q_{\chi2}$
CX2	Kinematic hardening parameter $C_{\chi2}$
C	Viscous material parameter C
P	Viscous material parameter P
LCE	Load curve defining Young's modulus as a function of temperature. E on card 1 is ignored with this option.
LCPR	Load curve defining Poisson's ratio as a function of temperature. PR on card 1 is ignored with this option.
LCSIGY	Load curve defining the initial yield stress as a function of temperature. SIGY on card 1 is ignored with this option.
LCR	Load curve for scaling the isotropic hardening parameters QR1 and QR2 or the stress given by the load curve LCSS as a function of temperature.
LCX	Load curve for scaling the isotropic hardening parameters QX1 and QX2 as a function of temperature.
LCALPH	Load curve defining the coefficient of thermal expansion as a function of temperature. ALPHA on card 1 is ignored with this option.

<u>VARIABLE</u>	<u>DESCRIPTION</u>
LCC	Load curve for scaling the viscous material parameter C as a function of temperature.
LCP	Load curve for scaling the viscous material parameter P as a function of temperature.

Remarks:

If LCSS is not given any value the uniaxial stress-strain curve has the form

$$\begin{aligned}\sigma(\varepsilon_{eff}^p) = & \sigma_0 + Q_{r1}(1 - \exp(-C_{r1}\varepsilon_{eff}^p)) + Q_{r2}(1 - \exp(-C_{r2}\varepsilon_{eff}^p)) \\ & + Q_{\chi1}(1 - \exp(-C_{\chi1}\varepsilon_{eff}^p)) + Q_{\chi2}(1 - \exp(-C_{\chi2}\varepsilon_{eff}^p))\end{aligned}$$

Viscous effects are accounted for using the Cowper and Symonds model, which, scales the yield stress with the factor:

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

*MAT_JOHNSON_HOLMQUIST_CERAMICS

This is Material Type 110. This Johnson-Holmquist Plasticity Damage Model is useful for modeling ceramics, glass and other brittle materials. A more detailed description can be found in a paper by Johnson and Holmquist [1993].

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	MID	RO	G	A	B	C	M	N
Type	I	F	F	F	F	F	F	F

Card 2 1 2 3 4 5 6 7 8

Variable	EPSI	T	SFMAX	HEL	PHEL	BETA		
Type	F	F	F	F	F	F		

Card 3 1 2 3 4 5 6 7 8

Variable	D1	D2	K1	K2	K3	FS		
Type	F	F	F	F	F	F		

VARIABLE

DESCRIPTION

- MID Material identification. A unique number has to be chosen.
- RO Density
- G Shear modulus
- A Intact normalized strength parameter
- B Fractured normalized strength parameter
- C Strength parameter (for strain rate dependence)
- M Fractured strength parameter (pressure exponent)
- N Intact strength parameter (pressure exponent).

VARIABLE	DESCRIPTION
EPSI	Reference strain rate.
T	Maximum tensile strength.
SFMAX	Maximum normalized fractured strength (if Eq.0, defaults to 1e20).
HEL	Hugoniot elastic limit.
PHEL	Pressure component at the Hugoniot elastic limit.
BETA	Fraction of elastic energy loss converted to hydrostatic energy.
D1	Parameter for plastic strain to fracture.
D2	Parameter for plastic strain to fracture (exponent).
K1	First pressure coefficient (equivalent to the bulk modulus).
K2	Second pressure coefficient.
K3	Elastic constants (k1 is the bulk modulus).
FS	Failure criteria. FS < 0 fail if $p^* + t^* < 0$ (tensile failure). FS = 0 no failure (default). FS > 0 fail if the strain > FS.

Remarks:

The equivalent stress for a ceramic-type material is given by

$$\sigma^* = \sigma_i^* - D(\sigma_i^* - \sigma_f^*)$$

where

$$\sigma_i^* = a(p^* + t^*)^n (1 + c \ln \dot{\epsilon})$$

represents the intact, undamaged behaviour,

$$D = \sum \Delta \epsilon^p / \epsilon_f^p$$

represents the accumulated damage based upon the increase in plastic strain per computational cycle and the plastic strain to fracture

$$\epsilon_f^p = d_1 (p^* + t^*)^{d_2}$$

and

$$\sigma_f^* = b(p^*)^m (1 + c \ln \dot{\epsilon}) \leq sfma$$

represents the damaged behaviour. In each case, the '*' indicates a normalized quantity, the stresses being normalized by the equivalent stress at the Hugoniot elastic limit (see below), the pressures by the pressure at the Hugoniot elastic limit (see below) and the strain rate by the reference strain rate. The parameter d1 controls the rate at which damage accumulates. If it is made 0, full damage occurs in one time step i.e. instantaneously. It is also the best parameter to vary if one attempts to reproduce results generated by another finite element program.

In undamaged material, the hydrostatic pressure is given by

$$P = k_1 \mu + k_2 \mu^2 + k_3 \mu^3$$

where $\mu = \rho / \rho_0 - 1$. When damage starts to occur, there is an increase in pressure. A fraction, between 0 and 1, of the elastic energy loss, β , is converted into hydrostatic potential energy (pressure). The details of this pressure increase are given in the reference.

Given *hel* and *g*, μ_{hel} can be found iteratively from

$$hel = k_1 \mu_{hel} + k_2 \mu_{hel}^2 + k_3 \mu_{hel}^3 + (4/3)g(\mu_{hel}/(1 + \mu_{hel}))$$

and, subsequently, for normalization purposes,

$$p_{hel} = k_1 \mu_{hel} + k_2 \mu_{hel}^2 + k_3 \mu_{hel}^3$$

and

$$\sigma_{hel} = 1.5(hel - p_{hel})$$

These are calculated automatically by LS-DYNA if p_{hel} is zero on input.

***MAT_JOHNSON_HOLMQUIST_CONCRETE**

This is Material Type 111. This model can be used for concrete subjected to large strains, high strain rates and high pressures. The equivalent strength is expressed as a function of the pressure, strain rate, and damage. The pressure is expressed as a function of the volumetric strain and includes the effect of permanent crushing. The damage is accumulated as a function of the plastic volumetric strain, equivalent plastic strain and pressure. A more detailed of this model can be found in the paper by Holmquist, Johnson, and Cook [1993].

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	MID	RO	G	A	B	C	N	FC
Type	I	F	F	F	F	F	F	F

Card 2 1 2 3 4 5 6 7 8

Variable	T	EPS0	EFMIN	SFMAX	PC	UC	PL	UL
Type	F	F	F	F	F	F	F	F

Card 3 1 2 3 4 5 6 7 8

Variable	D1	D2	K1	K2	K3	FS		
Type	F	F	F	F	F	F		

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MID	Material identification. A unique number must be chosen.
RO	Mass density.
G	Shear modulus.
A	Normalized cohesive strength.
B	Normalized pressure hardening.
C	Strain rate coefficient.

<u>VARIABLE</u>	<u>DESCRIPTION</u>
N	Pressure hardening exponent.
FC	Quasi-static uniaxial compressive strength.
T	Maximum tensile hydrostatic pressure.
EPS0	Reference strain rate.
EFMIN	Amount of plastic strain before fracture.
SFMAX	Normalized maximum strength..
PC	Crushing pressure.
UC	Crushing volumetric strain.
PL	Locking pressure.
UL	Locking volumetric strain.
D1	Damage constant.
D2	Damage constant.
K1	Pressure constant.
K2	Pressure constant.
K3	Pressure constant.
FS	Failure type

Remarks:

The normalized equivalent stress is defined as

$$\sigma^* = \frac{\sigma}{f_c'}$$

where σ is the actual equivalent stress, and f_c' is the quasi-static uniaxial compressive strength. The expression is defined as

$$\sigma^* = [A(1 - D) + BP^{*N}][1 - c \ln(\dot{\epsilon}^*)]$$

where D is the damage parameter, $P^* = P/f_c'$ is the normalized pressure and $\dot{\epsilon}^* = \dot{\epsilon}/\dot{\epsilon}_0$ is the dimensionless strain rate. The model accumulates damage both from equivalent plastic strain and

plastic volumetric strain, and is expressed as

$$D = \sum \frac{\Delta \varepsilon_p + \Delta \mu_p}{D_1(P^* + T^*)^{D_2}}$$

where $\Delta \varepsilon_p$ and $\Delta \mu_p$ are the equivalent plastic strain and plastic volumetric strain, D_1 and D_2 are material constants and $T^* = T / f_c'$ is the normalized maximum tensile hydrostatic pressure.

The pressure for fully dense material is expressed as

$$P = K_1 \bar{\mu} + K_2 \bar{\mu}^2 + K_3 \bar{\mu}^3$$

where K_1 , K_2 and K_3 are material constants and the modified volumetric strain is defined as

$$\bar{\mu} = \frac{\mu - \mu_{lock}}{1 + \mu_{lock}}$$

where μ_{lock} is the locking volumetric strain.

*MAT_FINITE_ELASTIC_STRAIN_PLASTICITY

This is Material Type 112. An elasto-plastic material with an arbitrary stress versus strain curve and arbitrary strain rate dependency can be defined. The elastic response of this model uses a finite strain formulation so that large elastic strains can develop before yielding occurs. This model is available for solid elements only. See Remarks below.

Card 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		

Card 2

Variable	C	P	LCSS	LCSR				
Type	F	F	F	F				
Default	0	0	0	0				

Card 3

Variable	EPS1	EPS2	EPS3	EPS4	EPS5	EPS6	EPS7	EPS8
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

Card 4

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 must 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; the curve ID, LCSR; EPS1-EPS8 and ES1-ES8 are ignored if a Table ID is defined.

VARIABLE	DESCRIPTION
LCSR	Load curve ID defining strain rate scaling effect on yield stress.
EPS1-EPS8	Effective plastic strain values (optional if SIGY is defined). At least 2 points should be defined. The first point must be zero corresponding to the initial yield stress. WARNING: If the first point is nonzero the yield stress is extrapolated to determine the initial yield. If this option is used SIGY and ETAN are ignored and may be input as zero.
ES1-ES8	Corresponding yield stress values to EPS1 - EPS8.

Remarks:

The stress strain behavior may be treated by a bilinear stress strain curve by defining the tangent modulus, ETAN. Alternately, a curve similar to that shown in Figure 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. $\dot{\epsilon} = \sqrt{\dot{\epsilon}_{ij} \dot{\epsilon}_{ij}}$.

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.

***MAT_LAYERED_LINEAR_PLASTICITY**

This is Material Type 114. A layered elastoplastic material with an arbitrary stress versus strain curve and an arbitrary strain rate dependency can be defined. This material must be used with the user defined integration rules, see *INTEGRATION-SHELL, for modeling laminated composite and sandwich shells where each layer can be represented by elastoplastic behavior with constitutive constants that vary from layer to layer. Lamination theory is applied to correct for the assumption of a uniform constant shear strain through the thickness of the shell. Unless this correction is applied, the stiffness of the shell can be grossly incorrect leading to poor results. Generally, without the correction the results are too stiff.. This model is available for shell elements only. Also, see Remarks below.

Card 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		

Card 2

Variable	C	P	LCSS	LCSR				
Type	F	F	F	F				
Default	0	0	0	0				

Card 3

Variable	EPS1	EPS2	EPS3	EPS4	EPS5	EPS6	EPS7	EPS8
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

Card 4

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;

<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.
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. $\dot{\epsilon} = \sqrt{\dot{\epsilon}_{ij} \dot{\epsilon}_{ij}}$.

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.

*MAT_UNIFIED_CREEP

This is Material Type 115. This is an elastic creep model for modeling creep behavior when plastic behavior is not considered.

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	MID	RO	E	PR	A	N	M	
Type	I	F	F	F	F	F	F	
Default	none							

VARIABLE

DESCRIPTION

MID	Material identification. A unique number must be chosen.
RO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.
A	Stress coefficient .
N	Stress exponent.
M	Time exponent.

Remarks:

The effective creep strain, $\bar{\epsilon}^c$, given as:

$$\bar{\epsilon}^c = A\bar{\sigma}^n \bar{t}^m$$

where A , n , and m are constants and \bar{t} is the effective time. The effective stress, $\bar{\sigma}$, is defined as:

$$\bar{\sigma} = \sqrt{\frac{3}{2} \sigma_{ij} \sigma_{ij}}$$

The creep strain, therefore, is only a function of the deviatoric stresses. The volumetric behavior for this material is assumed to be elastic. By varying the time constant m primary creep ($m < 1$), secondary creep ($m = 1$), and tertiary creep ($m > 1$) can be modeled. This model is described by Whirley and Henshall (1992).

***MAT_COMPOSITE_LAYUP**

This is Material Type 116. This material is for modeling the elastic responses of composite layups that have an arbitrary number of layers through the shell thickness. A pre-integration is used to compute the extensional, bending, and coupling stiffnesses for use with the Belytschko-Tsay resultant shell formulation. The angles of the local material axes are specified from layer to layer in the *SECTION_SHELL input. This material model must be used with the user defined integration rule for shells, see *INTEGRATION_SHELL, which allows the elastic constants to change from integration point to integration point. Since the stresses are not computed in the resultant formulation, the stresses output to the binary databases for the resultant elements are zero. Note that this shell *does not use laminated shell theory* and that storage is allocated for just one integration point (as reported in D3HSP) regardless of the layers defined in the integration rule.

Card 1 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 3

Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card 4

Variable	V1	V2	V3	D1	D2	D3	BETA	
Type	F	F	F	F	F	F	F	

VARIABLE	DESCRIPTION
MID	Material identification. A unique number 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	ν_{ba} , Poisson's ratio ba.
PRCA	ν_{ca} , Poisson's ratio ca.
PRCB	ν_{cb} , Poisson's ratio cb.
GAB	G_{ab} , shear modulus ab.
GBC	G_{bc} , shear modulus bc.
GCA	G_{ca} , shear modulus ca.
AOPT	Material axes option, see Figure 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. This option is for solid elements only.</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: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle, BETA, from a line in the plane of the element defined by the cross product of the vector \mathbf{v} with the element normal. The plane of a solid element is the midsurface between the inner surface and outer surface defined by the first four nodes and the last four nodes of the connectivity of the element, respectively.</p> <p>EQ. 4.0: locally orthotropic in cylindrical coordinate system with the material axes determined by a vector \mathbf{v}, and an originating point, P, which define the centerline axis. This option is for solid elements only.</p>
XP YP ZP	Define coordinates of point p for AOPT = 1 and 4.
A1 A2 A3	Define components of vector a for AOPT = 2.

VARIABLE	DESCRIPTION
V1 V2 V3	Define components of vector v for AOPT = 3 and 4.
D1 D2 D3	Define components of vector d for AOPT = 2:
BETA	Material angle in degrees for AOPT = 3, may be overridden on the element card, see *ELEMENT_SHELL_BETA.

Remarks:

This material law is based on standard composite lay-up theory. The implementation, [See Jones 1975], allows the calculation of the force, N , and moment, M , stress resultants from:

$$\begin{Bmatrix} N_x \\ N_y \\ N_{xy} \end{Bmatrix} = \begin{bmatrix} A_{11} & A_{12} & A_{16} \\ A_{21} & A_{22} & A_{26} \\ A_{16} & A_{26} & A_{66} \end{bmatrix} \begin{Bmatrix} \epsilon_x^0 \\ \epsilon_y^0 \\ \epsilon_z^0 \end{Bmatrix} + \begin{bmatrix} B_{11} & B_{12} & B_{16} \\ B_{21} & B_{22} & B_{26} \\ B_{16} & B_{26} & B_{66} \end{bmatrix} \begin{Bmatrix} \kappa_x \\ \kappa_y \\ \kappa_{xy} \end{Bmatrix}$$

$$\begin{Bmatrix} M_x \\ M_y \\ M_{xy} \end{Bmatrix} = \begin{bmatrix} B_{11} & B_{12} & B_{16} \\ B_{21} & B_{22} & B_{26} \\ B_{16} & B_{26} & B_{66} \end{bmatrix} \begin{Bmatrix} \epsilon_x^0 \\ \epsilon_y^0 \\ \epsilon_z^0 \end{Bmatrix} + \begin{bmatrix} D_{11} & D_{12} & D_{16} \\ D_{21} & D_{22} & D_{26} \\ D_{16} & D_{26} & D_{66} \end{bmatrix} \begin{Bmatrix} \kappa_x \\ \kappa_y \\ \kappa_{xy} \end{Bmatrix}$$

where A_{ij} is the extensional stiffness, D_{ij} is the bending stiffness, and B_{ij} is the coupling stiffness which is a null matrix for symmetric lay-ups. The mid-surface strains and curvatures are denoted by ϵ_{ij}^0 and κ_{ij} , respectively. Since these stiffness matrices are symmetric, 18 terms are needed per shell element in addition to the shell resultants which are integrated in time. This is considerably less storage than would typically be required with through thickness integration which requires a minimum of eight history variables per integration point, e.g., if 100 layers are used 800 history variables would be stored. Not only is memory much less for this model, but the CPU time required is also considerably reduced.

*MAT_COMPOSITE_MATRIX

This is Material Type 117. This material is used for modeling the elastic responses of composites where a pre-integration is used to compute the extensional, bending, and coupling stiffness coefficients for use with the Belytschko-Tsay resultant shell formulation. Since the stresses are not computed in the resultant formulation, the stresses output to the binary databases for the resultant elements are zero.

Card 1 1 2 3 4 5 6 7 8

Variable	MID	RO						
Type	I	F						

Card 2

Variable	C11	C12	C22	C13	C23	C33	C14	C24
Type	F	F	F	F	F	F	F	F

Card 3

Variable	C34	C44	C15	C25	C35	C45	C55	C16
Type	F	F	F	F	F	F	F	F

Card 4

Variable	C26	C36	C46	C56	C66	AOPT		
Type	F	F	F	F	F	F		

Card 5

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	

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MID	Material identification. A unique number must be chosen.
RO	Mass density.
CIJ	C_{ij} coefficients of stiffness matrix.
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. This option is for solid elements only. EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR. EQ. 3.0: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle, BETA, from a line in the plane of the element defined by the cross product of the vector \mathbf{v} with the element normal. The plane of a solid element is the midsurface between the inner surface and outer surface defined by the first four nodes and the last four nodes of the connectivity of the element, respectively.

VARIABLE	DESCRIPTION
	EQ. 4.0: locally orthotropic in cylindrical coordinate system with the material axes determined by a vector \mathbf{v} , and an originating point, P, which define the centerline axis. This option is for solid elements only.
XP YP ZP	Define coordinates of point p for AOPT = 1 and 4.
A1 A2 A3	Define components of vector a for AOPT = 2.
V1 V2 V3	Define components of vector v for AOPT = 3 and 4.
D1 D2 D3	Define components of vector d for AOPT = 2:
BETA	Material angle in degrees for AOPT = 3, may be overridden on the element card, see *ELEMENT_SHELL_BETA.

Remarks:

The calculation of the force, N_{ij} , and moment, M_{ij} , stress resultants is given in terms of the membrane strains, ϵ_i^0 , and shell curvatures, κ_i , as:

$$\begin{Bmatrix} N_x \\ N_y \\ N_{xy} \\ M_x \\ M_y \\ M_{xy} \end{Bmatrix} = \begin{bmatrix} C_{11} & C_{12} & C_{13} & C_{14} & C_{15} & C_{16} \\ C_{21} & C_{22} & C_{23} & C_{24} & C_{25} & C_{26} \\ C_{31} & C_{32} & C_{33} & C_{34} & C_{35} & C_{36} \\ C_{41} & C_{42} & C_{43} & C_{44} & C_{45} & C_{46} \\ C_{51} & C_{52} & C_{53} & C_{54} & C_{55} & C_{56} \\ C_{61} & C_{62} & C_{63} & C_{64} & C_{65} & C_{66} \end{bmatrix} \begin{Bmatrix} \epsilon_x^0 \\ \epsilon_y^0 \\ \epsilon_z^0 \\ \kappa_x \\ \kappa_y \\ \kappa_{xy} \end{Bmatrix}$$

where $C_{ij} = C_{ji}$. In this model this symmetric matrix is transformed into the element local system and the coefficients are stored as element history variables. In model type *MAT_COMPOSITE_DIRECT below, the resultants are already assumed to be given in the element local system which reduces the storage since the 21 coefficients are not stored as history variables as part of the element data.

The shell thickness is built into the coefficient matrix and, consequently, within the part ID, which references this material ID, the thickness must be uniform.

***MAT_COMPOSITE_DIRECT**

This is Material Type 118. This material is used for modeling the elastic responses of composites where a pre-integration is used to compute the extensional, bending, and coupling stiffness coefficients for use with the Belytschko-Tsay resultant shell formulation. Since the stresses are not computed in the resultant formulation, the stresses output to the binary databases for the resultant elements are zero.

Card 1 1 2 3 4 5 6 7 8

Variable	MID	RO						
Type	I	F						

Card 2

Variable	C11	C12	C22	C13	C23	C33	C14	C24
Type	F	F	F	F	F	F	F	F

Card 3

Variable	C34	C44	C15	C25	C35	C45	C55	C16
Type	F	F	F	F	F	F	F	F

Card 4

Variable	C26	C36	C46	C56	C66			
Type	F	F	F	F	F			

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MID	Material identification. A unique number must be chosen.
RO	Mass density.
CIJ	C_{ij} coefficients of the stiffness matrix.

Remarks:

The calculation of the force, N_{ij} , and moment, M_{ij} , stress resultants is given in terms of the membrane strains, ϵ_i^0 , and shell curvatures, κ_i , as:

$$\begin{Bmatrix} N_x \\ N_y \\ N_{xy} \\ M_x \\ M_y \\ M_{xy} \end{Bmatrix} = \begin{bmatrix} C_{11} & C_{12} & C_{13} & C_{14} & C_{15} & C_{16} \\ C_{21} & C_{22} & C_{23} & C_{24} & C_{25} & C_{26} \\ C_{31} & C_{32} & C_{33} & C_{34} & C_{35} & C_{36} \\ C_{41} & C_{42} & C_{43} & C_{44} & C_{45} & C_{46} \\ C_{51} & C_{52} & C_{53} & C_{54} & C_{55} & C_{56} \\ C_{61} & C_{62} & C_{63} & C_{64} & C_{65} & C_{66} \end{bmatrix} \begin{Bmatrix} \epsilon_x^0 \\ \epsilon_y^0 \\ \epsilon_z^0 \\ \kappa_x \\ \kappa_y \\ \kappa_{xy} \end{Bmatrix}$$

where $C_{ij} = C_{ji}$. In this model the stiffness coefficients are already assumed to be given in the element local system which reduces the storage. Great care in the element orientation and choice of the local element system, see *CONTROL_ACCURACY, must be observed if this model is used.

The shell thickness is built into the coefficient matrix and, consequently, within the part ID, which references this material ID, the thickness must be uniform.

*MAT_119

*MAT_GENERAL_NONLINEAR_6DOF_DISCRETE_BEAM

*MAT_GENERAL_NONLINEAR_6DOF_DISCRETE_BEAM

This is Material Type 119. This is a very general spring and damper model. This beam is based on the MAT_SPRING_GENERAL_NONLINEAR option. Additional unloading options have been included. The two nodes defining the beam may be coincident to give a zero length beam, or offset to give a finite length beam. For finite length discrete beams the absolute value of the variable SCOR in the SECTION_BEAM input should be set to a value of 2.0 or 3.0 to give physically correct behavior. A triad is used to orient the beam for the directional springs..

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	MID	RO	KT	KR	UNLDOPT	OFFSET	DAMPF	
Type	I	F	F	F	I	F	F	

Card 2

Variable	LCIDTR	LCIDTS	LCIDTT	LCIDRR	LCIDRS	LCIDRT		
Type	I	I	I	I	I	I		

Card 3

Variable	LCIDTUR	LCIDTUS	LCIDTUT	LCIDRUR	LCIDRUS	LCIDRUT		
Type	I	I	I	I	I	I		

Card 4

Variable	LCIDTDR	LCIDTDS	LCIDTDT	LCIDRDR	LCIDRDS	LCIDRDT		
Type	I	I	I	I	I	I		

Card 5

Variable	LCIDTER	LCIDTES	LCIDTET	LCIDRER	LCIDRES	LCIDRET		
Type	I	I	I	I	I	I		

Card 6

Variable	UTFAILR	UTFAILS	UTFAILT	WTFAILR	WTFAILS	WTFAILT		
Type	F	F	F	F	F	F		

Card 7

Variable	UCFAILR	UCFAILS	UCFAILT	WCFAILR	WCFAILS	WCFAILT		
Type	F	F	F	F	F	F		

Card 8

Variable	IUR	IUS	IUT	IWR	IWS	IWT		
Type	F	F	F	F	F	F		

VARIABLE

DESCRIPTION

- MID Material identification. A unique number has to be chosen.
- RO Mass density, see also volume in *SECTION_BEAM definition.
- KT Translational stiffness for unloading option 2.0.
- KR Rotational stiffness for unloading option 2.0.

VARIABLE	DESCRIPTION
DAMPF	Damping factor for stability. Values in the neighborhood of unity are recommended. This damping factor is properly scaled to eliminate time step size dependency. Also, it is active if and only if the local stiffness is defined.
UNLDOPT	Unloading option (Also see Figure 20.34): EQ.0.0: Loading and unloading follow loading curve EQ.1.0: Loading follows loading curve, unloading follows unloading curve. The unloading curve ID if undefined is taken as the loading curve. EQ.2.0: Loading follows loading curve, unloading follows unloading stiffness, KT or KR, to the unloading curve. The loading and unloading curves may only intersect at the origin of the axes. EQ.3.0: Quadratic unloading from peak displacement value to a permanent offset.
OFFSET	Offset factor between 0 and 1.0 to determine permanent set upon unloading if the UNLDOPT=3.0. The permanent sets in compression and tension are equal to the product of this offset value and the maximum compressive and tensile displacements, respectively.
LCIDTR	Load curve ID defining translational force resultant along local r-axis versus relative translational displacement. If zero, no stiffness related forces are generated for this degree of freedom. The loading curves must be defined from the most negative displacement to the most positive displacement. The force does not need to increase monotonically. The curves in this input are linearly extrapolated when the displacement range falls outside the curve definition.
LCIDTS	Load curve ID defining translational force resultant along local s-axis versus relative translational displacement.
LCIDTT	Load curve ID defining translational force resultant along local t-axis versus relative translational displacement.
LCIDRR	Load curve ID defining rotational moment resultant about local r-axis versus relative rotational displacement.
LCIDRS	Load curve ID defining rotational moment resultant about local s-axis versus relative rotational displacement.
LCIDRT	Load curve ID defining rotational moment resultant about local t-axis versus relative rotational displacement.

VARIABLE	DESCRIPTION
LCIDTUR	Load curve ID defining translational force resultant along local r-axis versus relative translational displacement during unloading. The force values defined by this curve must increase monotonically from the most negative displacement to the most positive displacement. For UNLDOPT=1.0, the slope of this curve must equal or exceed the loading curve for stability reasons. This is not the case for UNLDOPT=2.0. For loading and unloading to follow the same path simply set LCIDTUR=LCIDTR. For options UNLDOPT=0.0 or 3.0 the unloading curve is not required.
LCIDTUS	Load curve ID defining translational force resultant along local s-axis versus relative translational displacement during unloading.
LCIDTUT	Load curve ID defining translational force resultant along local t-axis versus relative translational displacement during unloading.
LCIDRUR	Load curve ID defining rotational moment resultant about local r-axis versus relative rotational displacement during unloading.
LCIDRUS	Load curve ID defining rotational moment resultant about local s-axis versus relative rotational displacement. during unloading.
LCIDRUT	Load curve ID defining rotational moment resultant about local t-axis versus relative rotational displacement during unloading. If zero, no viscous forces are generated for this degree of freedom.
LCIDTDR	Load curve ID defining translational damping force resultant along local r-axis versus relative translational velocity.
LCIDTDS	Load curve ID defining translational damping force resultant along local s-axis versus relative translational velocity.
LCIDTDT	Load curve ID defining translational damping force resultant along local t-axis versus relative translational velocity.
LCIDRDR	Load curve ID defining rotational damping moment resultant about local r-axis versus relative rotational velocity.
LCIDRDS	Load curve ID defining rotational damping moment resultant about local s-axis versus relative rotational velocity.
LCIDRDT	Load curve ID defining rotational damping moment resultant about local t-axis versus relative rotational velocity.
LCIDTER	Load curve ID defining translational damping force scale factor versus relative displacement in local r-direction.
LCIDTES	Load curve ID defining translational damping force scale factor versus relative displacement in local s-direction.
LCIDTET	Load curve ID defining translational damping force scale factor versus relative displacement in local t-direction.

VARIABLE	DESCRIPTION
LCIDRER	Load curve ID defining rotational damping moment resultant scale factor versus relative displacement in local r-rotation.
LCIDRES	Load curve ID defining rotational damping moment resultant scale factor versus relative displacement in local s-rotation.
LCIDRET	Load curve ID defining rotational damping moment resultant scale factor versus relative displacement in local t-rotation.
UTFAILR	Optional, translational displacement at failure in tension. If zero, the corresponding displacement, u_r , is not considered in the failure calculation.
UTFAILS	Optional, translational displacement at failure in tension. If zero, the corresponding displacement, u_s , is not considered in the failure calculation.
UTFAILT	Optional, translational displacement at failure in tension. If zero, the corresponding displacement, u_t , is not considered in the failure calculation.
WTFAILR	Optional, rotational displacement at failure in tension. If zero, the corresponding rotation, θ_r , is not considered in the failure calculation.
WTFAILS	Optional, rotational displacement at failure in tension. If zero, the corresponding rotation, θ_s , is not considered in the failure calculation.
WTFAILT	Optional rotational displacement at failure in tension. If zero, the corresponding rotation, θ_t , is not considered in the failure calculation.
UCFAILR	Optional, translational displacement at failure in compression. If zero, the corresponding displacement, u_r , is not considered in the failure calculation. Define as a positive number.
UCFAILS	Optional, translational displacement at failure in compression. If zero, the corresponding displacement, u_s , is not considered in the failure calculation. Define as a positive number.
UCFAILT	Optional, translational displacement at failure in compression. If zero, the corresponding displacement, u_t , is not considered in the failure calculation. Define as a positive number.
WCFAILR	Optional, rotational displacement at failure in compression. If zero, the corresponding rotation, θ_r , is not considered in the failure calculation. Define as a positive number.
WCFAILS	Optional, rotational displacement at failure in compression. If zero, the corresponding rotation, θ_s , is not considered in the failure calculation. Define as a positive number.
WCFAILT	Optional rotational displacement at failure in compression. If zero, the corresponding rotation, θ_t , is not considered in the failure calculation. Define as a positive number.

<u>VARIABLE</u>	<u>DESCRIPTION</u>
WCFAILT	Optional rotational displacement at failure in compression. If zero, the corresponding rotation, θ_t , is not considered in the failure calculation. Define as a positive number.
IUR	Initial translational displacement along local r-axis.
IUS	Initial translational displacement along local s-axis.
IUT	Initial translational displacement along local t-axis.
WCFAILT	Optional rotational displacement at failure in compression. If zero, the corresponding rotation, θ_t , is not considered in the failure calculation. Define as a positive number.
IUR	Initial translational displacement along local r-axis.
IUS	Initial translational displacement along local s-axis.
IUT	Initial translational displacement along local t-axis.
IWR	Initial rotational displacement about the local r-axis.
IWS	Initial rotational displacement about the local s-axis.
IWT	Initial rotational displacement about the local t-axis.

Remarks:

Catastrophic failure, which is based on displacement resultants, occurs if either of the following inequalities are satisfied:

$$\left(\frac{u_r}{u_r^{tfail}}\right)^2 + \left(\frac{u_s}{u_s^{tfail}}\right)^2 + \left(\frac{u_t}{u_t^{tfail}}\right)^2 + \left(\frac{\theta_r}{\theta_r^{tfail}}\right)^2 + \left(\frac{\theta_s}{\theta_s^{tfail}}\right)^2 + \left(\frac{\theta_t}{\theta_t^{tfail}}\right)^2 - 1. \geq 0.$$

$$\left(\frac{u_r}{u_r^{cfail}}\right)^2 + \left(\frac{u_s}{u_s^{cfail}}\right)^2 + \left(\frac{u_t}{u_t^{cfail}}\right)^2 + \left(\frac{\theta_r}{\theta_r^{cfail}}\right)^2 + \left(\frac{\theta_s}{\theta_s^{cfail}}\right)^2 + \left(\frac{\theta_t}{\theta_t^{cfail}}\right)^2 - 1. \geq 0.$$

After failure the discrete element is deleted. If failure is included either the tension failure or the compression failure or both may be used.

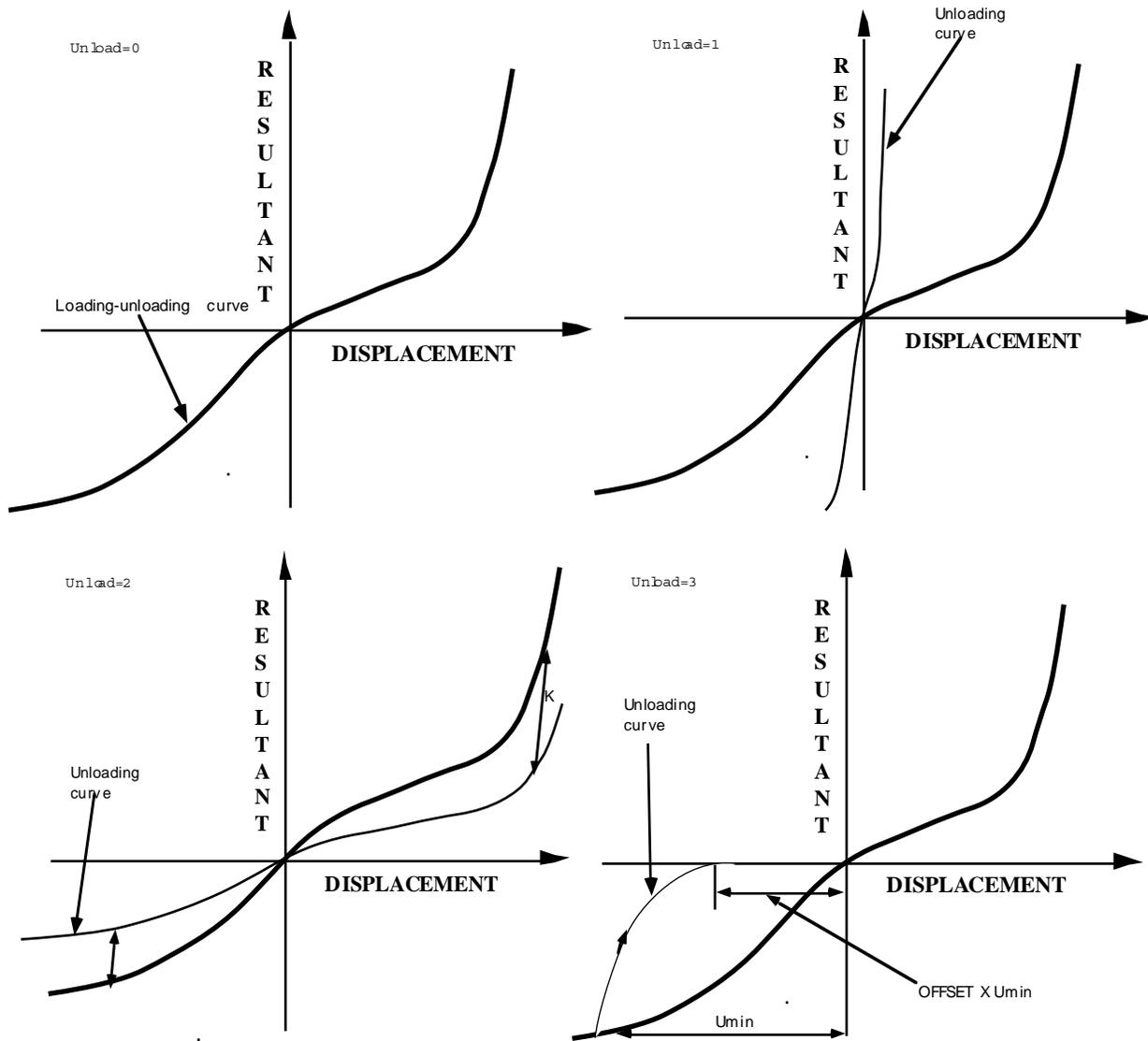


Figure 20.34. Load and unloading behavior.

*MAT_GURSON

This is Material Type 120. This is the Gurson dilational-plastic model. This model is currently available for shell elements only. A detailed description of this model can be found in the following references: Gurson [1975,1977]; Chu and Needleman[1980]; and Tvergaard and Needleman[1984]. The implementation in LS-DYNA is based on the implementation of Feucht [1998] and Faßnacht [1999], which was recoded at LSTC.

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	MID	RO	E	PR	SIGY	N	Q1	Q2
Type	I	F	F	F	F	F	F	F
Default	none	none	none	none	none	0.0	none	none

Card 2

Variable	FC	F0	EN	SN	FN	ETAN	ATYP	FF0
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

Card 3

Variable	EPS1	EPS2	EPS3	EPS4	EPS5	EPS6	EPS7	EPS8
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

Card 4

Variable	ES1	ES2	ES3	ES4	ES5	ES6	ES7	ES8
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

Card 5

Variable	L1	L2	L3	L4	FF1	FF2	FF3	FF4
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

Card 6

Variable	LCSS	LCLF	NUMINT					
Type	F	F	F					
Default	0	0	1					

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
E	Young's modulus.

VARIABLE	DESCRIPTION
PR	Poisson's ratio.
SIGY	Yield stress.
N	Exponent for Power law. This value is only used if ATYP=1 and LCSS=0.
Q1	Parameter q_1 .
Q1	Parameter q_2 .
FC	Critical void volume fraction f_c
F0	Initial void volume fraction f_0 .
EN	Mean nucleation strain ε_N .
SN	Standard deviation S_N of the normal distribution of ε_N .
FN	Void volume fraction of nucleating particles.
ETAN	Hardening modulus. This value is only used if ATYP=2 and LCSS=0.
ATYP	Type of hardening. EQ.1.0 Power law. EQ.2.0: Linear hardening. EQ.3.0: 8 points curve.
FF0	Failure void volume fraction. This value is used if no curve is given by the points L1,FF1 - L4,FF4 and LCLF=0.
EPS1-EPS8	Effective plastic strain values. The first point must be zero corresponding to the initial yield stress. This option is only used if ATYP equal to 3. At least 2 points should be defined. These values are used if ATYP=3 and LCSS=0.
ES1-ES8	Corresponding yield stress values to EPS1 - EPS8. These values are used if ATYP=3 and LCSS=0.
L1-L4	Element length values. These values are only used if LCLF=0.
FF1-FF4	Corresponding failure void volume fraction. These values are only used if LCLF=0.
LCSS	Load curve ID defining effective stress versus effective plastic strain. ATYP is ignored with this option.

VARIABLE	DESCRIPTION
LCLF	Load curve ID defining failure void volume fraction versus element length. the values L1-L4 and FF1-FF4 are ignored with this option.
NUMINT	Number of through thickness integration points which must fail before the element is deleted.

Remarks:

The Gurson flow function is defined as:

$$\Phi = \frac{\sigma_M^2}{\sigma_Y^2} + 2q_1 f^* \cosh\left(\frac{3q_2 \sigma_H}{2\sigma_M}\right) - 1 - (q_1 f^*)^2 = 0$$

where σ_M is the equivalent von Mises stress, σ_Y is the Yield stress, σ_H is the mean hydrostatic stress. The effective void volume fraction is defined as

$$f^*(f) = \begin{cases} f & f \leq f_c \\ f_c + \frac{1/q_1 - f_c}{f_F - f_c} (f - f_c) & f > f_c \end{cases}$$

The grow of void volume fraction is defined as

$$\dot{f} = \dot{f}_G + \dot{f}_N$$

where the grow of existing voids is defined as

$$\dot{f}_G = (1 - f) \dot{\epsilon}_{kk}^p$$

and nucleation of new voids is defined as

$$\dot{f}_N = A \dot{\epsilon}_p$$

where is defined as

$$A = \frac{f_N}{S_N \sqrt{2\pi}} \exp\left(-\frac{1}{2} \left(\frac{\epsilon_p - \epsilon_N}{S_N}\right)^2\right)$$

*MAT_GURSON_RCDC

This is an enhancement of material Type 120. This is the Gurson model with the Wilkins Rc-Dc added This model is currently available for shell elements only. A detailed description of this model can be found in the following references: Gurson [1975,1977]; Chu and Needleman[1980]; and Tvergaard and Needleman[1984].

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	MID	RO	E	PR	SIGY	N	Q1	Q2
Type	I	F	F	F	F	F	F	F
Default	none	none	none	none	none	0.0	none	none

Card 2

Variable	FC	F0	EN	SN	FN	ETAN	ATYP	FF0
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

Card 3

Variable	EPS1	EPS2	EPS3	EPS4	EPS5	EPS6	EPS7	EPS8
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

Card 4

Variable	ES1	ES2	ES3	ES4	ES5	ES6	ES7	ES8
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

Card 5

Variable	L1	L2	L3	L4	FF1	FF2	FF3	FF4
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

Card 6

Variable	LCSS	LCLF	NUMINT					
Type	F	F	F					
Default	0	0	1					

Card 7

Variable	ALPHA	BETA	GAMMA	D0	B	LAMBDA	DS	L
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

<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.
N	Exponent for Power law.This value is only used if ATYP=1 and LCSS=0.
Q1	Parameter q_1 .
Q1	Parameter q_2 .
FC	Critical void volume fraction f_c
F0	Initial void volume fraction f_0 .
EN	Mean nucleation strain ε_N .
SN	Standard deviation S_N of the normal distribution of ε_N .
FN	Void volume fraction of nucleating particles.
ETAN	Hardening modulus. This value is only used if ATYP=2 and LCSS=0.
ATYP	Type of hardening. EQ.1.0 Power law. EQ.2.0: Linear hardening. EQ.3.0: 8 points curve.
FF0	Failure void volume fraction. This value is used if noe curve is given by the points L1,FF1 - L4,FF4 and LCLF=0.
EPS1-EPS8	Effective plastic strain values.The first point must be zero corresponding to the initial yield stress. This option is only used if ATYP equal to 3. At least 2 points should be defined.These values are used if ATYP=3 and LCSS=0.
ES1-ES8	Corresponding yield stress values to EPS1 - EPS8. These values are used if ATYP=3 and LCSS=0.
L1-L4	Element length values.These values are only used if LCLF=0.
FF1-FF4	Corresponding failure void volume fraction. These values are only used if LCLF=0.

VARIABLE	DESCRIPTION
LCSS	Load curve ID defining effective stress versus effective plastic strain. ATYP is ignored with this option.
LCLF	Load curve ID defining failure void volume fraction versus element length. the values L1-L4 and FF1-FF4 are ignored with this option.
NUMINT	Number of through thickness integration points which must fail before the element is deleted.
ALPHA	Parameter α . for the Rc-Dc model
BETA	Parameter β . for the Rc-Dc model
GAMMA	Parameter γ . for the Rc-Dc model
D0	Parameter D_0 . for the Rc-Dc model
B	Parameter b . for the Rc-Dc model
LAMBDA	Parameter λ . for the Rc-Dc model
DS	Parameter D_s . for the Rc-Dc model
L	Characteristic element length for this material

Remarks:

The Gurson flow function is defined as:

$$\Phi = \frac{\sigma_M^2}{\sigma_Y^2} + 2q_1 f^* \cosh\left(\frac{3q_2 \sigma_H}{2\sigma_M}\right) - 1 - (q_1 f^*)^2 = 0$$

where σ_M is the equivalent von Mises stress, σ_Y is the Yield stress, σ_H is the mean hydrostatic stress. The effective void volume fraction is defined as

$$f^*(f) = \begin{cases} f & f \leq f_c \\ f_c + \frac{1/q_1 - f_c}{f_F - f_c} (f - f_c) & f > f_c \end{cases}$$

The grow of void volume fraction is defined as

$$\dot{f} = \dot{f}_G + \dot{f}_N$$

where the grow of existing voids is defined as

$$\dot{f}_G = (1 - f) \dot{\epsilon}_{kk}^p$$

and nucleation of new voids is defined as

$$\dot{f}_N = A \dot{\epsilon}_p$$

where A is defined as

$$A = \frac{f_N}{S_N \sqrt{2\pi}} \exp\left(-\frac{1}{2} \left(\frac{\epsilon_p - \epsilon_N}{S_N}\right)^2\right)$$

The Rc-Dc model is defined as the following:

The damage D is given by

$$D = \int \omega_1 \omega_2 d\epsilon^p$$

where ϵ^p is the equivalent plastic strain,

$$\omega_1 = \left(\frac{1}{1 - \gamma \sigma_m}\right)^\alpha$$

is a triaxial stress weighting term and

$$\omega_2 = (2 - A_D)^\beta$$

is a asymmetric strain weighting term.

In the above σ_m is the mean stress and

$$A_D = \min\left(\left|\frac{S_2}{S_3}\right|, \left|\frac{S_3}{S_2}\right|\right)$$

Fracture is initiated when the accumulation of damage is

$$\frac{D}{D_c} > 1$$

where D_c is the a critical damage given by

$$D_c = D_0 (1 + b |\nabla D|^\lambda)$$

A fracture fraction

$$F = \frac{D - D_c}{D_s}$$

defines the degradations of the material by the Rc-Dc model.

The characteristic element length is used in the calculation of ∇D . Calculation of this factor is only done for element with smaller element length than this value.

***MAT_GENERAL_NONLINEAR_1DOF_DISCRETE_BEAM**

This is Material Type 121. This is a very general spring and damper model. This beam is based on the MAT_SPRING_GENERAL_NONLINEAR option and is a one dimensional version of the 6DOF_DISCRETE_BEAM above. Additional unloading options have been included.

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	MID	RO	K	UNLDOPT	OFFSET	DAMPF		
Type	I	F	F	I	F	F		

Card 2

Variable	LCIDT	LCIDTU	LCIDTD	LCIDTE				
Type	I	I	I	I				

Card 3

Variable	UTFAIL	UCFAIL	IU					
Type	F	F	F					

<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 definition.
K	Translational stiffness for unloading option 2.0.
KR	Rotational stiffness for unloading option 2.0.
DAMPF	Damping factor for stability. Values in the neighborhood of unity are recommended. This damping factor is properly scaled to eliminate time step size dependency. Also, it is active if and only if the local stiffness is defined.

VARIABLE	DESCRIPTION
UNLDOPT	Unloading option (Also see Figure 20.35): EQ.0.0: Loading and unloading follow loading curve EQ.1.0: Loading follows loading curve, unloading follows loading curve. Also see Figure 20.35. The unloading curve ID if defined is ignored. EQ.2.0: Loading follows loading curve, unloading follows unloading stiffness, K, to the unloading curve. The loading and unloading curves intersect at the origin of the axes. EQ.3.0: Quadratic unloading from peak displacement value to permanent set.
OFFSET	Offset to determine permanent set upon unloading if the UNLDOPT=3.0. The permanent sets in compression and tension are equal to the product of this offset value and the maximum compressive and tensile displacements, respectively.
LCIDT	Load curve ID defining translational force resultant along the axis versus relative translational displacement. If zero, no stiffness related forces are generated for this degree of freedom. The loading curves must be defined from the most negative displacement to the most positive displacement. The force does not need to increase monotonically for the loading curve. The curves in this input are extrapolated when the displacement range falls outside the curve definition.
LCIDTU	Load curve ID defining translational force resultant along the axis versus relative translational displacement during unloading. The force values defined by this curve must increase monotonically from the most negative displacement to the most positive displacement. For UNLDOPT=1.0, the slope of this curve must equal or exceed the loading curve for stability reasons. This is not the case for UNLDOPT=2.0. For loading and unloading to follow the same path simply set LCIDTU=LCIDT.
LCIDTD	Load curve ID defining translational damping force resultant along local the axis versus relative translational velocity.
LCIDTE	Load curve ID defining translational damping force scale factor versus relative displacement in along axis.
UTFAIL	Optional, translational displacement at failure in tension. If zero, failure in tension is not considered.
UCFAIL	Optional, translational displacement at failure in compression. If zero, failure in compression is not considered.
IU	Initial translational displacement along axis.

*MAT_HILL_3R

This is Material Type 122. This is Hill's 1948 planar anisotropic material model with 3 R values.

Card Format

Card 1 1 2 3 4 5 6 7

Variable	MID	RO	E	PR	HR	P1	P2	
Type	I	F	F	F	F	F	F	

Card 2

Variable	R00	R45	R90	LCID	E0			
Type	F	F	F	F	F			

Card 3

Variable	AOPT							
Type	F							

Card 4

Variable				A1	A2	A3		
Type				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.
E	Young's modulus, E
PR	Poisson's ratio, ν
HR	Hardening rule: EQ.1.0: linear (default), EQ.2.0: exponential. EQ.3.0: load curve
P1	Material parameter: HR.EQ.1.0: Tangent modulus, HR.EQ.2.0: k, strength coefficient for exponential hardening
P2	Material parameter: HR.EQ.1.0: Yield stress HR.EQ.2.0: n, exponent
R00	R ₀₀ , Lankford parameter determined from experiments
R45	R ₄₅ , Lankford parameter determined from experiments
R90	R ₉₀ , Lankford parameter determined from experiments
LCID	load curve ID for the load curve hardening rule
E0	ϵ_0 for determining initial yield stress for exponential hardening. (Default=0.0)

VARIABLE	DESCRIPTION
AOPT	Material axes option (see MAT_OPTION TROPIC_ELASTIC for a more complete description): EQ. 0.0: locally orthotropic with material axes determined by element nodes 1, 2, and 4, as with *DEFINE_COORDINATE_NODES. EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR. EQ. 3.0: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle, BETA, from a line in the plane of the element defined by the cross product of the vector v with the element normal.
XP YP ZP	Coordinates of point p for AOPT = 1.
A1 A2 A3	Components of vector a for AOPT = 2.
V1 V2 V3	Components of vector v for AOPT = 3.
D1 D2 D3	Components of vector d for AOPT = 2.
BETA	Material angle in degrees for AOPT = 3, may be overridden on the element card, see *ELEMENT_SHELL_BETA.

*MAT_123

*MAT_MODIFIED_PIECEWISE_LINEAR_PLASTICITY

*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

<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; the curve ID, LCSR; EPS1-EPS8 and ES1-ES8 are ignored if a Table ID is defined.

<u>VARIABLE</u>	<u>DESCRIPTION</u>
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. 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.

*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						

VARIABLE	DESCRIPTION
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 aluminum honeycomb crushable foam materials with anisotropic behavior. Two yield surfaces are available. In the first, nonlinear elastoplastic material behavior can be defined separately for all normal and shear stresses, which are considered to be fully uncoupled. In the second, which will be available in June 2003 (the first updated release of version 970), a yield surface is defined that considers the effects of off axis loading. The second yield surface is transversely anisotropic. The choice of yield surfaces is flagged by the sign of the first load curve ID, LCA. The development of the second yield surface is based on experimental test results of aluminum honeycomb specimens at Toyota Motor Corporation. The default element for this material is solid type 0, a nonlinear spring type brick element. *The recommended hourglass control is the type 2 viscous formulation for one point integrated solid elements. The stiffness form of the hourglass control when used with this constitutive model can lead to nonphysical results since strain localization in the shear modes can be inhibited.*

Card 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	VREF	TREF	
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 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. This parameter is ignored for corotational solid elements, types 0 and 9.
MU	μ , material viscosity coefficient. (default=.05) Recommended.
BULK	Bulk viscosity flag: EQ.0.0: bulk viscosity is not used. This is recommended. EQ.1.0: bulk viscosity is active and $\mu=0$ This will give results identical to previous versions of LS-DYNA.
LCA	Load curve ID, see *DEFINE_CURVE: LCA.LT.0: Yield stress as a function of the angle off the material axis in degrees. LCA.GT.0: sigma-aa versus normal strain component aa. For the corotational solid elements, types 0 and 9, engineering strain is expected, but for all other solid element formulations a logarithmic strain is expected. See notes below.

VARIABLE	DESCRIPTION
LCB	Load curve ID, see *DEFINE_CURVE: LCA.LT.0: strong axis stress as a function of the volumetric strain. The abscissa values must range between 0 to 90 degrees, inclusive. LCA.GT.0: sigma-bb versus normal strain component bb. For the corotational solid elements, types 0 and 9, engineering strain is expected, but for all other solid element formulations a logarithmic strain is expected. Default LCB=LCA. See notes below.
LCC	Load curve ID, see *DEFINE_CURVE: LCA.LT.0: weak axis stress as a function of the volumetric strain. LCA.GT.0: sigma-cc versus normal strain component cc. For the corotational solid elements, types 0 and 9, engineering strain is expected, but for all other solid element formulations a logarithmic strain is expected. Default LCC=LCA. See notes below.
LCS	Load curve ID, see *DEFINE_CURVE: LCA.LT.0: damage curve giving shear stress multiplier as a function of the shear strain component. This curve definition is optional and may be used if damage is desired. LCA.GT.0: shear stress versus shear strain. For the corotational solid elements, types 0 and 9, engineering strain is expected, but for all other solid element formulations a shear strain based on the deformed configuration is used.. Default LCS=LCA. Each component of shear stress may have its own load curve. See notes below.
LCAB	Load curve ID, see *DEFINE_CURVE. Default LCAB=LCS: LCA.LT.0: damage curve giving shear ab-stress multiplier as a function of the ab-shear strain component. This curve definition is optional and may be used if damage is desired. LCA.GT.0: sigma-ab versus shear strain-ab. For the corotational solid elements, types 0 and 9, engineering strain is expected, but for all other solid element formulations a shear strain based on the deformed configuration is used. See notes below.
LCBC	Load curve ID, see *DEFINE_CURVE. Default LCBC=LCS: LCA.LT.0: damage curve giving bc-shear stress multiplier as a function of the ab-shear strain component. This curve definition is optional and may be used if damage is desired. LCA.GT.0: sigma-bc versus shear strain-bc. For the corotational solid elements, types 0 and 9, engineering strain is expected, but for all other solid element formulations a shear strain based on the deformed configuration is used. See notes below.
LCCA	Load curve ID, see *DEFINE_CURVE. Default LCCA=LCS: LCA.LT.0: damage curve giving ca-shear stress multiplier as a function of the ca-shear strain component. This curve definition is optional and may be used if damage is desired. LCA.GT.0: sigma-ca versus shear strain-ca. For the corotational solid elements, types 0 and 9, engineering strain is expected, but for all other solid element formulations a shear strain based on the deformed configuration is used. See notes below.

VARIABLE	DESCRIPTION
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 (see MAT_OPTION TROPIC_ELASTIC for a more complete description): EQ. 0.0: locally orthotropic with material axes determined by element nodes 1, 2, and 4, as with *DEFINE_COORDINATE_NODES. EQ. 1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center; this is the a-direction. EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR.
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).
VREF	This is an optional input parameter for solid elements types 1, 2, 3, 4, and 10. Relative volume at which the reference geometry is stored. At this time the element behaves like a nonlinear spring. The TREF, below, is reached first then VREF will have no effect.
TREF	This is an optional input parameter for solid elements types 1, 2, 3, 4, and 10. Element time step size at which the reference geometry is stored. When this time step size is reached the element behaves like a nonlinear spring. If VREF, above, is reached first then TREF will have no effect.

Remarks:

For efficiency it is strongly recommended that the load curve ID's: LCA, LCB, LCC, LCS, LCAB, LCBC, and LCCA, contain exactly the same number of points with corresponding strain values on the abscissa. If this recommendation is followed the cost of the table lookup is

insignificant. Conversely, the cost increases significantly if the abscissa strain values are not consistent between load curves.

For solid element formulations 1 and 2, the behavior before compaction is orthotropic where the components of the stress tensor are uncoupled, i.e., an a component of strain will generate resistance in the local a -direction with no coupling to the local b and c directions. The elastic moduli vary from their initial values to the fully compacted values linearly with the relative volume:

$$E_{aa} = E_{aau} + \beta(E - E_{aau}) \quad G_{ab} = G_{abu} + \beta(G - G_{abu})$$

$$E_{bb} = E_{bbu} + \beta(E - E_{bbu}) \quad G_{bc} = G_{bcu} + \beta(G - G_{bcu})$$

$$E_{cc} = E_{ccu} + \beta(E - E_{ccu}) \quad G_{ca} = G_{cau} + \beta(G - G_{cau})$$

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.

For corotational solid elements, types 0 and 9, the components of the stress tensor remain uncoupled and the uncompressed elastic moduli are used, that is, the fully compacted elastic moduli are ignored.

The load curves define the magnitude of the stress as the material undergoes deformation. The first value in the curve should be less than or equal to zero corresponding to tension and increase to full compaction. **Care should be taken when defining the curves so the extrapolated values do not lead to negative yield stresses.**

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

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

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

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

If $LCA > 0$, each component of the updated stress tensor is checked to ensure that it does not exceed the permissible value determined from the load curves, e.g., if

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

then

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

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

If $LCA < 0$, a transversely anisotropic yield surface is obtained where the uniaxial limit stress, $\sigma^y(\varphi, \epsilon^{vol})$, can be defined as a function of angle φ with the strong axis and volumetric strain, ϵ^{vol} . In order to facilitate the input of data to such a limit stress surface, the limit stress is written as:

$$\sigma^y(\varphi, \epsilon^{vol}) = \sigma^b(\varphi) + (\cos \varphi)^2 \sigma^s(\epsilon^{vol}) + (\sin \varphi)^2 \sigma^w(\epsilon^{vol})$$

where the functions σ^b , σ^s , and σ^w are represented by load curves LCA, LCB, LCC, respectively. The latter two curves can be used to include the stiffening effects that are observed as the foam material crushes to the point where it begins to lock up. To ensure that the limit stress decreases with respect to the off-angle the curves should be defined such that following equations hold:

$$\frac{\partial \sigma^b(\varphi)}{\partial \varphi} \leq 0$$

and

$$\sigma^s(\epsilon^{vol}) - \sigma^w(\epsilon^{vol}) \geq 0.$$

For fully compacted material (element formulations 1 and 2), we assume that the material behavior is elastic-perfectly plastic and updated the stress components according to:

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

where the deviatoric strain increment is defined as

$$\Delta \epsilon_{ij}^{dev} = \Delta \epsilon_{ij} - \frac{1}{3} \Delta \epsilon_{kk} \delta_{ij}.$$

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

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

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

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

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

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.

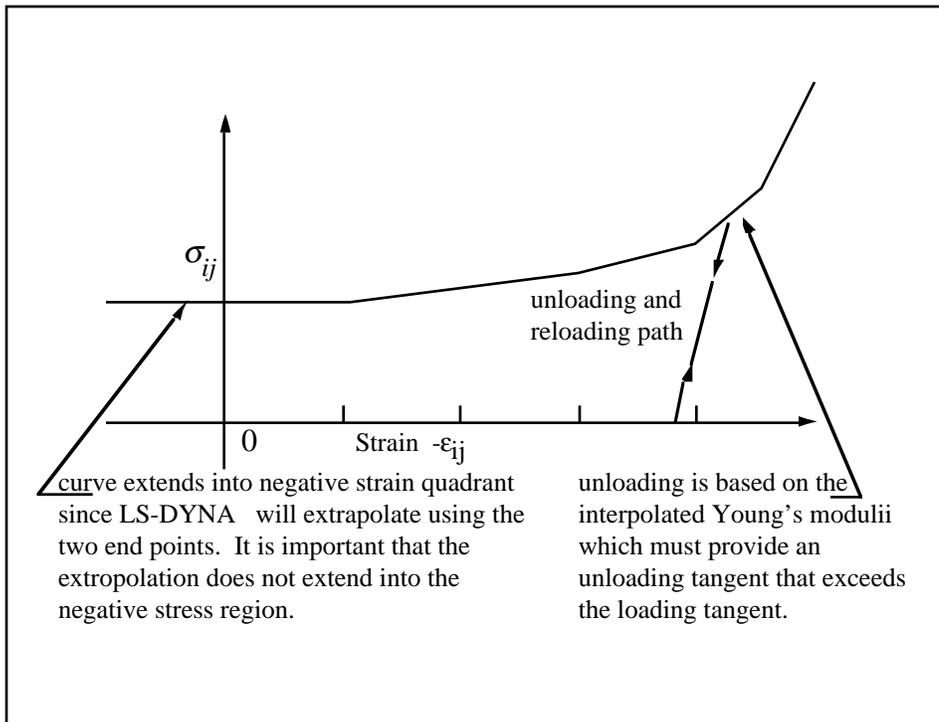


Figure 20.35 Stress quantity versus strain. Note that the “yield stress” at a strain of zero is nonzero. In the load curve definition the “time” value is the directional strain and the “function” value is the yield stress. Note that for element types 0 and 9 engineering strains are used, but for all other element types the rates are integrated in time.

*MAT_ARRUDA_BOYCE_RUBBER

This is Material Type 127. This material model provides a hyperelastic rubber model, see [Arruda and Boyce, 1993] combined optionally with linear viscoelasticity as outlined by [Christensen 1980].

Card 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						

VARIABLE	DESCRIPTION
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 βt are determined via a least squares fit. This relaxation curve is shown in Figure 20.25. This model ignores the constant stress.
TRAMP	Optional ramp time for loading.
NT	Number of Prony series terms in optional fit. If zero, the default is 6. Currently, the maximum number is 6. Values less than 6, possibly 3-5 are recommended, since each term used adds significantly to the cost. Caution should be exercised when taking the results from the fit. Always check the results of the fit in the output file. Preferably, all generated coefficients should be positive. Negative values may lead to unstable results. Once a satisfactory fit has been achieved it is recommended that the coefficients which are written into the output file be input in future runs.
GI	Optional shear relaxation modulus for the ith term
BETAI	Optional decay constant if ith term

Remarks:

Rubber is generally considered to be fully incompressible since the bulk modulus greatly exceeds the shear modulus in magnitude. To model the rubber as an unconstrained material a hydrostatic work term, $W_H(J)$, is included in the strain energy functional which is function of the relative volume, J , [Ogden, 1984]:

$$W(J_1, J_2, J) = nk\theta \left[\frac{1}{2}(J_1 - 3) + \frac{1}{20N}(J_1^2 - 9) + \frac{11}{1050N^2}(J_1^3 - 27) \right] \\ + nk\theta \left[\frac{19}{7000N^3}(J_1^4 - 81) + \frac{519}{673750N^4}(J_1^5 - 243) \right] + W_H(J) \\ J_1 = I_1 J^{-\frac{1}{3}} \\ J_2 = I_2 J$$

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, β_i . The viscoelastic behavior is optional and an arbitrary number of terms may be used.

***MAT_HEART_TISSUE**

This is Material Type 128. This material model provides a heart tissue model described in the paper by 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	

VARIABLE	DESCRIPTION
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 MAT_OPTION TROPIC_ELASTIC for a more complete description): EQ. 0.0: locally orthotropic with material axes determined by element nodes 1, 2, and 4, as with *DEFINE_COORDINATE_NODES. EQ. 1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center; this is the a-direction. EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR. EQ. 3.0: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle, BETA, from a line in the plane of the element defined by the cross product of the vector \mathbf{v} with the element normal. EQ. 4.0: locally orthotropic in cylindrical coordinate system with the material axes determined by a vector \mathbf{v} , and an originating point, P, which define the centerline axis.
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, may be overridden on the element card, see *ELEMENT_SOLID_ORTHO.

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	BETA1						
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	Δ , material coefficient.
ALPHA	α , material coefficient.
BETA	β , material coefficient.
C1	Material coefficient.
C2	Material coefficient.
LCID	Optional load curve ID of relaxation curve If constants βt are determined via a least squares fit. This relaxation curve is shown in Figure 20.25. 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:

$$\sigma_{ij} = \int_0^t g_{ijkl}(t-\tau) \frac{\partial \epsilon_{kl}}{\partial \tau} d\tau$$

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, β_i . The viscoelastic behavior is optional and an arbitrary number of terms may be used.

***MAT_SPECIAL_ORTHOTROPIC**

This is Material Type 130. This model is available the Belytschko-Tsay and the C0 triangular shell elements and is based on a resultant stress formulation. In-plane behavior is treated separately from bending in order to model perforated materials such as television shadow masks. If other shell formulations are specified, the formulation will be automatically switched to Belyschko-Tsay. As implemented, this material model cannot be used with user defined integration rules.

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				A1	A2	A3		
Type				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	v_{12p} , for in plane behavior.
V11P	v_{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	v_{12b} , for bending behavior.
V21B	v_{21b} , for bending behavior.
G12B	G_{12b} , for bending behavior.
AOPT	Material axes option (see <i>MAT_OPTION TROPIC_ELASTIC</i> for a more complete description): EQ. 0.0: locally orthotropic with material axes determined by element nodes 1, 2, and 4, as with *DEFINE_COORDINATE_NODES. EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR.

VARIABLE	DESCRIPTION
AOPT	EQ. 3.0: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle, BETA, from a line in the plane of the element defined by the cross product of the vector v with the element normal.
A1,A2,A3	a ₁ a ₂ a ₃ , define components of vector a for AOPT = 2.
D1,D2,D3	d ₁ d ₂ d ₃ , define components of vector d for AOPT = 2.
V1,V2,V3	v ₁ v ₂ v ₃ , define components of vector v for AOPT = 3.
BETA	Material angle in degrees for AOPT = 3, may be overridden on the element card, see *ELEMENT_SHELL_BETA.

Remarks:

The in-plane elastic matrix for in-plane, plane stress behavior is given by:

$$C_{in\ plane} = \begin{bmatrix} Q_{11p} & Q_{12p} & 0 & 0 & 0 \\ Q_{12p} & Q_{22p} & 0 & 0 & 0 \\ 0 & 0 & Q_{44p} & 0 & 0 \\ 0 & 0 & 0 & Q_{55p} & 0 \\ 0 & 0 & 0 & 0 & Q_{66p} \end{bmatrix}$$

The terms Q_{ijp} are defined as:

$$Q_{11p} = \frac{E_{11p}}{1 - \nu_{12p}\nu_{21p}}$$

$$Q_{22p} = \frac{E_{22p}}{1 - \nu_{12p}\nu_{21p}}$$

$$Q_{12p} = \frac{\nu_{12p}E_{11p}}{1 - \nu_{12p}\nu_{21p}}$$

$$Q_{44p} = G_{12p}$$

$$Q_{55p} = G_{23p}$$

$$Q_{66p} = G_{31p}$$

The elastic matrix for bending behavior is given by:

$$C_{bending} = \begin{bmatrix} Q_{11b} & Q_{12b} & 0 \\ Q_{12b} & Q_{22b} & 0 \\ 0 & 0 & Q_{44b} \end{bmatrix}$$

The terms Q_{ijp} are similarly defined.

*MAT_MODIFIED_FORCE_LIMITED

This is Material Type 139. This material for the Belytschko-Schwer resultant beam is an extension of material 29. In addition to the original plastic hinge and collapse mechanisms of material 29, yield moments may be defined as a function of axial force. After a hinge forms, the moment transmitted by the hinge is limited by a moment-plastic rotation relationship.

Card 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					

Card 7

Variable	LYS1	SYS1	LYS2	SYS2	LYT1	SYT1	LYT2	SYT2
Type	F	F	F	F	F	F	F	F
Default	0	1.0	0	1.0	0	1.0	0	1.0

Card 8

Variable	LYR	SYR						
Type	F	F						
Default	0	1.0						

Card 9

Variable	HMS1_1	HMS1_2	HMS1_3	HMS1_4	HMS1_5	HMS1_6	HMS1_7	HMS1_8
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

Card 10

Variable	LPMS1_1	LPMS1_2	LPMS1_3	LPMS1_4	LPMS1_5	LPMS1_6	LPMS1_7	LPMS1_8
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

Card 11

Variable	HMS2_1	HMS2_2	HMS2_3	HMS2_4	HMS2_5	HMS2_6	HMS2_7	HMS2_8
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

Card 12

Variable	LPMS2_1	LPMS2_2	LPMS2_3	LPMS2_4	LPMS2_5	LPMS2_6	LPMS2_7	LPMS2_8
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

Card 13

Variable	HMT1_1	HMT1_2	HMT1_3	HMT1_4	HMT1_5	HMT1_6	HMT1_7	HMT1_8
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

Card 14

Variable	LPMT1_1	LPMT1_2	LPMT1_3	LPMT1_4	LPMT1_5	LPMT1_6	LPMT1_7	LPMT1_8
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

Card 15

Variable	HMT2_1	HMT2_2	HMT2_3	HMT2_4	HMT2_5	HMT2_6	HMT2_7	HMT2_8
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

Card 16

Variable	LPMT2_1	LPMT2_2	LPMT2_3	LPMT2_4	LPMT2_5	LPMT2_6	LPMT2_7	LPMT2_8
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

Card 17

Variable	HMR_1	HMR_2	HMR_3	HMR_4	HMR_5	HMR_6	HMR_7	HMR_8
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

Card 18

Variable	LPMR_1	LPMR_2	LPMR_3	LPMR_4	LPMR_5	LPMR_6	LPMR_7	LPMR_8
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

VARIABLE

DESCRIPTION

MID	Material identification. A unique number 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!

VARIABLE	DESCRIPTION
AOPT	Axial load curve option: EQ.0.0: axial load curves are force versus strain, EQ.1.0: axial load curves are force versus change in length .
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)
LYS1	ID of curve defining yield moment as a function of axial force for the s-axis at node 1.
SYS1	Scale factor applied to load curve LYS1.
LYS2	ID of curve defining yield moment as a function of axial force for the s-axis at node 2.
SYS2	Scale factor applied to load curve LYS2.
LYT1	ID of curve defining yield moment as a function of axial force for the t-axis at node 1.
SYT1	Scale factor applied to load curve LYT1.
LYT2	ID of curve defining yield moment as a function of axial force for the t-axis at node 2.
SYT2	Scale factor applied to load curve LYT2.
LYR	ID of curve defining yield moment as a function of axial force for the torsional axis.
SYR	Scale factor applied to load curve LYR.
HMS1_n	Hinge moment for s-axis at node 1.
LPMS1_n	ID of curve defining plastic moment as a function of plastic rotation for the s-axis at node 1 for hinge moment HMS1_n
HMS2_n	Hinge moment for s-axis at node 2.
LPMS2_n	ID of curve defining plastic moment as a function of plastic rotation for the s-axis at node 2 for hinge moment HMS2_n
HMT1_n	Hinge moment for t-axis at node 1.

VARIABLE	DESCRIPTION
LPMT1_n	ID of curve defining plastic moment as a function of plastic rotation for the t-axis at node 1 for hinge moment HMT1_n
HMT2_n	Hinge moment for t-axis at node 2.
LPMT2_n	ID of curve defining plastic moment as a function of plastic rotation for the t-axis at node 2 for hinge moment HMT2_n
HMR_n	Hinge moment for the torsional axis.
LPMR_n	ID of curve defining plastic moment as a function of plastic rotation for the torsional axis for hinge moment HMR_n

Remarks:

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

Axial collapse occurs when the compressive axial load reaches the collapse load. Collapse load versus collapse deflection is specified in the form of a load curve. The points of the load curve are either (true strain, collapse force) or (change in length, collapse force). Both quantities should be entered as positive for all points, and will be interpreted as compressive. The first point should be (zero, initial collapse load).

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

Stiffness-proportional damping may be added using the damping factor λ . This is defined as follows:

$$\lambda = \frac{2*\xi}{\omega}$$

where ξ is the damping factor at the reference frequency ω (in radians per second). For example if 1% damping at 2Hz is required

$$\lambda = \frac{2*0.01}{2\pi*2} = 0.001592$$

If damping is used, a small 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 the current axial load, but it is possible to make hinge formation a function of axial load and subsequent plastic moment a function of the moment at the time the hinge formed. This is discussed in the next section.

Independent plastic hinge formation:

In addition to the moment interaction equation, Cards 7 through 18 allow plastic hinges to form independently for the s-axis and t-axis at each end of the beam and also for the torsional axis. A plastic hinge is assumed to form if any component of the current moment exceeds the yield moment as defined by the yield moment vs. axial force curves input on cards 7 and 8. If any of the 5 curves is omitted, a hinge will not form for that component. The curves can be defined for both compressive and tensile axial forces. If the axial force falls outside the range of the curve, the first or last point in the curve will be used. A hinge forming for one component of moment does not effect the other components.

Upon forming a hinge, the magnitude of that component of moment will not be permitted to exceed the current plastic moment.. The current plastic moment is obtained by interpolating between the plastic moment vs. plastic rotation curves input on cards 10, 12, 14, 16, or 18. Curves may be input for up to 8 hinge moments, where the hinge moment is defined as the yield moment at the time that the hinge formed. Curves must be input in order of increasing hinge moment and each curve should have the same plastic rotation values. The first or last curve will be used if the hinge moment falls outside the range of the curves. If no curves are defined, the plastic moment is obtain from the curves on cards 4 through 6. The plastic moment is scaled by the scale factors on lines 4 to 6.

A hinge will form if either the independent yield moment is exceeded or if the moment interaction equation is satisfied. If both are true, the plastic moment will be set to the minimum of the interpolated value and M_{rp} .

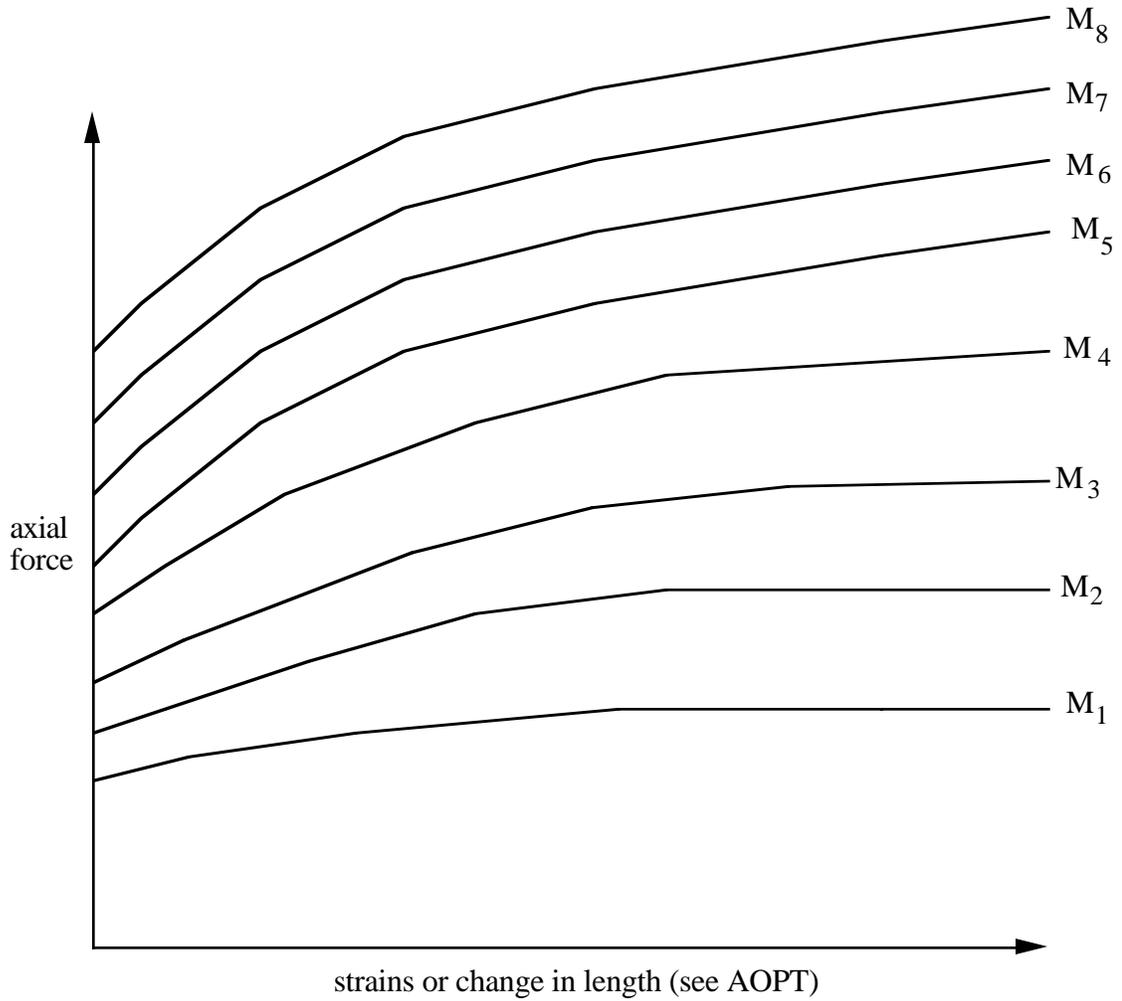


Figure 20.12. The force magnitude is limited by the applied end moment. For an intermediate value of the end moment LS-DYNA interpolates between the curves to determine the allowable force value.

*MAT_RATE_SENSITIVE_POLYMER

This is Material Type 141. This model is for the simulation of an isotropic ductile ploymer with strain rate effects, [Stouffer and Dame 1996]. Uniaxial test data has to be used.

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	MID	RO	E	PR	Do	N	Zo	q
Type	I	F	F	F	F	F	F	F

Card 2 1 2 3 4 5 6 7 8

Variable	Omega							
Type	F							

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MID	Material identification. A unique number has to be chosen.
RO	Mass density
E	Elastic modulus.
G	Shear modulus
PR	Poisson's ratio
Do	Refe
	rence strain rate (=1000*max strain rate used in the test).
N	Exponent (see inelastic strain rate equation below)
Zo	Initial hardness of material
q	(see equations below).
Omega	Maximum internal stress.

$$\varepsilon_{ij} = D_o \exp \left[-0.5 \left(\frac{Z_o^2}{3K_2} \right) \right] \left(\frac{S_{ij} - \Omega_{ij}}{\sqrt{K_2}} \right)$$

where the K_2 term is defined as follows:

$$K_2 = 0.5(S_{ij} - \Omega_{ij})(S_{ij} - \Omega_{ij})$$

and represent the second invariant of the overstress tensor. The elastic components of the strain are added to the inelastic strain to obtain the total strain. The following relationship defines the internal stress variable rate:

$$\dot{\Omega}_{ij} = \frac{2}{3} q \Omega_m \dot{\varepsilon}_{ij}^I - q \Omega_{ij} \dot{\varepsilon}_e^I$$

where q is a material constant, Ω_m is a material constant that represents the maximum value of the internal stress, and $\dot{\varepsilon}_e^I$ is the effective inelastic strain. Look into the theoretical manual for the procedure of calculation of the above parameters using series of tension tests under different strain rate.

***MAT_TRANSVERSELY_ANISOTROPIC_CRUSHABLE_FOAM**

This is Material Type 142. This model is for an extruded foam material that is transversely anisotropic, crushable, and of low density with no significant Poisson effect. This material is used in energy-absorbing structures to enhance automotive safety in low velocity (bumper impact) and medium high velocity (interior head impact and pedestrian safety) applications. The formulation of this foam is due to Hirth, Du Bois, and Weimar and is documented by Du Bois [2001]. This model behaves in a more physical way for off axis loading the material, *MAT_HONEYCOMB, which can exhibit nonphysical stiffening for loading conditions that are off axis. The load curves are used to define a yield surface that bounds the deviatoric stress tensor.

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	MID	RO	E11	E22	E12	E23	G	K
Type	I	F	F	F	F	F	F	

Card 2 1 2 3 4 5 6 7 8

Variable	I11	I22	I12	I23	IAA	NY	ANG	MU
Type	I	I	I	I	I	I	F	F

Card 3 1 2 3 4 5 6 7 8

Variable	AOPT							
Type	F							

Card 4

Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

***MAT_142 *MAT_TRANSVERSELY_ANISOTROPIC_CRUSHABLE_FOAM**

Card 5

Variable	D1	D2	D3					
Type	F	F	F					

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MID	Material identification. A unique number has to be chosen.
RO	Mass density
E11	Elastic modulus in axial direction.
E22	Elastic modulus in transverse direction (E22=E33).
E12	Elastic shear modulus (E12=E31).
E23	Elastic shear modulus in transverse plane.
G	Shear modulus.
K	Bulk modulus for contact stiffness.
I11	Load curve for nominal axial stress versus volumetric strain.
I22	Load curve ID for nominal transverse stresses versus volumetric strain (I22=I33).
I12	Load curve ID for shear stress component 12 and 31 versus volumetric strain.(I12=I31).
I23	Load curve ID for shear stress component 23 versus volumetric strain.
IAA	Load curve ID (optional) for nominal stress versus volumetric strain for load at angle, ANG, relative to the material axis.
NY	Set to unity for a symmetric yield surface.
ANG	Angle corresponding to load curve ID, IAA.
MU	Damping coefficient for tensor viscosity which acts in both tension and compression. Recommended values vary between 0.05 to 0.10. If zero, tensor viscosity is not used, but bulk viscosity is used instead. Bulk viscosity creates a pressure as the element compresses that is added to the normal stresses, which can have the effect of creating transverse deformations when none are expected.

<u>VARIABLE</u>	<u>DESCRIPTION</u>
AOPT	Material axes option (see MAT_OPTION TROPIC_ELASTIC for a more complete description): EQ. 0.0: locally orthotropic with material axes determined by element 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.
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.

Remarks:

Tensor viscosity, which is activated by a nonzero value for MU, is generally more stable than bulk viscosity. A damping coefficient less than 0.01 has little effect, and a value greater than 0.10 may cause numerical instabilities.

***MAT_WOOD_{OPTION}**

This is Material Type 143. This is a transversely isotropic material and is available for solid elements. The user has the option of inputting his or her own material properties (<BLANK>), or requesting default material properties for Southern yellow pine (**PINE**) or Douglas fir (**FIR**). This model was developed by Murray [2002] under a contract from the FHWA.

Options include:

- PINE**
- FIR**
- <BLANK>**

such that the keyword cards appear as:

- *MAT_WOOD_PINE**
- *MAT_WOOD_FIR**
- *MAT_WOOD**

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	MID	RO	NPLOT	ITERS	IRATE	HARD	IFAIL	
Type	I	F	I	I	I	F	I	

Define the following card for the PINE and FIR options.

Card 2 1 2 3 4 5 6 7 8

Variable	MOIS	TEMP	QUAL_T	QUAL_C	UNITS	IQUAL		
Type	F	F	F	F	I	I		

Define the following cards, 2-6, for the <BLANK> option.

Card 2

Variable	EL	ET	GLT	GTR	PR			
Type	F	F	F	F	F			

Card 3

Variable	XT	XC	YT	YC	SXY	SYZ		
Type	F	F	F	F	F	F		

Card 4

Variable	GF1	GF2	BFIT	DMAX	GF1 _⊥	GF2 _⊥	DFIT	DMAX _⊥
Type	F	F	F	F	F	F	F	F

Card 5

Variable	FLPAR	FLPARC	POWPAR	FLPER	FLPERC	POWPER		
Type	F	F	F	F	F	F		

Card 6

Variable	NPAR	CPAR	NPER	CPER				
Type	F	F	F	F				

Define the following three cards for all options.

Card 3/7 1 2 3 4 5 6 7 8

Variable	AOPT							
Type	F							

Card 4/8

Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card 5/9

Variable	D1	D2	D3					
Type	F	F	F					

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MID	Material identification. A unique number has to be chosen.
RO	Mass density
NPLOT	Plotting Options: EQ. 1. Parallel damage (default). EQ. 2. Perpendicular damage.
ITERS	Number of plasticity algorithm iterations. The default is one iteration.
IRATE	Rate effects option: EQ. 0. Rate effects model turned off (default). EQ. 1. Rate effects model turned on.
HARD	Perfect plasticity override. Values greater than or equal to zero are allowed. Positive values model late time hardening in compression (an increase in strength with increasing strain). A zero value models perfect plasticity (no increase in strength with increasing strain). The default is zero..
IFAIL	Erosion perpendicular to the grain. EQ. 0. No (default). EQ. 1. Yes (not recommended except for debugging).
MOIS	Percent moisture content. If left blank, moisture content defaults to saturated at 30%.
TEMP	Temperature in °C. If left blank, temperature defaults to room temperature at 20 °C

VARIABLE	DESCRIPTION
QUAL_T	<p>Quality factor options. These quality factors reduce the clear wood tension, shear, and compression strengths as a function of grade.</p> <p>EQ. 0: Grade 1, 1D, 2, 2D. Predefined strength reduction factors are: Pine: Qual_T=0.47 in tension/shear. Qual_C=0.63 in compression. Fir: Qual_T=0.40 in tension/shear Qual_C=0.73 in compression.</p> <p>EQ.-1: DS-65 or SEI STR (pine and fir). Predefined strength reduction factors are: Qual_T=0.80 in tension/shear. Qual_C=0.93 in compression.</p> <p>EQ.-2: Clear wood. No strength reduction factors are applied: Qual_T=1.0. Qual_C=1.0.</p> <p>GT. 0: User defined quality factor in tension. Values between 0 and 1 are expected. Values greater than one are allowed, but may not be realistic.</p>
QUAL_C	<p>User defined quality factor in compression. This input value is used if Qual_T>0. Values between 0 and 1 are expected. Values greater than one are allowed, but may not be realistic. If left blank, a default value of Qual_C=Qual_T is used.</p>
UNITS	<p>Units options: EQ.0: GPa, mm, msec, Kg/mm³, kN. EQ.1: MPa, cm, msec, g/mm³, Nt. EQ.2: MPa, mm, sec, Mg/mm³, Nt. EQ.3: Psi, inch, sec, lb-s²/inch⁴, lb</p>
IQUAL	<p>Apply quality factors perpendicular to the grain: EQ.0: Yes (default). EQ 1: No.</p>
EL	Parallel normal modulus
ET	Perpendicular normal moduls.
GLT	Parallel shear modulus (GLT=GLR).
GTR	Perpendicular shear modulus.
PR	Parallel major Poisson's ratio.
XT	Parallel tensile strength.
XC	Parallel compressive strength.
YT	Perpendicular tensile strength.

<u>VARIABLE</u>	<u>DESCRIPTION</u>
YC	Perpendicular compressive strength.
SXY	Parallel shear strength.
SYZ	Perpendicular shear strength.
GF1	Parallel fracture energy in tension.
GF2	Parallel fracture energy in shear.
BFIT	Parallel softening parameter.
DMAX	Parallel maximum damage.
GF1 _⊥	Perpendicular fracture energy in tension.
GF2 _⊥	Perpendicular fracture energy in shear.
DFIT	Perpendicular softening parameter.
DMAX _⊥	Perpendicular maximum damage.
FLPAR	Parallel fluidity parameter for tension and shear.
FLPARC	Parallel fluidity parameter for compression.
POWPAR	Parallel power.
FLPER	Perpendicular fluidity parameter for tension and shear.
FLPERC	Perpendicular fluidity parameter for compression.
POWPER	Perpendicular power.
NPAR	Parallel hardening initiation.
CPAR	Parallel hardening rate.
NPER	Perpendicular hardening initiation.
CPER	Perpendicular hardening rate.
AOPT	Material axes option (see MAT_OPTION TROPIC_ELASTIC for a more complete description): EQ. 0. Locally orthotropic with material axes determined by element. EQ. 1. 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. Globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_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.

Remarks:

Material property data is for clear wood (small samples without defects like knots), whereas real structures are composed of graded wood. Clear wood is stronger than graded wood. Quality factors (strength reduction factors) are applied to the clear wood strengths to account for reductions in strength as a function of grade. One quality factor (Qual_T) is applied to the tensile and shear strengths. A second quality factor (Qual_C) is applied to the compressive strengths. As a option, predefined quality factors are provided based on correlations between LS-DYNA calculations and test data for pine and fir posts impacted by bogie vehicles. By default, quality factors are applied to both the parallel and perpendicular to the grain strengths. An option is available (IQUAL) to eliminate application perpendicular to the grain.

***MAT_PITZER_CRUSHABLE_FOAM**

This is Material Type 144. This model is for the simulation of isotropic crushable forms with strain rate effects. Uniaxial and triaxial test data have to be used. For the elastic response, the Poisson ratio is set to zero.

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	MID	RO	K	G	PR	TY	SRTV	
Type	I	F	F	F	F	F	F	

Card 2 1 2 3 4 5 6 7 8

Variable	LCPY	LCUYS	LCSR	VC	DFLG			
Type	I	I	I	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
PR	Poisson's ratio
TY	Tension yield.
SRTV	Young's modulus (E)
LCPY	Load curve ID giving pressure versus volumetric strain, see Figure 20.24.
LCUYS	Load curve ID giving uniaxial stress versus volumetric strain, see Figure 20.24.
LCSR	Load curve ID giving strain rate scale factor versus volumetric strain rate.
VC	Viscous damping coefficient (.05<recommended value<.50).

<u>VARIABLE</u>	<u>DESCRIPTION</u>
DFLG	Density flag: EQ.0.0: use initial density EQ.1.0: use current density (larger step size with less mass scaling).

Remarks:

The logarithmic volumetric strain is defined in terms of the relative volume, V , as:

$$\gamma = -\ln(V)$$

In defining the curves the stress and strain pairs should be positive values starting with a volumetric strain value of zero.

***MAT_SCHWER_MURRAY_CAP_MODEL**

This is Material Type 145. The Schwer & Murray Cap Model, a.k.a. Continuous Surface Cap Model, is a three invariant extension of the Geological Cap Model (Material Type 25) that also includes viscoplasticity for rate effects and damage mechanics to model strain softening. The primary references are Schwer and Murray (1994), Schwer (1994), and Murray and Lewis (1994). The model is appropriate for geomaterials including soils, concrete, and rocks.

The next six card images define the user specified parameters for the Schwer & Murray Cap Model. For LS-DYNA *structured* input the card images A through F correspond to Cards 3 through 8, respectively. For *keyword* input, these cards correspond to Cards 1 through 6. Note for keyword input the Material Identification (MID) and density (RO) must appear in the first two fields of Card A. *Warning:* no default input parameter values are assumed, but recommendations for the more obscure parameters are provided in the descriptions that follow.

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	MID	RO	SHEAR	BULK	GRUN	SHOCK	PORE	
Type	I	F	F	F	F	F	F	

Card 2 1 2 3 4 5 6 7 8

Variable	ALPHA	THETA	GAMMA	BETA	EFIT	FFIT	ALPHAN	CALPHA
Type	F	F	F	F	F	F	F	F

Card 3 1 2 3 4 5 6 7 8

Variable	RO	XO	IROCK	SECP	AFIT	BFIT	RDAMO	VPDAM
Type	F	F	F	F	F	F	F	F

Card 4 1 2 3 4 5 6 7 8

Variable	W	D1	D2	NPLOT	EPSMAX	CFIT	DFIT	TFAIL
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
ALPHAN	Kinematic strain hardening paramter, N^α
CALPHAN	Kinematic strain hardening parameter, c^α
R0	Initial cap surface ellipticity, R
X0	Initial cap surface J_1 (mean stress) axis intercept, $X(\kappa_0)$
IROCK	EQ.0: soils (cap can contract) EQ.1: rock/concrete
SECP	Shear enhanced compaction
AFIT	Ductile damage mechanics parameter (=1 no damage)
BFIT	Ductile damage mechanics parameter (=0 no damage)
RDAM0	Ductile damage mechanics parameter
VPDAM	Viscoplastic damage parameter
W	Plastic Volume Strain parameter, W
D1	Plastic Volume Strain parameter, D_1
D2	Plastic Volume Strain parameter, D_2
	$\varepsilon_v^p = W(1 - \exp[-D_1(X(\kappa) - X(\kappa_0)) - D_2(X(\kappa) - X(\kappa_0))^2])$
NPLOT	History variable post-processed as effective plastic strain (See Table 1 for history variables available for plotting)
EPSMAX	Maximum permitted strain increment (default = 0) $\Delta\varepsilon_{\max} = 0.05(\alpha - N^\alpha - \gamma) \min\left(\frac{1}{G}, \frac{R}{9K}\right)$ (calculated default)
CFIT	Brittle damage mechanics parameter (=1 no damage)
DFIT	Brittle damage mechanics parameter (=0 no damage)
TFAIL	Tensile failure stress
FAILFG	Failure Flag, failed element: EQ.0: stresses zeroed (use for ALE and EFG) EQ.1: removed from database (preferred)
DBETA	Rounded vertices parameter, $\Delta\beta_0$

<u>VARIABLE</u>	<u>DESCRIPTION</u>
DDELTA	Rounded vertices parameter, δ
VPTAU	Viscoplasticity relaxation time parameter, τ
ALPHA1	Torsion scaling parameter, α_1 $\alpha_1 < 0 \rightarrow \alpha_1 = \text{Friction Angle (degrees)}$
THETA1	Torsion scaling parameter, θ_1
GAMMA1	Torsion scaling parameter, γ_1
BETA1	Torsion scaling parameter, β_1 $Q_1 = \alpha_1 - \gamma_1 \exp(-\beta_1 J_1) + \theta_1 J_1$
ALPHA2	Tri-axial extension scaling parameter, α_2
THETA2	Tri-axial extension scaling parameter, θ_2
GAMMA2	Tri-axial extension scaling parameter, γ_2
BETA2	Tri-axial extension scaling parameter, β_2 $Q_2 = \alpha_2 - \gamma_2 \exp(-\beta_2 J_1) + \theta_2 J_1$

Output History Variables

All the output parameters listed in Table 1 is available for post-processing using LS-POST and its displayed list of History Variables. The LS-DYNA input parameter NEIPH should be set to 26; see for example the keyword input for *DATABASE_EXTENT_BINARY.

Table 1. Output variables for psot-processing using NPLOT parameter.

NPLOT	Function	Description
1	$X(\kappa)$	J_1 intercept of cap surface
2	$L(\kappa)$	J_1 value at cap-shear surface intercept
3	R	Cap surface ellipticity
4	\tilde{R}	Rubin function
5	ϵ_v^p	Plastic volume strain
6		Yield Flag (= 0 elastic)
7		Number of strain sub-increments
8	G^α	Kinematic hardening parameter
9	J_2^α	Kinematic hardening back stress
10		Effective strain rate
11		Ductile damage
12		Ductile damage threshold
13		Strain energy
14		Brittle damage
15		Brittle damage threshold
16		Brittle energy norm
17		J_1 (w/o visco-damage/plastic)
18		J_2' (w/o visco-damage/plastic)
19		J_3' (w/o visco-damage/plastic)
20		\hat{J}_3 (w/o visco-damage/plastic)
21	β	Lode Angle
22		future variable
23		future variable
24		future variable
25		future variable
26		future variable

Sample Input for Concrete

Gran and Senseny (1996) report the axial stress versus strain response for twelve unconfined compression tests of concrete, used in scale-model reinforced-concrete wall tests. The Schwer & Murray Cap Model parameters provided below were used, see Schwer (2001), to model the unconfined compression test stress-strain response for the nominal 40 MPa strength concrete reported by Gran & Senseny. The basic units for the provided parameters are length in millimeters (mm), time in milliseconds (msec), and mass in grams (g). This base unit set yields units of force in Newtons (N) and pressure in Mega-Pascals (MPa).

Keyword:

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	SHEAR	BULK	GRUN	SHOCK	PORE	
Type	I	2.3E-2	1.048E4	1.168E4	0.0	0.0	1.	

Card 2	1	2	3	4	5	6	7	8
Variable	ALPHA	THETA	GAMMA	BETA	EFIT	FFIT	ALPHAN	CALPHA
Type	190.0	0.0	184.2	2.5E-3	0.999	0.7	2.5	2.5E3

Card 3	1	2	3	4	5	6	7	8
Variable	R0	X0	IROCK	SECP	AFIT	BFIT	RDAM0	VPDAM
Type	5.0	100.0	1.0	0.0	0.999	0.3	0.94	0.0

Card 4	1	2	3	4	5	6	7	8
Variable	W	D1	D2	NPLOT	EPSMAX	CFIT	DFIT	TFAIL
Type	5.0E-2	2.5E-4	3.5E-7	23.0	0.0	1.0	300.0	92.2

Card 5	1	2	3	4	5	6	7	8
Variable	FAILFL		DBETA	DDELTA	VPTAU			
Type	1.0		0.0	0.0	0.0			

Card 6	1	2	3	4	5	6	7	8
Variable	ALPHA1	THETA1	GAMMA1	BETA1	ALPHA2	THETA2	GAMMA2	BETA2
Type	0.747	3.3E-4	0.17	5.0E-2	0.66	4.0E-4	0.16	5.0E-2

***MAT_1DOF_GENERALIZED_SPRING**

This is Material Type 146. This is a linear spring or damper that allows different degrees-of-freedom at two nodes to be coupled with a liner spring and/or damper.

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	MID	RO	K	C	SCLN1	SCLN2	DOFN1	DOFN2
Type	I	F	F	F	F	F	I	I

Card 2

Variable	CID							
Type	I							

<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 definition.
K	Spring stiffness.
C	Damping constant.
SCLN1	Scale factor on force at node 1. Default=1.0.
SCLN2	Scale factor on force at node 2. Default=1.0.
DOFN1	Active degree-of-freedom at node 1, a number between 1 to 6 where 1 in x-translation and 4 is x-rotation. If this parameter is defined in the SECTION_BEAM definition or on the ELEMENT_BEAM_SCALAR card, then the value here, if defined, is ignored.
DOFN2	Active degree-of-freedom at node 2, a number between 1 to 6. If this parameter is defined in the SECTION_BEAM definition or on the ELEMENT_BEAM_SCALAR card, then the value here, if defined, is ignored.
CID	Local coordinate system at node 1. This coordinate system can be overwritten by a local system specified on the *ELEMENT_BEAM or *SECTION_BEAM keyword input. If no coordinate system is specified, the global system is used.

*MAT_FHWA_SOIL

This is Material Type 147. This is an isotropic material with damage and is available for solid elements. The model has a modified Mohr-Coulomb surface to determine the pressure dependent peak shear strength. It was developed for applications involving roadbase soils by Lewis [1999] for the FHWA, who extended the work of Abbo and Sloan [1995] to include excess pore water effects.

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	MID	RO	NPLOT	SPGRAV	RHOWAT	VN	GAMMAR	INTRMX
Type	I	F	I	F	F	F	F	I
Default	none	none	1	none	1.0	0.0	0.0	1

Card 2

Variable	K	G	PHIMAX	AHYP	COH	ECCEN	AN	ET
Type	F	F	F	F	F	F		
Default	none	none	none	none	none	none	none	none

Card 3

Variable	MCONT	PWD1	PWKSK	PWD2	PHIRES	DINT	VDFM	DAMLEV
Type	F	F	F	F	F	F	F	F
Default	none	none	none	none	0.0	none	none	none

Card 4 1 2 3 4 5 6 7 8

Variable	EPSMAX								
Type	F								
Default	none								

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

MID	Material identification. A unique number has to be chosen.
RO	Mass density
NPLOT	Plotting options EQ.1: Effective Strain EQ.2: Damage Criterion Threshold EQ.3: Damage (diso) EQ.4: Current Damage Criterion EQ.5: Not used EQ.6: Current Friction Angle (phi)
SPGRAV	Specific Gravity of Soil used to get porosity.
RHOWAT	Density of water in model units - used to determine air void strain (saturation)
VN	Viscoplasticity parameter (strain-rate enhanced strength)
GAMMAR	Viscoplasticity parameter (strain-rate enhanced strength)
ITERMAX	Maximum number of plasticity iterations (default 1)
K	Bulk Modulus (non-zero)
G	Shear modulus (non-zero)
PHIMAX	Peak Shear Strength Angle (friction angle) (radians)
AHYP	Coefficient A for modified Drucker-Prager Surface
COH	Cohesion \bar{n} Shear Strength at zero confinement (overburden)
ECCEN	Eccentricity parameter for third invariant effects
AN	Strain hardening percent of phimax where non-linear effects start

<u>VARIABLE</u>	<u>DESCRIPTION</u>
ET	Strain Hardening Amount of non-linear effects
MCONT	Moisture Content of Soil (Determines amount of air voids) (0-1.00)
PWD1	Parameter for pore water effects on bulk modulus
PWKSK	Skeleton bulk modulus- Pore water parameter ñ set to zero to eliminate effects
PWD2	Parameter for pore water effects on the effective pressure (confinement)
PHIRES	The minimum internal friction angle, radians (residual shear strength)
DINT	Volumetric Strain at Initial damage threshold, EMBED Equation.3
VDFM	Void formation energy (like fracture energy)
DAMLEV	Level of damage that will cause element deletion (0.0-1.0)
EPSMAX	Maximum principle failure strain

***MAT_FHWA_SOIL_NEBRASKA**

This is an option to use the default properties determined for soils used at the University of Nebraska (Lincoln). The default units used for this material are millimeter, millisecond, and kilograms. If different units are desired, the conversion factors must be input.

This is Material Type 147. This is an isotropic material with damage and is available for solid elements. The model has a modified Mohr-Coulomb surface to determine the pressure dependent peak shear strength. It was developed for applications involving roadbase soils.

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	MID	FCTIM	FCTMAS	FCTLEN				
Type	I	F	I	F	F	F	F	I
Default	none	none	1	none	1.0	0.0	0.0	1

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MID	Material identification. A unique number has to be chosen.
FCTIM	Factor to multiply milliseconds by to get desired time units
FCTMAS	Factor to multiply kilograms by to get desired mass units
FCTLEN	Factor to multiply millimeters by to get desired length units

*MAT_GAS_MIXTURE

This is Material Type 148. This model is for the simulation of gas mixtures and it only works together with the multi-material ALE formulation. This keyword needs to be combined with *INITIAL_GAS_MIXTURE for the initialization of gas densities and temperature.

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	MID							
Type	I							

Card 2

Variable	CV1	CV2	CV3	CV4	CV5	CV6	CV7	CV8
Type	F	F	F	F	F	F	F	F

Card 2

Variable	CP1	CP2	CP3	CP4	CP5	CP6	CP7	CP8
Type	F	F	F	F	F	F	F	F

VARIABLE

DESCRIPTION

- MID Material identification. A unique number has to be chosen.
- CV1-CV8 Heat capacity at constant volume for up to eight different gases.
- CP1-CP8 Heat capacity at constant pressure for up to eight different gases..

Remarks:

The initial temperature and the density of the different gas species are defined with the keyword *INITIAL_GAS_MIXTURE.

***MAT_CFD_{OPTION}**

This is material type 150. It allows constant, isotropic fluid properties to be defined for the incompressible/low-Mach CFD solver. The *MAT_CFD_ cards allow fluid properties to be defined in a stand-alone fluid analysis or in a coupled fluid/structure analysis, see *CONTROL_SOLUTION.

Options include:

<BLANK>

CONSTANT

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	MID	RHO	MU	K	CP	BETA	TREF	
Type	I	F	F	F	F	F	F	
Default	-	-	-	-	-	-	-	

Card 2 1 2 3 4 5 6 7 8

Variable	GX	GY	GZ	DIFF1	DIFF2	DIFF3	DIFF4	DIFF5
Type	F	F	F	F	F	F	F	F
Default	-	-	-	-	-	-	-	-

Card 3 1 2 3 4 5 6 7 8

Variable	DIFF6	DIFF7	DIFF8	DIFF9	DIFF10			
Type	F	F	F	F	F			
Default	-	-	-	-	-			

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MID	Material identification, a unique number has to be chosen.
RHO	Fluid density
MU	Fluid viscosity
K	Thermal Conductivity
CP	Heat capacity
BETA	Coefficient of expansion
TREF	Reference temperature
GX	Gravitational acceleration in x-direction
GY	Gravitational acceleration in y-direction
GZ	Gravitational acceleration in z-direction
DIFF1	Species-1 diffusivity
DIFF2	Species-2 diffusivity
DIFF3	Species-3 diffusivity
...	
DIFF10	Species-10 diffusivity

***MAT_DESHPANDE_FLECK_FOAM**

This is material type 154 for solid elements. This material is for modeling aluminum foam used as a filler material in aluminum extrusions to enhance the energy absorbing capability of the extrusion. Such energy absorbers are used in vehicles to dissipate energy during impact. This model was developed by Reyes, Hopperstad, Berstad, and Langseth [2002] and is based on the foam model by Deshpande and Fleck [2000].

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	MID	RHO	E	PR	ALPHA	GAMMA		
Type	I	F	F	F	F	F		
Default	-	-	-	-	-	-		

Card 2 1 2 3 4 5 6 7 8

Variable	EPSD	ALPHA2	BETA	SIGP	DERFI	CFAIL		
Type	F	F	F	F	F	F		
Default	-	-	-	-	-	-		

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MID	Material identification, a unique number has to be chosen.
RHO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.
ALPHA	Controls shape of yield surface.
GAMMA	See remarks.
EPSD	Densification strain.
ALPHA2	See remarks.

VARIABLE	DESCRIPTION
BETA	See remarks.
SIGP	See remarks.
DERFI	Type of derivation used in material subroutine EQ.0: Numerical derivation EQ.1: Analytical derivation
CFAIL	Failure strain.

The yield stress function Φ is defined by:

$$\Phi = \hat{\sigma} - \sigma_y$$

The equivalent stress $\hat{\sigma}$ is given by:

$$\hat{\sigma}^2 = \frac{\sigma_{VM}^2 + \alpha^2 \sigma_m^2}{1 + \left(\frac{\alpha}{3}\right)^2}$$

where, σ_{VM} , is the von Mises effective stress:

$$\sigma_{VM} = \sqrt{\frac{2}{3} \sigma^{dev} : \sigma^{dev}}$$

In this equation σ_m and σ^{dev} are the mean and deviatoric stress:

$$\sigma^{dev} = \sigma - \sigma_m I$$

The yield stress σ_y can be expressed as:

$$\sigma_y = \sigma_p + \gamma \frac{\hat{\epsilon}}{\epsilon_D} + \alpha_2 \left(\frac{1}{1 - \left(\frac{\hat{\epsilon}}{\epsilon_D}\right)^\beta} \right)$$

Here, σ_p , α_2 , γ and β are material parameters. The densification strain ϵ_D is defined as:

$$\epsilon_D = -\ln\left(\frac{\rho_f}{\rho_{f0}}\right)$$

where ρ_f is the foam density and ρ_{f0} is the density of the virgin material.

***MAT_COMPOSITE_{OPTION}_MSC**

Available options are:

<BLANK>

DMG

These are Material Types 161 and 162. These models may be used to model the progressive failure analysis for composite materials consisting of unidirectional and woven fabric layers. The progressive layer failure criteria have been established by adopting the methodology developed by Hashin [1980] with a generalization to include the effect of highly constrained pressure on composite failure. These failure models can be used to effectively simulate fiber failure, matrix damage, and delamination behavior under all conditions - opening, closure, and sliding of failure surfaces. The model with DMG option (material 162) is a generalization of the basic layer failure model of Material 161 by adopting the damage mechanics approach for characterizing the softening behavior after damage initiation. These models require an additional license from Materials Sciences Corporation, which developed and supports these models.

Card 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	AOPT				
Type	F	F	F	F				

Card 3

Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card 4

Variable	V1	V2	V3	D1	D2	D3	BETA	
Type	F	F	F	F	F	F	F	

Card 5

Variable	SAT	SAC	SBT	SBC	SCT	SFC	SFS	SAB
Type	F	F	F	F	F	F	F	F

Card 6

Variable	SBC	SCA	SFFC	AMODEL	PHIC	E_LIMT	S_DELM	
Type	F	F	F	F	F	F	F	

Card 7

Variable	OMGMX	ECRSH	EEXPXN	CERATE1	ERTRF	AM1		
Type	F	F	F	F	F	F		

Define the following cards if and only if the option DMG is specified

Card 8

Variable	AM2	AM3	AM4	CERATE2	CERATE3	CERATE4	IVISCO	
Type	F	F	F	F	F	F	F	

Define the following card if and only if IVISCO=1

Card 9 1 2 3 4 5 6 7 8

Variable	RGIAB	RGIBC	RGICA	DBETA				
Type	F	F	F	F				

<u>VARIABLE</u>	<u>DESCRIPTION</u>
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 – through thickness direction
PRBA	ν_{ba} , Poisson's ratio ba
NUCA	ν_{ca} , Poisson's ratio ca
NUCB	ν_{cb} , Poisson's ratio cb
GAB	G_{ab} , shear modulus ab
GBC	G_{bc} , shear modulus bc
GCA	G_{ca} , shear modulus ca
AOPT	Material axes option, see Figure 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 by *DEFINE_COORDINATE_NODES. EQ. 1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center, to define the a-direction. EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR.
XP YP ZP	Define coordinates of point p for AOPT = 1.
A1 A2 A3	Define components of vector a for AOPT = 2.

<u>VARIABLE</u>	<u>DESCRIPTION</u>
V1 V2 V3	Define components of vector v for AOPT = 3.
D1 D2 D3	Define components of vector d for AOPT = 2.
BETA	Layer in-plane rotational angle in degrees.
SAT	Longitudinal tensile strength
SAC	Longitudinal compressive strength
SBT	Transverse tensile strength
SBC	Transverse compressive strength
SCT	Through thickness tensile strength
SFC	Crush strength
SFS	Fiber mode shear strength
SAB	Matrix mode shear strength, ab plane, see below.
SBC	Matrix mode shear strength, bc plane, see below.
SCA	Matrix mode shear strength, ca plane, see below.
SFFC	Scale factor for residual compressive strength
AMODEL	Material models: <p style="margin-left: 40px;">EQ. 1: Unidirectional layer model EQ. 2: Fabric layer model</p>
PHIC	Coulomb friction angle for matrix and delamination failure, <90
E_LIMT	Element eroding axial strain
S_DELM	Scale factor for delamination criterion
OMGMX	Limit damage parameter for elastic modulus reduction
ECRSH	Limit compressive volume strain for element eroding
EEXPN	Limit tensile volume strain for element eroding
CERATE1	Coefficient for strain rate dependent strength properties
ERTRF	Reference strain rate
AM1	Coefficient for strain rate softening property for fiber damage in a direction.
AM2	Coefficient for strain rate softening property for fiber damage in b direction.

VARIABLE	DESCRIPTION
AM3	Coefficient for strain rate softening property for fiber crush and punch shear damage.
AM4	Coefficient for strain rate softening property for matrix and delamination damage.
CERATE2	Coefficient for strain rate dependent axial moduli.
CERATE3	Coefficient for strain rate dependent shear moduli.
CERATE4	Coefficient for strain rate dependent transverse moduli.
IVISCO	Viscoelastic analysis: <div style="margin-left: 40px;">EQ. 0: Do not perform viscoelastic analysis. EQ. 1: Perform viscoelastic analysis.</div>
RGIAB	Ratio of long-time shear modulus ab and GAB.
RGIBC	Ratio of long-time shear modulus bc and GBC.
RGICA	Ratio of long-time shear modulus ca and GCA.
VBETA	Viscoelastic decay constant

Material Models:

The unidirectional and fabric layer failure criteria and the associated property degradation models for material 161 are described as follows. All the failure criteria are expressed in terms of stress components based on ply level stresses ($\sigma_a, \sigma_b, \sigma_c, \tau_{ab}, \tau_{bc}, \tau_{ca}$) and the associated elastic moduli are ($E_a, E_b, E_c, G_{ab}, G_{bc}, G_{ca}$). Note that for the unidirectional model, a, b and c denote the fiber, in-plane transverse and out-of-plane directions, respectively, while for the fabric model, a, b and c denote the in-plane fill, in-plane warp and out-of-plane directions, respectively.

Unidirectional lamina model

Three criteria are used for fiber failure, one in tension/shear, one in compression and another one in crush under pressure. They are chosen in terms of quadratic stress forms as follows:

Tensile/shear fiber mode:

$$f_1 = \left(\frac{\langle \sigma_a \rangle}{S_{aT}} \right)^2 + \left(\frac{\tau_{ab}^2 + \tau_{ca}^2}{S_{FS}^2} \right) - 1 = 0$$

Compression fiber mode:

$$f_2 = \left(\frac{\langle \sigma'_a \rangle}{S_{ac}} \right)^2 - 1 = 0, \quad \sigma'_a = -\sigma_a + \left\langle -\frac{\sigma_b + \sigma_c}{2} \right\rangle$$

Crush mode:

$$f_3 = \left(\frac{\langle p \rangle}{S_{FC}} \right)^2 - 1 = 0, \quad p = -\frac{\sigma_a + \sigma_b + \sigma_c}{3}$$

where $\langle \rangle$ are Macaulay brackets, S_{aT} and S_{ac} are the tensile and compressive strengths in the fiber direction, and S_{FS} and S_{FC} are the layer strengths associated with the fiber shear and crush failure, respectively.

Matrix mode failures must occur without fiber failure, and hence they will be on planes parallel to fibers. For simplicity, only two failure planes are considered: one is perpendicular to the planes of layering and the other one is parallel to them. The matrix failure criteria for the failure plane perpendicular and parallel to the layering planes, respectively, have the forms:

Perpendicular matrix mode:

$$f_4 = \left(\frac{\langle \sigma_b \rangle}{S_{bT}} \right)^2 + \left(\frac{\tau_{bc}}{S'_{bc}} \right)^2 + \left(\frac{\tau_{ab}}{S_{ab}} \right)^2 - 1 = 0$$

Parallel matrix mode (Delamination):

$$f_5 = S^2 \left\{ \left(\frac{\langle \sigma_c \rangle}{S_{bT}} \right)^2 + \left(\frac{\tau_{bc}}{S''_{bc}} \right)^2 + \left(\frac{\tau_{ca}}{S_{ca}} \right)^2 \right\} - 1 = 0$$

where S_{bT} is the transverse tensile strength. Based on the Coulomb-Mohr theory, the shear strengths for the transverse shear failure and the two axial shear failure modes are assumed to be the forms,

$$S_{ab} = S_{ab}^{(0)} + \tan(\varphi) \langle -\sigma_b \rangle$$

$$S'_{bc} = S_{bc}^{(0)} + \tan(\varphi) \langle -\sigma_b \rangle$$

$$S_{ca} = S_{ca}^{(0)} + \tan(\varphi) \langle -\sigma_c \rangle$$

$$S''_{bc} = S_{bc}^{(0)} + \tan(\varphi) \langle -\sigma_c \rangle$$

where φ is a material constant as $\tan(\varphi)$ is similar to the coefficient of friction, and $S_{ab}^{(0)}$, $S_{ca}^{(0)}$ and $S_{bc}^{(0)}$ are the shear strength values of the corresponding tensile modes.

Failure predicted by the criterion of f_4 can be referred to as transverse matrix failure, while the matrix failure predicted by f_5 , which is parallel to the layer, can be referred as the delamination mode when it occurs within the elements that are adjacent to the ply interface. Note that a scale factor S is introduced to provide better correlation of delamination area with experiments. The scale factor S can be determined by fitting the analytical prediction to experimental data for the delamination area.

When fiber failure in tension/shear mode is predicted in a layer by f_1 , the load carrying capacity of that layer is completely eliminated. All the stress components are reduced to zero instantaneously (100 time steps to avoid numerical instability). For compressive fiber failure, the layer is assumed to carry a residual axial load, while the transverse load carrying capacity is reduced to zero. When the fiber compressive failure mode is reached due to f_2 , the axial layer compressive strength stress is assumed to reduce to a residual value S_{RC} ($=SFFC * S_{AC}$). The axial stress is then assumed to remain constant, i.e., $\sigma_a = -S_{RC}$, for continuous compressive loading, while the subsequent unloading curve follows a reduced axial modulus to zero axial stress and strain state. When the fiber crush failure occurs, the material is assumed to behave elastically for compressive pressure, $p > 0$, and to carry no load for tensile pressure, $p < 0$.

When a matrix failure (delamination) in the a-b plane is predicted, the strength values for $S_{ca}^{(0)}$ and $S_{bc}^{(0)}$ are set to zero. This results in reducing the stress components σ_c , τ_{bc} and τ_{ca} to the fractured material strength surface. For tensile mode, $\sigma_c > 0$, these stress components are reduced to zero. For compressive mode, $\sigma_c < 0$, the normal stress σ_c is assumed to deform elastically for the closed matrix crack. Loading on the failure envelop, the shear stresses are assumed to 'slide' on the fractured strength surface (frictional shear stresses) like in an ideal plastic material, while the subsequent unloading shear stress-strain path follows reduced shear moduli to the zero shear stress and strain state for both τ_{bc} and τ_{ca} components.

The post failure behavior for the matrix crack in the a-c plane due to f_4 is modeled in the same fashion as that in the a-b plane as described above. In this case, when failure occur, $S_{ab}^{(0)}$ and $S_{bc}^{(0)}$ are reduced to zero instantaneously. The post fracture response is then governed by failure criterion of f_5 with $S_{ab}^{(0)} = 0$ and $S_{bc}^{(0)} = 0$. For tensile mode, $\sigma_b > 0$, σ_b , τ_{ab} and τ_{bc} are zero. For compressive mode, $\sigma_b < 0$, σ_b is assumed to be elastic, while τ_{ab} and τ_{bc} 'slide' on the fracture strength surface as in an ideal plastic material, and the unloading path follows reduced shear moduli to the zero shear stress and strain state. It should be noted that τ_{bc} is governed by both the failure functions and should lie within or on each of these two strength surfaces.

Fabric lamina model

The fiber failure criteria of Hashin for a unidirectional layer are generalized to characterize the fiber damage in terms of strain components for a plain weave layer. The fill and warp fiber tensile/shear failure are given by the quadratic interaction between the associated axial and shear stresses, i.e.

$$f_6 = \left(\frac{\langle \sigma_a \rangle}{S_{aT}} \right)^2 + \frac{(\tau_{ab}^2 + \tau_{ca}^2)}{S_{aFS}^2} - 1 = 0$$

$$f_7 = \left(\frac{\langle \sigma_b \rangle}{S_{bT}} \right)^2 + \frac{(\tau_{ab}^2 + \tau_{bc}^2)}{S_{bFS}^2} - 1 = 0$$

where S_{aT} and S_{bT} are the axial tensile strengths in the fill and warp directions, respectively, and S_{aFS} and S_{bFS} are the layer shear strengths due to fiber shear failure in the fill and warp directions. These failure criteria are applicable when the associated σ_a or σ_b is positive. It is assumed $S_{aFS} = SFS$, and

$$S_{bFS} = SFS * S_{bT} / S_{aT}.$$

When σ_a or σ_b is compressive, it is assumed that the in-plane compressive failure in both the fill and warp directions are given by the maximum stress criterion, i.e.

$$f_8 = \left[\frac{\langle \sigma_a' \rangle}{S_{aC}} \right]^2 - 1 = 0, \quad \sigma_a' = -\sigma_a + \langle -\sigma_c \rangle$$

$$f_9 = \left[\frac{\langle \sigma_b' \rangle}{S_{bC}} \right]^2 - 1 = 0, \quad \sigma_b' = -\sigma_b + \langle -\sigma_c \rangle$$

where S_{aC} and S_{bC} are the axial compressive strengths in the fill and warp directions, respectively. The crush failure under compressive pressure is

$$f_{10} = \left(\frac{\langle p \rangle}{S_{FC}} \right)^2 - 1 = 0, \quad p = -\frac{\sigma_a + \sigma_b + \sigma_c}{3}$$

A plain weave layer can fail under in-plane shear stress without the occurrence of fiber breakage. This in-plane matrix failure mode is given by

$$f_{11} = \left(\frac{\tau_{ab}}{S_{ab}} \right)^2 - 1 = 0$$

where S_{ab} is the layer shear strength due to matrix shear failure.

Another failure mode, which is due to the quadratic interaction between the thickness stresses, is expected to be mainly a matrix failure. This through the thickness matrix failure criterion is

$$f_{12} = S^2 \left\{ \left(\frac{\langle \sigma_c \rangle}{S_{cT}} \right)^2 + \left(\frac{\tau_{bc}}{S_{bc}} \right)^2 + \left(\frac{\tau_{ca}}{S_{ca}} \right)^2 \right\} - 1 = 0$$

where S_{cT} is the through the thickness tensile strength, and S_{bc} , and S_{ca} are the shear strengths assumed to depend on the compressive normal stress s_c , i.e.,

$$\begin{Bmatrix} S_{ca} \\ S_{bc} \end{Bmatrix} = \begin{Bmatrix} S_{ca}^{(0)} \\ S_{bc}^{(0)} \end{Bmatrix} + \tan(\varphi) \langle -\sigma_c \rangle$$

When failure predicted by this criterion occurs within elements that are adjacent to the ply interface, the failure plane is expected to be parallel to the layering planes, and, thus, can be referred to as the delamination mode. Note that a scale factor S is introduced to provide better correlation of delamination area with experiments. The scale factor S can be determined by fitting the analytical prediction to experimental data for the delamination area.

Similar to the unidirectional model, when fiber tensile/shear failure is predicted in a layer by f_6 or f_7 , the load carrying capacity of that layer in the associated direction is completely eliminated. For compressive fiber failure due to by f_8 or f_9 , the layer is assumed to carry a residual axial load in the failed direction, while the load carrying capacity transverse to the failed direction is assumed unchanged. When the compressive axial stress in a layer reaches the compressive axial strength S_{ac} or S_{bc} , the axial layer stress is assumed to be reduced to the residual strength S_{arc} or S_{brc} where $S_{arc} = SFFC * S_{ac}$ and $S_{brc} = SFFC * S_{bc}$. The axial stress is assumed to remain constant, i.e., $\sigma_a = -S_{arc}$ or $\sigma_b = -S_{brc}$, for continuous compressive loading, while the subsequent unloading curve follows a reduced axial modulus. When the fiber crush failure is occurred, the material is assumed to behave elastically for compressive pressure, $p > 0$, and to carry no load for tensile pressure, $p < 0$.

When the in-plane matrix shear failure is predicted by f_{11} the axial load carrying capacity within a failed element is assumed unchanged, while the in-plane shear stress is assumed to be reduced to zero.

For through the thickness matrix (delamination) failure given by equations f_{12} , the in-plane load carrying capacity within the element is assumed to be elastic, while the strength values for the tensile mode, $S_{ca}^{(0)}$ and $S_{bc}^{(0)}$, are set to zero. For tensile mode, $\sigma_c > 0$, the through the thickness stress components are reduced to zero. For compressive mode, $\sigma_c < 0$, σ_c is assumed to be elastic, while τ_{bc} and τ_{ca} 'slide' on the fracture strength surface as in an ideal plastic material, and the unloading path follows reduced shear moduli to the zero shear stress and strain state.

The effect of strain-rate on the layer strength values of the fiber failure modes is modeled by the strain-rate dependent functions for the strength values $\{S_{RT}\}$ as

$$\{S_{RT}\} = \{S_0\} \left(1 + C_{rate1} \ln \frac{\{\dot{\epsilon}\}}{\dot{\epsilon}_0} \right)$$

$$\{S_{RT}\} = \begin{Bmatrix} S_{aT} \\ S_{aC} \\ S_{bT} \\ S_{bC} \\ S_{FC} \\ S_{FS} \end{Bmatrix} \text{ and } \{\dot{\epsilon}\} = \begin{Bmatrix} |\dot{\epsilon}_a| \\ |\dot{\epsilon}_a| \\ |\dot{\epsilon}_b| \\ |\dot{\epsilon}_b| \\ |\dot{\epsilon}_c| \\ \left(\dot{\epsilon}_{ca}^2 + \dot{\epsilon}_{bc}^2\right)^{1/2} \end{Bmatrix}$$

where C_{rate} is the strain-rate constants, and $\{S_0\}$ are the strength values of $\{S_{RT}\}$ at the reference strain-rate $\dot{\epsilon}_0$.

Damage model

The damage model is a generalization of the layer failure model of Material 161 by adopting the MLT damage mechanics approach, Matzenmiller et al. [1995], for characterizing the softening behavior after damage initiation. Complete model description is given in Yen [2002]. The damage functions, which are expressed in terms of ply level engineering strains, are converted from the above failure criteria of fiber and matrix failure modes by neglecting the Poisson’s effect. Elastic moduli reduction is expressed in terms of the associated damage parameters ϖ_i :

$$E_i' = (1 - \varpi_i)E_i$$

$$\varpi_i = 1 - \exp(-r_i^{m_i} / m_i) \quad r_i \geq 0 \quad i = 1, \dots, 6$$

where E_i are the initial elastic moduli, E_i' are the reduced elastic moduli, r_i are the damage thresholds computed from the associated damage functions for fiber damage, matrix damage and delamination, and m_i are material damage parameters, which are currently assumed to be independent of strain-rate. The damage function is formulated to account for the overall nonlinear elastic response of a lamina including the initial ‘hardening’ and the subsequent softening beyond the ultimate strengths.

In the damage model (material 162), the effect of strain-rate on the nonlinear stress-strain response of a composite layer is modeled by the strain-rate dependent functions for the elastic moduli $\{E_{RT}\}$ as

$$\{E_{RT}\} = \{E_0\} \left(1 + \{C_{rate}\} \ln \frac{\{\dot{\epsilon}\}}{\dot{\epsilon}_0} \right)$$

$$\{E_{RT}\} = \begin{Bmatrix} E_a \\ E_b \\ E_c \\ G_{ab} \\ G_{bc} \\ G_{ca} \end{Bmatrix}, \{\dot{\epsilon}\} = \begin{Bmatrix} |\dot{\epsilon}_a| \\ |\dot{\epsilon}_b| \\ |\dot{\epsilon}_c| \\ |\dot{\epsilon}_{ab}| \\ |\dot{\epsilon}_{bc}| \\ |\dot{\epsilon}_{ca}| \end{Bmatrix} \text{ and } \{C_{rate}\} = \begin{Bmatrix} C_{rate2} \\ C_{rate2} \\ C_{rate4} \\ C_{rate3} \\ C_{rate3} \\ C_{rate3} \end{Bmatrix}$$

where $\{C_{rate}\}$ are the strain-rate constants. $\{E_0\}$ are the modulus values of $\{E_{RT}\}$ at the reference strain-rate $\dot{\epsilon}_0$.

Element Erosion:

A failed element is eroded in any of three different ways:

1. If fiber tensile failure in a unidirectional layer is predicted in the element and the axial tensile strain is greater than E_LIMIT. For a fabric layer, both in-plane directions are failed and exceed E_LIMIT.
2. If compressive relative volume in a failed element is smaller than ECRSH.
3. If tensile relative volume in a failed element is greater than EEXPN.

Damage History Parameters:

Information about the damage history variables for the associated failure modes can be plotted in LSPOST and LS-TAURUS. These additional variables are tabulated below:

History Variable	Description	Value	LS-POST components	LS-TAURUS components
1. efa(I)	Fiber mode in a	0-elastic	7	87
2. efb(I)	Fiber mode in b		8	88
3. efp(I)	Fiber crush mode		9	89
4. em(I)	Perpendicular matrix mode	≥1-failed	10	90
5. ed(I)	Parallel matrix/ delamination mode		11	91
6. delm(I)	delamination mode		12	92

*MAT_MODIFIED_CRUSHABLE_FOAM

This is Material Type 163 which is dedicated to modeling crushable foam with optional damping, tension cutoff, and strain rate effects. Unloading is fully elastic. Tension is treated as elastic-perfectly-plastic at the tension cut-off value.

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	MID	RO	E	PR	TID	TSC	DAMP	NCYCLE
Type	I	F	F	F	F	F	F	F
Default	none	none	none	none	none	0.0	0.10	12.

Card 2

Variable	SRCLMT							
Type	F							
Default	1.E+20							

<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
TID	Table ID defining yield stress versus volumetric strain, γ , at different strain rates.
TSC	Tensile stress cutoff. A nonzero, positive value is strongly recommended for realistic behavior.
DAMP	Rate sensitivity via damping coefficient (.05<recommended value<.50).
NCYCLE	Number of cycles to determine the average volumetric strain rate.
SRCLMT	Strain rate change limit.

Remarks:

The volumetric strain is defined in terms of the relative volume, V , as:

$$\gamma = 1.-V$$

The relative volume is defined as the ratio of the current to the initial volume. In place of the effective plastic strain in the D3PLOT database, the integrated volumetric strain is output.

This material is an extension of material 63, *MAT_CRUSHABLE_FOAM. It allows the yield stress to be a function of both volumetric strain rate and volumetric strain. Rate effects are accounted for by defining a table of curves using *DEFINE_TABLE. Each curve defines the yield stress versus volumetric strain for a different strain rate. The yield stress is obtained by interpolating between the two curves that bound the strain rate.

To prevent high frequency oscillations in the strain rate from causing similar high frequency oscillations in the yield stress, a modified volumetric strain rate is used when interpolating to obtain the yield stress. The modified strain rate is obtained as follows. If NYCLE is >1, then the modified strain rate is obtained by a time average of the actual strain rate over NYCLE solution cycles. For SRCLMT>0, the modified strain rate is capped so that during each cycle, the modified strain rate is not permitted to change more than SRCLMT multiplied by the solution time step.

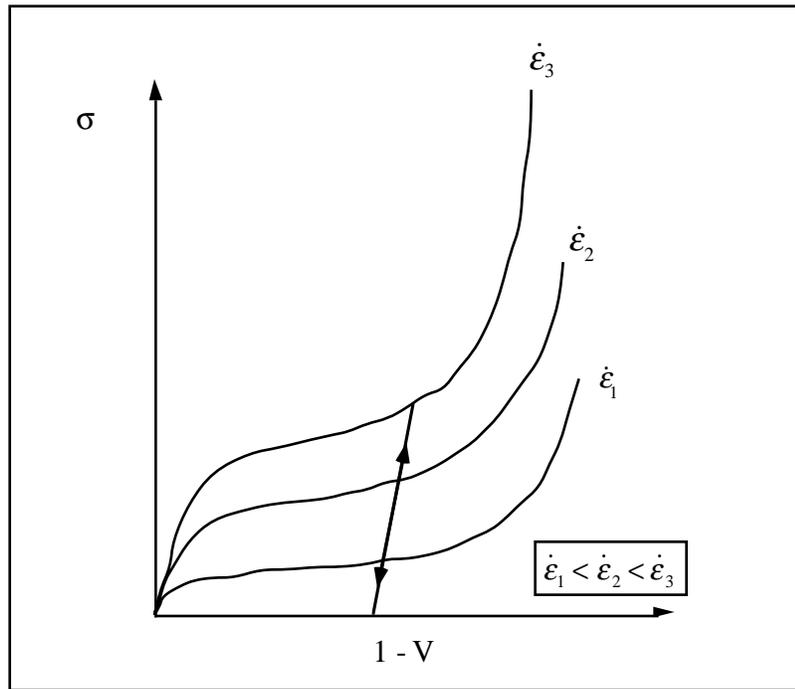


Figure 20.36. Rate effects are defined by a family of curves giving yield stress versus volumetric strain where V is the relative volume.

*MAT_QUASILINEAR_VISCOELASTIC

Purpose: This is Material Type 176. This is a quasi-linear, isotropic, viscoelastic material introduced by Fung [1993], which represents biological soft tissues such as brain, skin, kidney, spleen, etc. This model is implemented for solid elements only.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	K	LC1	LC2	N	GSTART	M
Type	I	F	F	I	I	F	F	F
Default	none	none	none	0	0	6	1/TMAX	6
Card 2								
Variable	SO							
Type	F							
Default	0.0							

Define the following 3 cards if and only if LC1 is 0.

Card 3	1	2	3	4	5	6	7	8
Variable	G1	BETA1	G2	BETA2	G3	BETA3	G4	BETA4
Type	F	F	F	F	F	F	F	F
Card 4								
Variable	G5	BETA5	G6	BETA6	G7	BETA7	G8	BETA8
Type	F	F	F	F	F	F	F	F

Card 4								
Variable	G9	BETA9	G10	BETA10	G11	BETA11	G12	BETA12
Type	F	F	F	F	F	F	F	F

Define the following card if and only if LC2 is 0.

Card 5	1	2	3	4	5	6	7	8
Variable	C1	C2	C3	C4	C5	C6		
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.
K	Bulk modulus.
LC1	Load curve ID that defines the relaxation function in shear. This curve is used to fit the coefficients G_i and $BETA_i$. If zero, define the coefficients directly. The latter is recommended.
LC2	Load curve ID that defines the instantaneous elastic reponse in shear. This curve is used to fit the coefficients C_i . If zero, define the coefficients directly. The latter is recommended.
N	Number of terms used in the Prony series, a number less than or equal to 12. This number should be equal to the number of decades of time covered by the experimental data. Define this number if LC1 is nonzero. Carefully check the fit in the D3HSP file to ensure that it is valid, since the least square fit is not always reliable.
GSTART	Starting value for least square fit. If zero, a default value is set equal to the inverse of the largest time in the experiment. Define this number if LC1 is nonzero.
M	Number of terms used to determine the instantaneous elastic reponse. Define this number if LC2 is nonzero. Carefully check the fit in the D3HSP file to ensure that it is valid, since the least square fit is not always reliable.

VARIABLE	DESCRIPTION
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.
Gi	Coefficients of the relaxation function. Define these coefficients if LC1 is set to zero. At least 2 coefficients must be nonzero.
BETAi	Decay constants of the relaxation function. Define these coefficients if LC1 is set to zero.
Ci	Coefficients of the instantaneous elastic response. Define these coefficients only if LC2 is set to zero.

The equations for this model are give as:

$$\sigma(t) = \int_0^t G(t-\tau) \frac{\partial \sigma_\varepsilon[\varepsilon(\tau)]}{\partial \varepsilon} \frac{\partial \varepsilon}{\partial \tau} d\tau$$

$$G(t) = \sum_{i=1}^n G_i e^{-\beta t}$$

$$\sigma_\varepsilon(\varepsilon) = \sum_{i=1}^k C_i \varepsilon^i$$

where G is the shear modulus. In place of the effective plastic strain in the D3PLOT database, the effective strain is output:

$$\varepsilon^{effective} = \sqrt{\frac{2}{3} \varepsilon_{ij} \varepsilon_{ij}}$$

The polynomial for instantaneous elastic response should contain only odd terms if symmetric tension-compression response is desired.

***MAT_HILL_FOAM**

Purpose: This is Material Type 177. This is a highly compressible foam based on the strain-energy function proposed by Hill [1978]; also see Storakers [1986] Poisson's ration effects are taken into account.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	K	N	MU	LCID	FITYPE	LCSR
Type	I	F	F	F	F	I	I	I
Default	none	none	none	0	0	0	0	0

Define the following 2 cards if and only if LCID is 0.

Card 2	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 3								
Variable	B1	B2	B3	B4	B5	B6	B7	B8
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.
K	Bulk modulus. This modulus is used for determining the contact interface stiffness.

<u>VARIABLE</u>	<u>DESCRIPTION</u>
N	Material constant. Define if LCID=0 below; otherwise, N is fit from the load curve data. See equations below.
MU	Damping coefficient.
LCID	Load curve ID that defines the force per unit area versus the stretch ratio. This curve can be given for either uniaxial or biaxial data depending on FITTYPE.
FITTYPE	Type of fit: EQ.1: uniaxial data, EQ.2: biaxial data.
LCSR	Load curve ID that defines the uniaxial or biaxial stress ratio (see FITTYPE) versus the transverse stretch ratio.
Ci	Material constants. See equations below. Define up to 8 coefficients if LCID=0.
Bi	Material constants. See equations below. Define up to 8 coefficients if LCID=0.

Remarks:

If load curve data is defined, the fit generated by LS-DYNA must be closely checked in the D3HSP output file. It may occur that the nonlinear least squares procedure in LS-DYNA, which is used to fit the data, is inadequate.

The Hill strain energy density function for this highly compressible foam is given by:

$$W = \sum_{j=1}^m \frac{C_j}{b_j} \left[\lambda_1^{b_j} + \lambda_2^{b_j} + \lambda_3^{b_j} - 3 + \frac{1}{n} (J^{-nb_j} - 1) \right]$$

where C_j , b_j , and n are material constants and $J = \lambda_1 \lambda_2 \lambda_3$ represents the ratio of the deformed to the undeformed state. The principal Cauchy stresses are

$$t_i = \sum_{j=1}^m \frac{C_j}{J} \left[\lambda_i^{b_j} - J^{-nb_j} \right] \quad i = 1, 2, 3$$

From the above equations the shear modulus is:

$$\mu = \frac{1}{2} \sum_{j=1}^m C_j b_j$$

and the bulk modulus is:

$$K = 2\mu\left(n + \frac{1}{3}\right)$$

The value for K defined in the input is used in the calculation of contact forces and for the material time step. Generally, this value should be equal to or greater than the K given in the above equation.

*MAT_VISCOELASTIC_HILL_FOAM

Purpose: This is Material Type 178. This is a highly compressible foam based on the strain-energy function proposed by Hill [1978]; also see Storakers [1986] The extension to include large strain viscoelasticity is due to Feng and Hallquist [2002].

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	K	N	MU	LCID	FITTYPE	LCSR
Type	I	F	F	F	F	I	I	I
Default	none	none	none	0	0	0	0	0
Card 2								
Variable	LCVE	NT	GSTART					
Type	I	F	F					
Default	0	6	1/TMAX					

Define the following 2 cards if and only if LCID is 0.

Card 3	1	2	3	4	5	6	7	8
Variable	C1	C2	C3	C4	C5	C6	C7	C8
Type	F	F	F	F	F	F	F	F
Card 4								
Variable	B1	B2	B3	B4	B5	B6	B7	B8
Type	F	F	F	F	F	F	F	F

Card Format for Viscoelastic Constants. Up to 12 cards may be input. A keyword card (with a "*" in column 1) terminates this input if less than 12 cards are used.

Optional Cards 1 2 3 4 5 6 7 8

Variable	GI	BETAI						
Type	F	F						

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
K	Bulk modulus. This modulus is used for determining the contact interface stiffness.
N	Material constant. Define if LCID=0 below; otherwise, N is fit from the load curve data. See equations below.
MU	Damping coefficient.
LCID	Load curve ID that defines the force per unit area versus the stretch ratio. This curve can be given for either uniaxial or biaxial data depending on FITTYPE. Load curve LCSR below must also be defined.
FITTYPE	Type of fit: EQ.1: uniaxial data, EQ.2: biaxial data.
LCSR	Load curve ID that defines the uniaxial or biaxial stress ratio (see FITTYPE) versus the transverse stretch ratio.
LCVE	Optional load curve ID that defines the relaxation function in shear. This curve is used to fit the coefficients G_i and $BETA_i$. If zero, define the coefficients directly. The latter is recommended.
NT	Number of terms used to fit the Prony series, which is a number less than or equal to 12. This number should be equal to the number of decades of time covered by the experimental data. Define this number if LCVE is nonzero. Carefully check the fit in the D3HSP file to ensure that it is valid, since the least square fit is not always reliable.
GSTART	Starting value for least square fit. If zero, a default value is set equal to the inverse of the largest time in the experiment. Define this number if LC1 is nonzero, C_i , Material constants. See equations below. Define up to 8 coefficients.

<u>VARIABLE</u>	<u>DESCRIPTION</u>
Ci	Material constants. See equations below. Define up to 8 coefficients if LCID=0.
Bi	Material constants. See equations below. Define up to 8 coefficients if LCID=0.
GI	Optional shear relaxation modulus for the ith term
BETAI	Optional decay constant if ith term

Remarks:

If load curve data is defined, the fit generated by LS-DYNA must be closely checked in the D3HSP output file. It may occur that the nonlinear least squares procedure in LS-DYNA, which is used to fit the data, is inadequate.

The Hill strain energy density function for this highly compressible foam is given by:

$$W = \sum_{j=1}^m \frac{C_j}{b_j} \left[\lambda_1^{b_j} + \lambda_2^{b_j} + \lambda_3^{b_j} - 3 + \frac{1}{n} (J^{-nb_j} - 1) \right]$$

where C_j , b_j , and n are material constants and $J = \lambda_1 \lambda_2 \lambda_3$ represents the ratio of the deformed to the undeformed state. The principal Cauchy stresses are

$$t_i = \sum_{j=1}^m \frac{C_j}{J} \left[\lambda_i^{b_j} - J^{-nb_j} \right] \quad i = 1, 2, 3$$

From the above equations the shear modulus is:

$$\mu = \frac{1}{2} \sum_{j=1}^m C_j b_j$$

and the bulk modulus is:

$$K = 2\mu \left(n + \frac{1}{3} \right)$$

The value for K defined in the input is used in the calculation of contact forces and for the material time step. Generally, this value should be equal to or greater than the K given in the above equation.

Rate effects are taken into account through linear viscoelasticity by a convolution integral of the form:

$$\sigma_{ij} = \int_0^t g_{ijkl}(t - \tau) \frac{\partial \epsilon_{kl}}{\partial \tau} d\tau$$

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:

$$g(t) = \alpha_0 + \sum_{m=1}^N \alpha_m e^{-\beta_m t}$$

given by,

$$g(t) = \sum_{i=1}^n G_i e^{-\beta_i t}$$

This model is effectively a Maxwell fluid which consists of a dampers and springs in series. We characterize this in the input by shear moduli, G_i , and decay constants, β_i . The viscoelastic behavior is optional and an arbitrary number of terms may be used.

*MAT_LOW_DENSITY_SYNTHETIC_FOAM_{OPTION}

This is Material Type 179 (and 180 if the ORHTO option below is active) for modeling rate independent low density foams, which have the property that the hysteresis in the loading-unloading curve is considerably reduced after the first loading cycle. In this material we assume that the loading-unloading curve is identical after the first cycle of loading is completed and that the damage is isotropic, i.e., the behavior after the first cycle of loading in the orthogonal directions also follows the second curve. The main application at this time is to model the observed behavior in the synthetic foams that are used in some bumper designs.

Options include:

<BLANK>

ORTHO

If the foam develops orthotropic behavior, i.e., after the first loading and unloading cycle the material in the orthogonal directions are unaffected then the ORTHO option should be used. If the ORTHO option is active the directionality of the loading is stored. This option is requires additional storage to store the history variables related to the orthogonality and is slightly more expensive.

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	MID	RO	E	LCID1	LCID2	HU	BETA	DAMP
Type	I	F	F	F	F	F	F	F
Default						1.		0.05
Remarks						3	1	

Card 2

Variable	SHAPE	FAIL	BVFLAG	ED	BETA1	KCON	REF	TC
Type	F	F	F	F	F	F	F	F
Default	1.0	0.0	0.0	0.0	0.0	0.0	0.0	1.E+20
Remarks	3		2	5	5	6		

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MID	Material identification. A unique number has to be chosen.
RO	Mass density
E	Young's modulus
LCID1	Load curve ID, see *DEFINE_CURVE, for nominal stress versus strain for the first loading cycle.
LCID2	Load curve ID, see *DEFINE_CURVE, for nominal stress versus strain for the loading cycles after the first loading cycle is completed. TC Tension cut-off stress
HU	Hysteretic unloading factor between 0 and 1 (default=1, i.e., no energy dissipation), see also Figure 20.16.
BETA	β , decay constant to model creep in unloading
DAMP	Viscous coefficient (.05< recommended value <.50) to model damping effects. LT.0.0: DAMP is the load curve ID, which defines the damping constant as a function of the maximum strain in compression defined as: $\epsilon_{\max} = \max(1 - \lambda_1, 1 - \lambda_2, 1 - \lambda_3).$ In tension, the damping constant is set to the value corresponding to the strain at 0. The abscissa should be defined from 0 to 1.
SHAPE	Shape factor for unloading. Active for nonzero values of the hysteretic unloading factor. Values less than one reduces the energy dissipation and greater than one increases dissipation, see also Figure 20.16.
FAIL	Failure option after cutoff stress is reached: EQ.0.0: tensile stress remains at cut-off value, EQ.1.0: tensile stress is reset to zero.
BVFLAG	Bulk viscosity activation flag, see remark below: EQ.0.0: no bulk viscosity (recommended), EQ.1.0: bulk viscosity active.
ED	Optional Young's relaxation modulus, E_d , for rate effects. See comments below.
BETA1	Optional decay constant, β_1 .

VARIABLE	DESCRIPTION
KCON	Stiffness coefficient for contact interface stiffness. If undefined the maximum slope in stress vs. strain curve is used. When the maximum slope is taken for the contact, the time step size for this material is reduced for stability. In some cases Δt may be significantly smaller, and defining a reasonable stiffness is recommended.
REF	Use reference geometry to initialize the stress tensor. The reference 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:

This model is based on *MAT_LOW_DENSITY_FOAM. The uniaxial response is shown below with a large shape factor and small hysteretic factor. If the shape factor is not used, the unloading will occur on the loading curve for the second and subsequent cycles.

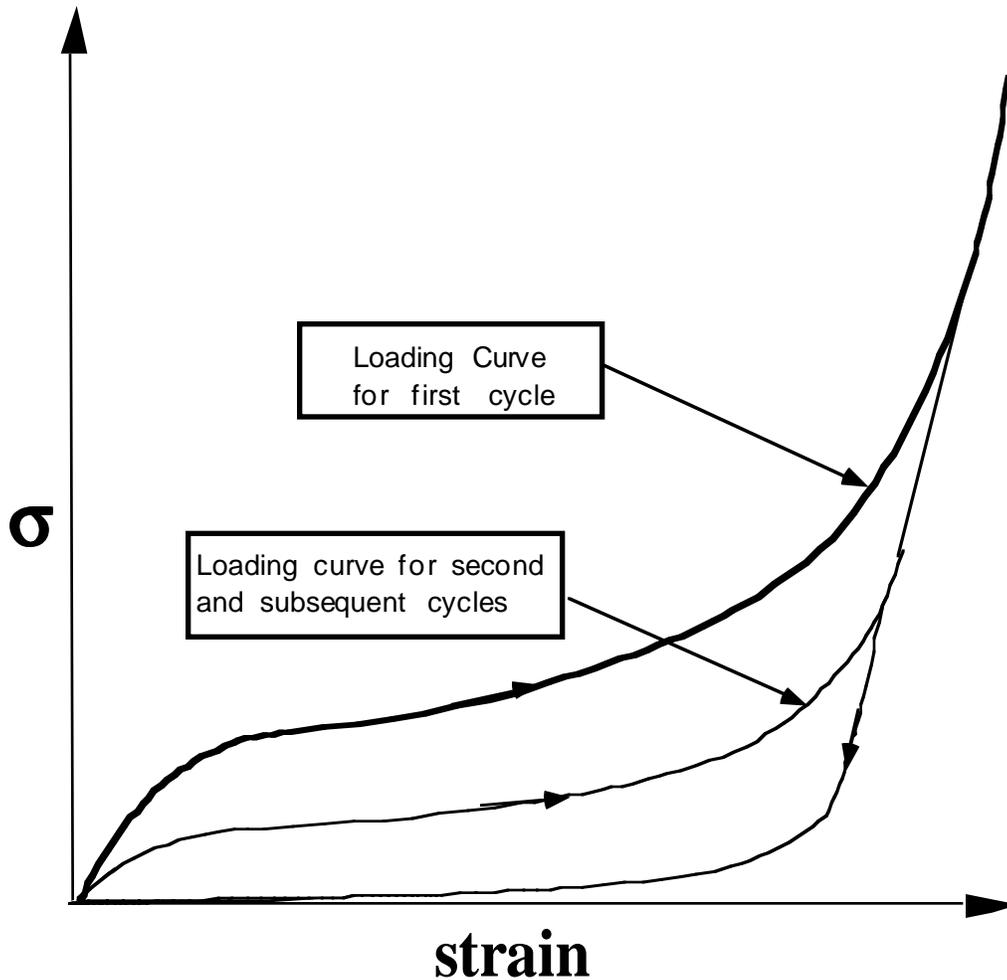


Figure 20.35. Loading and reloading curves.

***MAT_SIMPLIFIED_RUBBER**

This is Material Type 181. This material model provides a rubber model defined by a single uniaxial load curve or by a family of curves at discrete strain rates.

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	MID	RO	K	MU	G	SIGF		
Type	I	F	F	F	F	F		

Card 2

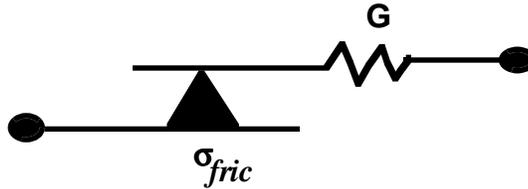
Variable	SGL	SW	ST	LC/TBID	TENSION	RTYPE	AVGOPT	
Type	F	F	F	F	F	F	F	

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MID	Material identification. A unique number has to be chosen.
RO	Mass density
K	Linear bulk modulus.
MU	Damping coefficient.
G	Shear modulus for frequency independent damping. Frequency independent damping is based of a spring and slider in series. The critical stress for the slider mechanism is SIGF defined below. For the best results, the value of G should be 250-1000 times greater than SIGF.
SIGF	Limit stress for frequency independent, frictional, damping.
SGL	Specimen gauge length
SW	Specimen width
ST	Specimen thickness

VARIABLE	DESCRIPTION
LC/TBID	Load curve or table ID, see *DEFINE_TABLE, defining the force versus actual change in the gauge length. If the table definition is used a family of curves are defined for discrete strain rates. The load curves should cover the complete range of expected loading, i.e., the smallest stretch ratio to the largest.
TENSION	Parameter that controls how the rate effects are treated. Applicable to the table definition. EQ.-1.0: rate effects are considered for loading either in tension or compression, but not for unloading, EQ. 0.0: rate effects are considered for compressive loading only, EQ. 1.0: rate effects are treated identically in tension and compression.
RTYPE	Strain rate type if a table is defined: EQ.0.0: true strain rate, EQ.1.0: engineering strain rate
AVGOPT	Averaging option determine strain rate to reduce numerical noise. EQ.0.0: simple average of twelve time steps, EQ.1.0: running 12 point average.

Remarks:

The frequency independent damping is obtained by the having a spring and slider in series as shown in the following sketch:



***MAT_SEISMIC_BEAM**

Purpose: This is Material Type 191. This material enables lumped plasticity to be developed at the 'node 2' end of Belytschko-Schwer beams (resultant formulation). The plastic yield surface allows interaction between the two moments and the axial force.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	AOPT	FTYPE		
Type	I	F	F	F	F	I		
Default	none	none	none	none	0.1	1		

Card 2	1	2	3	4	5	6	7	8
Variable	LCPMS	SFS	LCPMT	SFT	LCAT	SFAT	LCAC	SFAC
Type	F	F	F	F	F	F	F	F
Default	none	1.0	LCMPS	1.0	none	1.0	LCAT	1.0

Define the following card for interaction formulation, FTYPE, type 1 (Default)

Card 3	1	2	3	4	5	6	7	8
Variable	ALPHA	BETA	GAMMA	DELTA	A	B		
Type	F	F	F	F	F	F		
Default	see note							

Define the following card for interaction formulation, FTYPE, type 2

Card 3	1	2	3	4	5	6	7	8
Variable	SIGY	D	W	TF	TW			
Type	F	F	F	F	F			
Default	none	none	none	none	none			

<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.
AOPT	Axial force option EQ:0.0 Axial load curves are collapse load vs. Strain NE:0.0 Axial load curves are collapse load vs. Change in length
FTYPE	Formulation type for interaction EQ:1 Parabolic coefficients, axial load and biaxial bending (default). EQ:2 Japanese code, axial force and major axis bending.
LCPMS	Load curve ID giving plastic moment vs. Plastic rotation at node 2 about local s-axis. See *DEFINE_CURVE.
SFS	Scale factor on s-moment at node 2.
LCPMT	Load curve ID giving plastic moment vs. Plastic rotation at node 2 about local t-axis. See *DEFINE_CURVE.
SFT	Scale factor on t-moment at node 2.
LCAT	Load curve ID giving axial tensile yield force vs. total tensile (elastic + plastic) strain or vs. elongation. See AOPT above. All values are positive. See *DEFINE_CURVE.
SFAT	Scale factor on axial tensile force.

<u>VARIABLE</u>	<u>DESCRIPTION</u>
LCAC	Load curve ID giving compressive yield force vs. total compressive (elastic + plastic) strain or vs. elongation. See AOPT above. All values are positive. See *DEFINE_CURVE.
SFAC	Scale factor on axial tensile force.
ALPHA	Parameter to define yield surface.
BETA	Parameter to define yield surface.
GAMMA	Parameter to define yield surface.
DELTA	Parameter to define yield surface.
A	Parameter to define yield surface.
B	Parameter to define yield surface.
SIGY	Yield stress of material.
D	Depth of section used to calculate interaction curve.
W	Width of section used to calculate interaction curve.
TF	Flange thickness of section used to calculate interaction curve.
TW	Web thickness used to calculate interaction curve.

Note:

For formulation type 1, if ALPHA, BETA, GAMMA, DELTA A and B are all set to zero then the following default values are used:

ALPHA	=	2.0
BETA	=	2.0
GAMMA	=	2.0
DELTA	=	4.0
A	=	2.0
B	=	-1.0

*MAT_SOIL_BRICK

Purpose: This is Material Type 192. This material enables enables clay type soil materials to be modelled accurately, although there will be some penalty of cpu cost. The stress history of the soil is calculated prior to the initialization of the model. This data is written to a temporary file which is deleted soon after the initalization is complete.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	RLAMDA	RKAPPA	RIOTA	RBETA1	RBETA2	RMU
Type	I	F	F	F	F	F	F	F
Default								1.0

Card 2	1	2	3	4	5	6	7	8
Variable	RNU	RLCID	TOL	PGCL	SUB-INC	BLK	GRAV	
Type	F	F	F	F	F	F		
Default			0.0005				9.807	

VARIABLE

DESCRIPTION

- MID Material identification number, must be unique
- RO Mass density
- RLAMDA Material coefficient
- RKAPPA Material coefficient
- RIOTA Material coefficient
- RBETA1 Material coefficient
- RBETA2 Material coefficient

<u>VARIABLE</u>	<u>DESCRIPTION</u>
RMU	Shape factor coefficient. This parameter will modify the shape of the yield surface used. 1.0 implies a von mises type surface, but 1.1 to 1.25 is more indicative of soils. The default value is 1.0.
RNU	Poisson's ratio
RLCID	Loadcurve identification number referring to a curve defining upto 10 pairs of 'string-length' vs G / Gmax points.
TOL	User defined tolerance for convergence checking. Default value is set to 0.02.
PGCL	Pre-consolidation ground level. This parameter defines the maximum surface level (relative to $z = 0.0$ in the model) of the soil throughout geological history. This is used calculate the maximum over burden pressure on the soil elements.
SUB-INC	User defined strain increment size. This is the maximum strain increment that the material model can normally cope with. If the value is exceeded a warning is echoed to the d3hsp file.
BLK	The elastic bulk stiffness of the soil. This is used for the contact stiffness only.
GRAV	The gravitational acceleration. This is used to calculate the element stresses due the overlying soil. Default is set to 9.807 m/s^2 .

Remarks:

1. This material type requires that the model is oriented such that the z-axis is defined in the upward direction. The key parameters are defined such that may vary with depth (i.e. the z-axis).
2. The shape factor for a typical soil would be 1.25, but should not be pushed further than 1.35

***MAT_DRUCKER_PRAGER**

Purpose: This is Material Type 193. This material enables enables soil to be modelled effectively. The parameters used to define the yield surface are familiar geotechnical parameters (i.e. angle of friction). The modified Drucker-Prager yield surface is used in this material model enabling the shape of the surface to be distorted into a more realistic definition for soils.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	GMOD	RNU	RKF	PHI	CVAL	PSI
Type	I	F	F	F	F	F	F	F
Default					1.0			0.0

Card 2	1	2	3	4	5	6	7	8
Variable	STR_LIM							
Type	F							
Default	0.005							

Card 3	1	2	3	4	5	6	7	8
Variable	GMODDP	PHIDP	CVALDP	PSIDP	GMODGR	PHIGR	CVALGR	PSIGR
Type	F	F	F	F	F	F	F	F
Default	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MID	Material identification number, must be unique.
RO	Mass density
GMOD	Elastic shear modulus
RNU	Poisson's ratio
RKF	Failure surface shape parameter
PHI	Angle of friction (radians)
CVAL	Cohesion value
PSI	Dilation angle (radians)
STR_LIM	Minimum shear strength of material is given by STR_LIM*CVAL
GMODDP	Depth at which shear modulus (GMOD) is correct
PHIDP	Depth at which angle of friction (PHI) is correct
CVALDP	Depth at which cohesion value (CVAL) is correct
PSIDP	Depth at which dilation angle (PSI) is correct
GMODGR	Gradient at which shear modulus (GMOD) increases with depth
PHIGR	Gradient at which friction angle (PHI) increases with depth
CVALGR	Gradient at which cohesion value (CVAL) increases with depth
PSIGR	Gradient at which dilation angle (PSI) increases with depth

Remarks:

1. This material type requires that the model is oriented such that the z-axis is defined in the upward direction. The key parameters are defined such that may vary with depth (i.e. the z-axis)
2. The shape factor for a typical soil would be 0.8, but should not be pushed further than 0.75.
3. If STR_LIM is set to less than 0.005, the value is reset to 0.005.

***MAT_RC_SHEAR_WALL**

Purpose: This is Material Type 194. It is for shell elements only. It uses empirically-derived algorithms to model the effect of cyclic shear loading on reinforced concrete walls. It is primarily intended for modelling squat shear walls, but can also be used for slabs. Because the combined effect of concrete and reinforcement is included in the empirical data, crude meshes can be used. The model has been designed such that the minimum amount of input is needed: generally, only the variables on the first card need to be defined.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	TMAX			I
Type	I	F	F	F	F			
Default	none	none	none	0.0	0.0			

Define the following data if “Uniform Building Code” formula for maximum shear strength or tensile cracking are required – otherwise leave blank.

Card 2	1	2	3	4	5	6	7	8
Variable	Fc'	PREF	FYIELD	SIG0	UNCONV	ALPHA	FT	ERIENF
Type	F	F	F	F	F	F	F	F
Default	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0
Card 3	1	2	3	4	5	6	7	8
Variable	A	B	C	D	E	F		
Type	F	F	F	F	F	F		
Default	0.05	0.55	0.125	0.66	0.25	1.0		

Card 4	1	2	3	4	5	6	7	8
Variable	Y1	Y2	Y3	Y4	Y5			I
Type	F	F	F	F	F			
Default	0.0	0.0	0.0	0.0	0.0			
Card 5	1	2	3	4	5	6	7	8
Variable	T1	T2	T3	T4	T5			
Type	F	F	F	F	F			
Default	0.0	0.0	0.0	0.0	0.0			
Card 6	1	2	3	4	5	6	7	8
Variable	AOPT							
Type	F							
Default	0.0							
Card 7	1	2	3	4	5	6	7	8
Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		
Default	0.0	0.0	0.0	0.0	0.0	0.0		

Card 8	1	2	3	4	5	6	7	8
Variable	V1	V2	V3	D1	D2	D3	BETA	
Type	F	F	F	F	F	F	F	
Default	0.0	0.0	0.0	0.0	0.0	0.0	0.0	

VARIABLE**DESCRIPTION**

MID	Material identification, A unique number must be used.
RO	mass density
E	Young=s Modulus
PR	Poisson=s Ratio
TMAX	Ultimate shear stress. If set to zero, LS-DYNA will calculate tmax based on the formulae in the Universal Building Code, using the data on card 2. See notes below.
Fc=	Unconfined Compressive Strength of concrete (used in the calculation of ultimate shear stress; crushing behaviour is not modelled)
PREF	Percent reinforcement, e.g if 1.2% reinforcement, enter 1.2
FYIELD	Yield stress of reinforcement
SIG0	Overburden stress (in-plane compressive stress) - used in the calculation of ultimate shear stress. Usually sig0 is left as zero.
UCONV	Unit conversion factor. $UCONV = \sqrt{1\text{psi}}$ in the model stress units). This is needed because the ultimate tensile stress of concrete is expressed as $\sqrt{f_c=}$ where $f_c=$ is in psi. Therefore a unit conversion factor of $\sqrt{\text{psi}/\text{stress unit}}$ is required. Examples: $UCONV = 0.083$ if stress unit is MN/m ² or N/mm ² $UCONV = 83.3$ if stress unit is N/m ²
ALPHA	Shear span factor - see below.
FT	Cracking stress in direct tension - see notes below. Default is 8% of the cylinder strength.

VARIABLE	DESCRIPTION
ERIENF	Youngs modulus of reinforcement. Used in calculation of post-cracked stiffness - see notes below.
A	Hysteresis constants determining the shape of the hysteresis loops.
B	Hysteresis constants determining the shape of the hysteresis loops.
C	Hysteresis constants determining the shape of the hysteresis loops.
D	Hysteresis constants determining the shape of the hysteresis loops.
E	Hysteresis constants determining the shape of the hysteresis loops.
F	Strength degradation factor. After the ultimate shear stress has been achieved, F multiplies the maximum shear stress from the curve for subsequent reloading. F=1.0 implies no strength degradation (default). F=0.5 implies that the strength is halved for subsequent reloading.
Y1,Y2...Y5	Shear strain points on stress-strain curve. By default these are calculated from the values on card 1. See below for more guidance.
T1,T2...T5	Shear stress points on stress-strain curve. By default these are calculated from the values on card 1. See below for more guidance.
AOPT	Material axes option: EQ. 0.0: locally orthotropic with material axes determined by element nodes as shown in Figure 19.1. Nodes 1, 2, and 4 of an element are identical to the Nodes used for the definition of a coordinate system as by *DEFINE_COORDINATE_NODES. EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR. EQ. 3.0: applicable to shell elements only. This option determines locally orthotropic material axes by offsetting the material axes by an angle to be specified from a line in the plane of the shell determined by taking the cross product of the vector v defined below with the shell normal vector.
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.

The element is linear elastic except for in-plane shear and tensile cracking effects. Crushing due to direct compressive stresses are modelled only insofar as there is an in-plane shear stress component. It is not recommended that this model be used where nonlinear response to direct compressive or loads is important.

Note that the in-plane shear stress is defined as the shear stress in the element's local x-y plane (txy). This is not necessarily equal to the maximum shear stress in the plane: for example, if the principal stresses are at 45 degrees to the local axes, txy is zero. Therefore it is important to ensure that the local axes are appropriate - for a shear wall the local axes should be vertical or horizontal. By default, local X points from node 1 to node 2 of the element. It is possible to change the local axes by using AOPT>0.

If TMAX is set to zero, the ultimate shear stress is calculated using a formula in the Uniform Building Code 1997, section 1921.6.5:

$$\text{max shear stress (UBC)} = V_u/A_{cv} = u_{\text{conv}} \cdot \alpha \cdot \sqrt{f_c} + \rho \cdot f_y$$

where,

uconv = unit conversion factor, 0.083 for SI units (MN)

Alpha = aspect ratio, = 2.0 unless ratio h/l < 2.0 in which case alpha varies linearly from 2.0 at h/l=2.0 to 3.0 at h/l=1.5.

Fc = unconfined compressive strength of concrete

ro = fraction of reinforcement = percent reinforcement/100

fy = yield stress of reinforcement

To this we add shear stress due to the overburden to obtain the ultimate shear stress:

$$t_{\text{max}} = \text{max shear stress (UBC)} + \text{sig0}$$

where

sig0 = in-plane compressive stress under static equilibrium conditions

The UBC formula for ultimate shear stress is generally conservative (predicts that the wall is weaker than shown in test), sometimes by 50% or more. A less conservative formula is that of Fukuzawa:

$$t_{\text{max}} = a_1 \cdot 2.7 \cdot (1.9 - M/LV) \cdot U_{\text{CONV}} \cdot \sqrt{f_c} + \rho \cdot f_y \cdot 0.5 + \text{sig0}$$

where

a1 = max((0.4 + Ac/Aw), 1.0)

Ac/Aw = ratio of area of supporting columns/flanges etc to area of wall

M/LV = Aspect ratio of wall (height/length)

Other terms are as above. This formula is not included in the material model: tmax should be calculated by hand and entered on Card 1 if the Fukuzawa formula is required.

It should be noted that none of the available formulae, including Fuzukawa, predict the ultimate shear stress accurately for all situations. Variance from the experimental results can be as great as 50%.

The shear stress vs shear strain curve is then constructed as follows, using the algorithm of Fukuzawa extended by Arup:

Assume ultimate shear strain (yu) = 0.0048

First point on curve (concrete cracking) at $0.3t_u$, $\text{strain}=0.3t_u/G$ where G is the elastic shear modulus given by $E/2(1+\nu)$

Second point (reinforcement yield) at $0.5y_u$, $0.8t_{\text{max}}$

Third point (ultimate strength) at y_u , t_{max}

Fourth point (onset of strength reduction) at $2y_u$, t_{max}

Fifth point (failure) at $3y_u$, $0.6t_{\text{max}}$.

After failure, the shear stress drops to zero. The curve points can be entered by the user if desired, in which case they over-ride the automatically calculated curve. However, it is anticipated that in most cases the default curve will be preferred due to ease of input.

Hysteresis follows the algorithm of Shiga as for the squat shearwall spring (see *MAT_SPRING_SQUAT_SHEARWALL). The hysteresis constants A,B,C,D,E can be entered by the user if desired but it is generally recommended that the default values be used.

Cracking in tension is checked for the local x and y directions only. A trilinear response is assumed, with turning points at concrete cracking and reinforcement yielding. The three regimes are:

1. Pre-cracking, linear elastic response is assumed using the overall Young's modulus on Card 1.
2. Cracking occurs in the local x or y directions when the stress in that direction exceeds f_t (by default, this is set to 8% of the cylinder strength). Post-cracking, a linear stress-strain response is assumed up to reinforcement yield (defined by reinforcement yield stress divided by reinforcement Young's Modulus).
3. Post-yield, a constant stress is assumed (no work hardening).

Unloading returns to the origin of the stress-strain curve.

For compressive strains the response is always linear elastic using the overall Young's modulus on Card 1.

If insufficient data is entered, no cracking occurs in the model. As a minimum, f_c , % and f_y are needed.

*MAT_CONCRETE_BEAM

This is Material Type 195 for beam elements. An elasto-plastic material with an arbitrary stress versus strain curve and arbitrary strain rate dependency can be defined. See also Remark below. Also, failure based on a plastic strain or a minimum time step size can be defined.

Card 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	10.E+20
Card 2								
Variable	C	P	LCSS	LCSR				
Type	F	F	F	F				
Default	0	0	0	0				
Card 3								
Variable	NOTEN	TENCUT	SDR					
Type	I	F	F					
Default	0	E15.0	0.0					

<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 19.5. The stress versus effective plastic strain curve for the lowest value of strain rate is used if the strain rate falls below the minimum value. Likewise, the stress versus effective plastic strain curve for the highest value of strain rate is used if the strain rate exceeds the maximum value. The strain rate parameters: C and P;
LCSR	Load curve ID defining strain rate scaling effect on yield stress.
NOTEN	No-tension flag, EQ:0 beam takes tension, EQ:1 beam takes no tension, EQ:2 beam takes tension up to value given by TENCUT.
TENCUT	Tension cutoff value.
SDR	Stiffness degradation factor.

The stress strain behaviour may be treated by a bilinear stress strain curve by defining the tangent modulus, ETAN. An effective stress versus effective plastic strain curve (LCSS) may be input instead of defining ETAN. The cost is roughly the same for either approach. The most general approach is to use the table definition (LCSS) discussed below.

Three options to account for strain rate effects are possible.

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

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

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.

***MAT_GENERAL_SPRING_DISCRETE_BEAM**

This is Material Type 196. This model permits elastic and elastoplastic springs with damping to be represented with a discrete beam element type6 by using six springs each acting about one of the six local degrees-of-freedom. For elastic behavior, a load curve defines force or moment versus displacement or rotation. For inelastic behavior, a load curve yield force or moment versus plastic deflection or rotation, which can vary in tension and compression. The two nodes defining a beam may be coincident to give a zero length beam, or offset to give a finite length beam. For finite length discrete beams the absolute value of the variable SCOOR in the SECTION_BEAM input should be set to a value of 2.0, which causes the local r-axis to be aligned along the two nodes of the beam to give physically correct behavior. The distance between the nodes of a beam should not affect the behavior of this material model. A triad is used to orient the beam for the directional springs.

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	MID	RO						
Type	I	F						

Define the following cards, 2 and 3, for each active degree of freedom. This data is terminated by the next "*" card or when all six degrees-of-freedom are defined.

Card 2 1 2 3 4 5 6 7 8

Variable	DOF	TYPE	K	D	CDF	TDF		
Type	I	I	F	F	F	F		

Card 2 1 2 3 4 5 6 7 8

Variable	FLCID	HLCID	C1	C2	DLE	GLCID		
Type	F	F	F	F	F	I		

<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 definition.
DOF	Active degree-of-freedom, a number between 1 and 6 inclusive. Each value of DOF can only be used once. The active degree-of-freedom is measured in the local coordinate system for the discrete beam element.
TYPE	The default behavior is elastic. For inelastic behavior input 1.
K	Elastic loading/unloading stiffness. This is required input for inelastic behavior.
D	Optional viscous damping coefficient.
CDF	Compressive displacement at failure. Input as a positive number. After failure, no forces are carried. This option does not apply to zero length springs. EQ.0.0: inactive.
TDF	Tensile displacement at failure. After failure, no forces are carried. EQ.0.0: inactive.
FLCID	Load curve ID, see *DEFINE_CURVE. For option TYPE=0, this curve defines force or moment versus deflection for nonlinear elastic behavior. For option TYPE=1, this curve defines the yield force versus plastic deflection. If the origin of the curve is at (0,0) the force magnitude is identical in tension and compression, i.e., only the sign changes. If not, the yield stress in the compression is used when the spring force is negative. The plastic displacement increases monotonically in this implementation. The load curve is required input.
HLCID	Load curve ID, see *DEFINE_CURVE, defining force versus relative velocity (Optional). If the origin of the curve is at (0,0) the force magnitude is identical for a given magnitude of the relative velocity, i.e., only the sign changes.
C1	Damping coefficient.
C2	Damping coefficient
DLE	Factor to scale time units.
GLCID	Optional load curve ID, see *DEFINE_CURVE, defining a scale factor versus deflection for load curve ID, HLCID. If zero, a scale factor of unity is assumed.

Remarks:

If TYPE=0, elastic behavior is obtained. In this case, if the linear spring stiffness is used, the force, F , is given by:

$$F = F_0 + K\Delta L + D\Delta\dot{L}$$

but if the load curve ID is specified, the force is then given by:

$$F = F_0 + K f(\Delta L) \left[1 + C1 \cdot \Delta\dot{L} + C2 \cdot \text{sgn}(\Delta\dot{L}) \ln \left(\max \left\{ 1, \frac{|\Delta\dot{L}|}{DLE} \right\} \right) \right] + D\Delta\dot{L} + g(\Delta L)h(\Delta\dot{L})$$

In these equations, ΔL is the change in length

$$\Delta L = \text{current length} - \text{initial length}$$

If TYPE=1, inelastic behavior is obtained. In this case, the yield force is taken from the load curve:

$$F^Y = F_y(\Delta L^{plastic})$$

where $L^{plastic}$ is the plastic deflection. A trial force is computed as:

$$F^T = F^n + K\Delta\dot{L}(\Delta t)$$

and is checked against the yield force to determine F :

$$F = \begin{cases} F^Y & \text{if } F^T > F^Y \\ F^T & \text{if } F^T \leq F^Y \end{cases}$$

The final force, which includes rate effects and damping, is given by:

$$F^{n+1} = F \cdot \left[1 + C1 \cdot \Delta\dot{L} + C2 \cdot \text{sgn}(\Delta\dot{L}) \ln \left(\max \left\{ 1, \frac{|\Delta\dot{L}|}{DLE} \right\} \right) \right] + D\Delta\dot{L} + g(\Delta L)h(\Delta\dot{L})$$

Unless the origin of the curve starts at (0,0), the negative part of the curve is used when the spring force is negative where the negative of the plastic displacement is used to interpolate, F_y . The positive part of the curve is used whenever the force is positive.

The cross sectional area is defined on the section card for the discrete beam elements, See *SECTION_BEAM. The square root of this area is used as the contact thickness offset if these elements are included in the contact treatment.

*MAT_SEISMIC_ISOLATOR

This is Material Type 197 for discrete beam elements. Sliding and elastomeric seismic isolation bearings can be modelled, applying bi-directional coupled plasticity theory. The hysteretic behaviour was proposed by Wen [1976] and Park, Wen, and Ang [1986]. The sliding bearing behaviour is recommended by Zayas, Low and Mahin [1990]. The algorithm used for implementation was presented by Nagarajaiah, Reinhorn, and Constantinou [1991]. Element formulation type 6 must be used. Local axes are defined on *SECTION_BEAM; the default is the global axis system. It is expected that the local z-axis will be vertical.

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	MID	RO	A	GAMMA	BETA	DISPY	STIFFV	ITYPE
Type	I	F	F	F	F	F	F	I
Default	none	None	1.0	0.5	0.5	0.0	0.0	0.0

Card 2

Variable	PRELOAD	DAMP	MX1	MX2	MY1	MY2		
Type	F	F	F	F	F	F		
Default	0	1.0	0	0	0	0		

Card 3 for sliding isolator - leave blank for elastomeric isolator:

Card 3 1 2 3 4 5 6 7 8

Variable	FMAX	DELF	AFRIC	RADX	RADY	RADB	STIFFL	STIFFTS
Type	F	F	F	F	F	F	F	F
Default	0	0	0	1.0e20	1.0e20	1.0e20	=STIFFV	0

Card 4 for elastomeric isolator - leave blank for sliding isolator:

Card 4 1 2 3 4 5 6 7 8

Variable	FORCEY	ALPHA	STIFFT	DFAIL				
Type	F	F	F	F				
Default	0	0	0.5STIFF V	1.0e20				

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
A	Nondimensional variable - see below
GAMMA	Nondimensional variable - see below
BETA	Nondimensional variable - see below
DISPY	Yield displacement (length units - must be > 0.0)
STIFFV	Vertical stiffness (force/length units)
ITYPE	Type: 0=sliding 1=elastomeric
PRELOAD	Vertical preload not explicitly modelled (force units)
DAMP	Damping ratio (nondimensional)
MX1, MX2	Moment factor at ends 1 and 2 in local X-direction
MY1, MY2	Moment factor at ends 1 and 2 in local Y-direction
FMAX (*)	Maximum friction coefficient (dynamic)
DELF (*)	Difference between maximum friction and static friction coefficient
AFRIC (*)	Velocity multiplier in sliding friction equation (time/length units)
RADX (*)	Radius for sliding in local X direction

<u>VARIABLE</u>	<u>DESCRIPTION</u>
RADY (*)	Radius for sliding in local Y direction
RADB (*)	Radius of retaining ring
STIFFL (*)	Stiffness for lateral contact against the retaining ring
STIFFTS (*)	Stiffness for tensile vertical response (sliding isolator - default = 0)
FORCEY (+)	Yield force
ALPHA (+)	Ratio of postyielding stiffness to preyielding stiffness
STIFFT (+)	Stiffness for tensile vertical response (elastomeric isolator)
DFAIL (+)	Lateral displacement at which the isolator fails

(*) - Used for sliding type. Leave blank for elastomeric type

(+) - Used for elastomeric type. Leave blank for sliding type

Remarks:

The horizontal behaviour of both types is governed by plastic history variables Z_x , Z_y that evolve according to equations given in the reference; A , γ and β and the yield displacement are the input parameters for this. The intention is to provide smooth build-up, rotation and reversal of forces in response to bidirectional displacement histories in the horizontal plane. The theoretical model has been correlated to experiments on seismic isolators.

The RADX, RADY inputs for the sliding isolator are optional. If left blank, the sliding surface is assumed to be flat. A cylindrical surface is obtained by defining either RADX or RADY; a spherical surface can be defined by setting RADX=RADY. The effect of the curved surface is to add a restoring force proportional to the horizontal displacement from the centre. As seen in elevation, the top of the isolator will follow a curved trajectory, lifting as it displaces away from the centre.

The vertical behaviour for the elastomeric type is linear elastic; in the case of uplift, the tensile stiffness will be different to the compressive stiffness. For the sliding type, compression is treated as linear elastic but no tension can be carried.

Vertical preload can be modelled either explicitly (for example, by defining gravity), or by using the PRELOAD input. PRELOAD does not lead to any application of vertical force to the model. It is added to the compression in the element before calculating the friction force and tensile/compressive vertical behaviour.

DAMP is the fraction of critical damping for free vertical vibration of the isolator, based on the mass of the isolator (including any attached lumped masses) and its vertical stiffness. The viscosity is reduced automatically if it would otherwise infringe numerical stability. Damping is generally recommended: oscillations in the vertical force would have a direct effect on friction forces in sliding isolators; for isolators with curved surfaces, vertical oscillations can be excited as the isolator slides up and down the curved surface. It may occasionally be necessary to increase DAMP if these oscillations become significant.

This element has no rotational stiffness - a pin joint is assumed. However, if required, moments can be generated according to the vertical load times the lateral displacement of the isolator. The moment about the local X-axis (i.e. the moment that is dependent on lateral displacement in the local Y-direction) is reacted on nodes 1 and 2 of the element in the proportions MX1 and MX2 respectively. Similarly, moments about the local Y-axis are reacted in the proportions MY1, MY2. These inputs effectively determine the location of the pin joint: for example, a pin at the base of the column could be modelled by setting MX1=MY1=1.0, MX2=MY2=0.0 and ensuring that node 1 is on the foundation, node 2 at the base of the column - then all the moment is transferred to the foundation. For the same model, MX1=MY1=0.0, MX2=MY2=1.0 would imply a pin at the top of the foundation - all the moment is transferred to the column. Some isolator designs have the pin at the bottom for moments about one horizontal axis, and at the top for the other axis - these can be modelled by setting MX1=MY2=1.0, MX2=MY1=0.0. It is expected that all MX1,2, etc lie between 0 and 1, and that MX1+MX2=1.0 (or both can be zero) - e.g. MX1=MX2=0.5 is permitted - but no error checks are performed to ensure this; similarly for MY1+MY2.

Density should be set to a reasonable value, say 2000 to 8000 kg//m³. The element mass will be calculated as density x volume (volume is entered on *SECTION_BEAM).

Note on values for *SECTION_BEAM:

- Set ELFORM to 6 (discrete beam)
- VOL (the element volume) might typically be set to 0.1m³
- INER needs to be non-zero (say 1.0) but the value has no effect on the solution since the element has no rotational stiffness.
- CID can be left blank if the isolator is aligned in the global coordinate system, otherwise a coordinate system should be referenced.
- By default, the isolator will be assumed to rotate with the average rotation of its two nodes. If the base of the column rotates slightly the isolator will no longer be perfectly horizontal: this can cause unexpected vertical displacements coupled with the horizontal motion. To avoid this, rotation of the local axes of the isolator can be eliminated by setting RRCON, SRCON and TRCON to 1.0. This does not introduce any rotational restraint to the model, it only prevents the orientation of the isolator from changing as the model deforms.
- All other parameters on *SECTION_BEAM can be left blank.

Post-processing note: as with other discrete beam material models, the force described in post-processors as “Axial” is really the force in the local X-direction; “Y-Shear” is really the force in the local Y-direction; and “Z-Shear” is really the force in the local Z-direction.

*MAT_JOINTED_ROCK

This is Material Type 198. Joints (planes of weakness) are assumed to exist throughout the material at a spacing small enough to be considered ubiquitous. The planes are assumed to lie at constant orientations defined on this material card. Up to three planes can be defined for each material. The matrix behaviour is modified Drucker Prager, as per material type 193.

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	MID	RO	GMOD	RNU	RKF	PHI	CVAL	PSI
Type	I	F	F	F	F	F	F	F
Default					1.0			0.0

Card 2 1 2 3 4 5 6 7 8

Variable	STR_LIM	NPLANES	ELASTIC	LCCPDR	LCCPT	LCCJDR	LCCJT	LCSFAC
Type	F	I	I	I	I	I	I	I
Default	0.005	0	0	0	0	0	0	0

Card 3 1 2 3 4 5 6 7 8

Variable	GMODDP	PHIDP	CVALDP	PSIDP	GMODGR	PHIGR	CVALGR	PSIGR
Type	F	F	F	F	F	F	F	F
Default	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0

Repeat Card 4 for each plane (maximum 3 planes):

Card 3 1 2 3 4 5 6 7 8

Variable	DIP	STRIKE	CPLANE	FRPLANE	TPLANE	SHRMAX	LOCAL	
Type	F	F	F	F	F	F	F	
Default	0.0	0.0	0.0	0.0	0.0	1.e20	0.0	

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MID	Material identification number, must be unique.
RO	Mass density
GMOD	Elastic shear modulus
RNU	Poisson's ratio
RKF	Failure surface shape parameter
PHI	Angle of friction (radians)
CVAL	Cohesion value
PSI	Dilation angle (radians)
STR_LIM	Minimum shear strength of material is given by STR_LIM*CVAL
NPLANES	Number of joint planes (maximum 3)
ELASTIC	Flag = 1 for elastic behaviour only
LCCPDR	Loadcurve for extra cohesion for parent material (dynamic relaxation)
LCCPT	Loadcurve for extra cohesion for parent material (transient)
LCCJDR	Loadcurve for extra cohesion for joints (dynamic relaxation)
LCCJT	Loadcurve for extra cohesion for joints (transient)
LCSFAC	Loadcurve giving factor on strength vs time
GMODDP	Depth at which shear modulus (GMOD) is correct
PHIDP	Depth at which angle of friction (PHI) is correct

<u>VARIABLE</u>	<u>DESCRIPTION</u>
CVALDP	Depth at which cohesion value (CVAL) is correct
PSIDP	Depth at which dilation angle (PSI) is correct
GMODGR	Gradient at which shear modulus (GMOD) increases with depth
PHIGR	Gradient at which friction angle (PHI) increases with depth
CVALGR	Gradient at which cohesion value (CVAL) increases with depth
PSIGR	Gradient at which dilation angle (PSI) increases with depth
DIP	Angle of the plane in degrees below the horizontal
DIPANG	Plan view angle (degrees) of downhill vector drawn on the plane
CPLANE	Cohesion for shear behaviour on plane
PHPLANE	Friction angle for shear behaviour on plane (degrees)
TPLANE	Tensile strength across plane (generally zero or very small)
SHRMAX	Max shear stress on plane (upper limit, independent of compression)
LOCAL	=0: DIP and DIPANG are with respect to the global axes =1: DIP and DIPANG are with respect to the local element axes

Remarks:

1. The joint plane orientations are defined by the angle of a “downhill vector” drawn on the plane, i.e. the vector is oriented within the plane to obtain the maximum possible downhill angle. DIP is the angle of this line below the horizontal. DIPANG is the plan-view angle of the line (pointing down hill) measured clockwise from the global Y-axis about the global Z-axis.
2. The joint planes rotate with the rigid body motion of the elements, irrespective of whether their initial definitions are in the global or local axis system.
3. The full facilities of the modified Drucker Prager model for the matrix material can be used – see description of Material type 193. Alternatively, to speed up the calculation, the ELASTIC flag can be set to 1, in which case the yield surface will not be considered and only RO, GMOD, RNU, GMODDP, GMODGR and the joint planes will be used.
4. This material type requires that the model is oriented such that the z-axis is defined in the upward direction. The key parameters are defined such that may vary with depth (i.e. the z-axis)
5. The shape factor for a typical soil would be 0.8, but should not be pushed further than 0.75.
6. If STR_LIM is set to less than 0.005, the value is reset to 0.005.

7. A correction has been introduced into the Drucker Prager model, such that the yield surface never infringes the Mohr-Coulomb criterion. This means that the model does not give the same results as a “pure” Drucker Prager model.
8. The loadcurves LCCPDR, LCCPT, LCCJDR, LCCJT allow additional cohesion to be specified as a function of time. The cohesion is additional to that specified in the material parameters. This is intended for use during the initial stages of an analysis to allow application of gravity or other loads without cracking or yielding, and for the cracking or yielding then to be introduced in a controlled manner. This is done by specifying extra cohesion that exceeds the expected stresses initially, then declining to zero. If no curves are specified, no extra cohesion is applied.
9. The loadcurve for factor on strength applies simultaneously to the cohesion and $\tan(\text{friction angle})$ of parent material and all joints. This feature is intended for reducing the strength of the material gradually, to explore factors of safety. If no curve is present, a constant factor of 1 is assumed. Values much greater than 1.0 may cause problems with stability.
10. Extra variables for plotting. By setting NEIPH on *DATABASE_EXTENT_BINARY to 15, the following variables can be plotted in D3PLOT and T/HIS:
 - Extra Variable 1: Mobilised strength fraction for base material
 - Extra Variable 2: rk0 for base material
 - Extra Variable 3: rlamda for base material
 - Extra Variable 4: crack opening strain for plane 1
 - Extra Variable 5: crack opening strain for plane 2
 - Extra Variable 6: crack opening strain for plane 3
 - Extra Variable 7: crack accumulated shear strain for plane 1
 - Extra Variable 8: crack accumulated shear strain for plane 2
 - Extra Variable 9: crack accumulated shear strain for plane 3
 - Extra Variable 10: current shear utilisation for plane 1
 - Extra Variable 11: current shear utilisation for plane 2
 - Extra Variable 12: current shear utilisation for plane 3
 - Extra Variable 13: maximum shear utilisation to date for plane 1
 - Extra Variable 14: maximum shear utilisation to date for plane 2
 - Extra Variable 15: maximum shear utilisation to date for plane 3
14. Joint planes would generally be defined in the global axis system if they are taken from survey data. However, the material model can also be used to represent masonry, in which case the weak planes represent the cement and lie parallel to the local element axes.

***MAT_SPRING_ELASTIC**

This is Material Type 1 for discrete springs and dampers. This provides a translational or rotational elastic spring located between two nodes. Only one degree of freedom is connected.

Card 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 is Material Type 2 for discrete springs and dampers. This material provides a linear translational or rotational damper located between two nodes. Only one degree of freedom is then connected.

Card 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 is Material Type 3 for discrete springs and dampers. This material provides an elastoplastic translational or rotational spring with isotropic hardening located between two nodes. Only one degree of freedom is connected.

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	MID	K	KT	FY				
Type	I	F	F	F				

<u>VARIABLE</u>	<u>DESCRIPTION</u>
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 is Material Type 4 for discrete springs and dampers. This material provides a nonlinear elastic translational and rotational spring with arbitrary force versus displacement and moment versus rotation, respectively. Optionally, strain rate effects can be considered through a velocity dependent scale factor. With the spring located between two nodes, only one degree of freedom is connected.

Card 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 is Material Type 5 for discrete springs and dampers. This material provides a viscous translational damper with an arbitrary force versus velocity dependency, or a rotational damper with an arbitrary moment versus rotational velocity dependency. With the damper located between two nodes, only one degree of freedom is connected.

Card 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 is Material Type 6 for discrete springs and dampers. This material provides a general nonlinear translational or rotational spring with arbitrary loading and unloading definitions. Optionally, hardening or softening can be defined. With the spring located between two nodes, only one degree of freedom is connected.

Card 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 20.35.
LCDU	Load curve identification describing force versus displacement resp. moment versus rotation relationship for unloading, see Figure 20.34.
BETA	Hardening parameter, β : 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 without strain softening , EQ.1.0: isotropic hardening without strain softening .
TYI	Initial yield force in tension (> 0)
CYI	Initial yield force in compression (< 0)

Remarks:

Load curve points are in the format (displacement, force or rotation, moment). The points must be in order starting with the most negative (compressive) displacement 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.

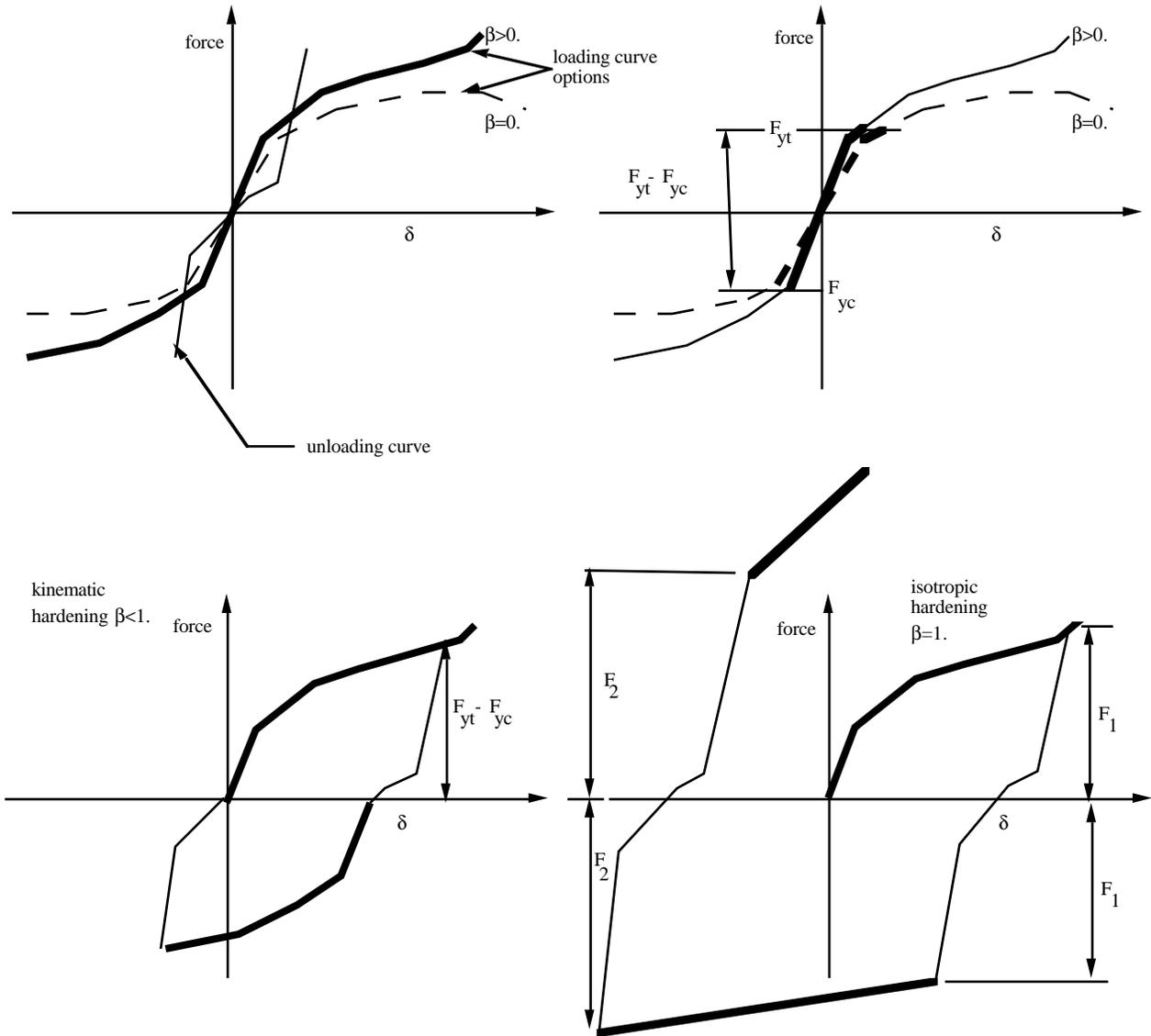


Figure 20.35. General nonlinear material for discrete elements.

***MAT_SPRING_MAXWELL**

This is Material Type 7 for discrete springs and dampers. This material provides a three Parameter Maxwell Viscoelastic translational or rotational spring. Optionally, a cutoff time with a remaining constant force/moment can be defined.

Card 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 ²⁰	0	0	

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MID	Material identification. A unique number has to be chosen.
K0	K_0 , short time stiffness
KI	K_∞ , long time stiffness
BETA	Decay parameter.
TC	Cut off time. After this time a constant force/moment is transmitted.
FC	Force/moment after cutoff time
COPT	Time implementation option: EQ.0: incremental time change, NE.0: continuous time change.

Remarks:

The time varying stiffness $K(t)$ may be described in terms of the input parameters as

$$K(t) = K_\infty + (K_0 - K_\infty)e^{-\beta t} .$$

This equation was implemented by Schwer [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 is Material Type 8 for discrete springs and dampers. This material provides an inelastic tension or compression only, translational or rotational spring. Optionally, a user-specified unloading stiffness can be taken instead of the maximum loading stiffness.

Card 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_TRILINEAR_DEGRADING**

This is Material Type 13 for discrete springs and dampers. This material allows concrete shearwalls to be modelled as discrete elements under applied seismic loading. It represents cracking of the concrete, yield of the reinforcement and overall failure. Under cyclic loading, the stiffness of the spring degrades but the strength does not.

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	MID	DEFL1	F1	DEFL2	F2	DEFL3	F3	FFLAG
Type	I	F	F	F	F	F	F	F

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MID	Material identification. A unique number has to be chosen.
DEFL1	Deflection at point where concrete cracking occurs.
F1	Force corresponding to DEFL1
DEFL2	Deflection at point where reinforcement yields
F2	Force corresponding to DEFL2
DEFL3	Deflection at complete failure
F3	Force corresponding to DEFL3
FFLAG	Failure flag.

*MAT_SPRING_SQUAT_SHEARWALL

This is Material Type 14 for discrete springs and dampers. This material allows squat shearwalls to be modelled using discrete elements. The behaviour modelled captures concrete cracking, reinforcement yield, ultimate strength followed by degradation of strength finally leading to collapse.

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	MID	A14	B14	C14	D14	E14	LCID	FSD
Type	I	F	F	F	F	F	I	F

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MID	Material identification. A unique number has to be chosen.
A14	Material coefficient A
B14	Material coefficient B
C14	Material coefficient C
D14	Material coefficient D
E14	Material coefficient E
LCID	Loadcurve id referencing the maximum strength envelope curve
FSD	Sustained strength reduction factor

Material coefficients A, B, C and D are empirically defined constants used to define the shape of the polynomial curves which govern the cyclic behaviour of the discrete element. A different polynomial relationship is used to define the loading and unloading paths allowing energy absorption through hysteresis. Coefficient E is used in the definition of the path used to ‘jump’ from the loading path to the unloading path (or vice versa) where a full hysteresis loop is not completed. The loadcurve referenced is used to define the force displacement characteristics of the shear wall under monotonic loading. This curve is the basis to which the polynomials defining the cyclic behaviour refer to. Finally, on the second and subsequent loading / unloading cycles the shear wall will have reduced strength. The variable FSD is the sustained strength reduction factor.

***MAT_SPRING_MUSCLE**

This is Material Type 15 for discrete springs and dampers. This material is a Hill-type muscle model with activation. It is for use with discrete elements. The LS-DYNA implementation is due to Dr. J.A. Weiss.

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	MID	L0	VMAX	SV	A	FMAX	TL	TV
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	FPE	LMAX	KSH					
Type	F	F	F					
Default	0.0							

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MID	Material identification. A unique number has to be chosen.
L0	Initial muscle length, <i>L₀</i> .
VMAX	Maximum CE shortening velocity, <i>V_{max}</i> .
SV	Scale factor, <i>S_v</i> , for <i>V_{max}</i> 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 <i>A</i> is used
FMAX	Peak isometric force, <i>F_{max}</i> .

<u>VARIABLE</u>	<u>DESCRIPTION</u>
TL	Active tension vs. length function. LT.0: absolute value gives load curve ID GE.0: constant value of 1.0 is used
TV	Active tension vs. velocity function. LT.0: absolute value gives load curve ID GE.0: constant value of 1.0 is used
FPE	Force vs. length function, F_{pe} , for parallel elastic element. LT.0: absolute value gives load curve ID EQ.0: exponential function is used (see below) GT.0: constant value of 0.0 is used
LMAX	Relative length when F_{pe} reaches F_{max} . Required if $F_{pe}=0$ above.
KSH	Constant, K_{sh} , governing the exponential rise of F_{pe} . Required if $F_{pe}=0$ 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 20.36). 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.

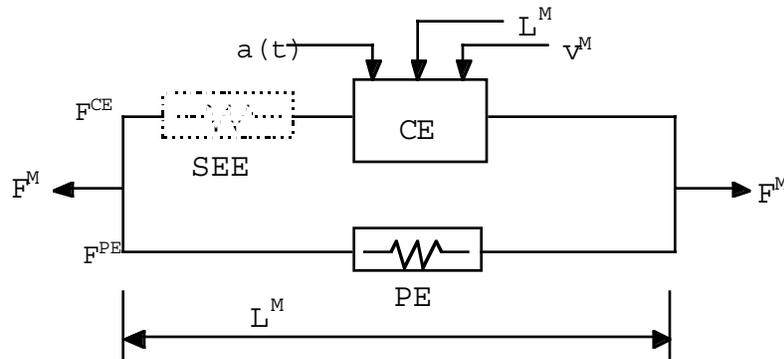


Figure 20.36 Discrete model for muscle contraction dynamics, based on a Hill-type representation. The total force is the sum of passive force F^{PE} and active force F^{CE} . The passive element (PE) represents energy storage from muscle elasticity, while the contractile element (CE) represents force 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:

$$F^{\text{CE}} = a(t) * F_{\max} * f_{\text{TL}}(L) * f_{\text{TV}}(V)$$

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 20.36).

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):

$$f_{\text{PE}} = \frac{F^{\text{PE}}}{F_{\text{MAX}}} = 0, \quad L \leq 1$$

$$f_{\text{PE}} = \frac{F^{\text{PE}}}{F_{\text{MAX}}} = \frac{1}{\exp(K_{\text{sh}}) - 1} \left[\exp\left(\frac{K_{\text{sh}}}{L_{\max}}(L - 1)\right) - 1 \right], \quad L > 1$$

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 20.37).

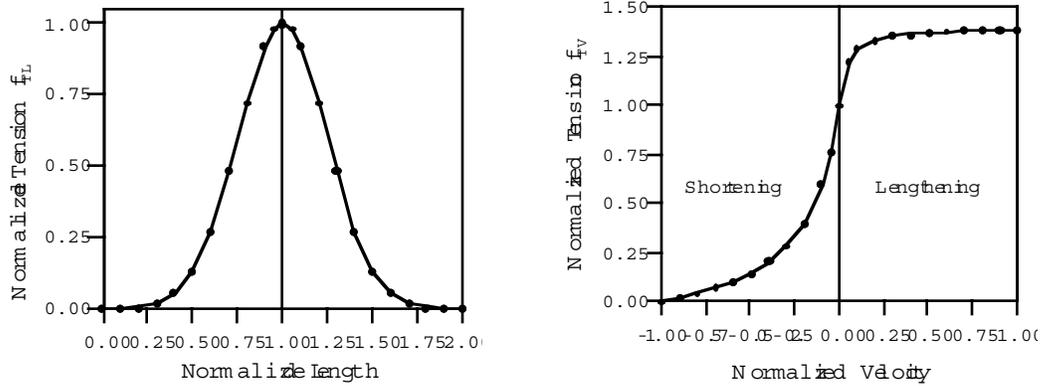


Figure 20.37 Typical normalized tension-length (TL) and tension-velocity (TV) curves for skeletal muscle.

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 20.37). 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			

VARIABLE

DESCRIPTION

MID	Belt material number. A unique number has to be chosen.
MPUL	Mass per unit length
LLCID	Load curve identification for loading (force vs. engineering strain).
ULCID	Load curve identification for unloading (force vs. engineering strain).
LMIN	Minimum length (for elements connected to slip rings and retractors), see notes below.

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 is absorbed into the retractors. One-tenth of a typical initial element length is usually a good choice.

**MAT_THERMAL_OPTION*

Options include:

ISOTROPIC

ORTHOTROPIC

ISOTROPIC_TD

ORTHOTROPIC_TD

ISOTROPIC_PHASE_CHANGE

ISOTROPIC_TD_LC

The **MAT_THERMAL_* cards allow thermal properties to be defined in coupled structural/thermal and thermal only analyses, see **CONTROL_SOLUTION*. Thermal properties must be defined for all solid and shell elements in such analyses. Thermal properties need not be defined for beam or discrete elements as these elements are not accounted for in the thermal phase of the calculation. However dummy thermal properties will be echoed for these elements in the D3HSP file.

Thermal material properties are specified by a thermal material ID number (TMID), this number is independent of the material ID number (MID) defined on all other **MAT_..* property cards. In the same analysis identical TMID and MID numbers may exist. The TMID and MID numbers are related through the **PART* card.

***MAT_THERMAL_ISOTROPIC**

This is thermal material property type 1. It allows isotropic thermal properties to be defined.

Card 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	TMID	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					

<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.
HC	Heat capacity
K1	Thermal conductivity K_1 in local x-direction
K2	Thermal conductivity K_2 in local y-direction
K3	Thermal conductivity K_3 in local z-direction
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_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	TMID	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	TMID	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) ₁ ... (K1) ₈	Thermal conductivity K_1 in local x-direction at T1 ... T8
(K2) ₁ ... (K2) ₈	Thermal conductivity K_2 in local y-direction at T1 ... T8
(K3) ₁ ... (K3) ₈	Thermal conductivity K_3 in local z-direction at T1 ... T8
XP, YP, ZP	Define coordinate of point p for AOPT = 1
A1, A2, A3	Define components of vector a for AOPT = 2
D1, D2, D3	Define components of vector v for AOPT = 2

***MAT_THERMAL_ISOTROPIC_PHASE_CHANGE**

This is thermal material property type 5. It allows temperature dependent isotropic properties with phase change to be defined. The latent heat of the material is defined together with the 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	TMD	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					

<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
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 $\lambda = \int_{T_s}^{T_L} C(T) dT$

λ = latent heat

c = heat capacity

*MAT_THERMAL_ISOTROPIC_TD_LC

This is thermal material property type 6. It allows isotropic thermal properties that are temperature dependent specified by load curves to be defined. The properties must be defined for the temperature range that the material will see in the analysis.

Card 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**

Three keywords are defined in this section.

***NODE**

***NODE_RIGID_SURFACE**

***NODE_SCALAR**

*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. The nodal point ID must be unique relative to other nodes defined in the *NODE section.

Card Format (I8,3E16.0,2F8.0)

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

Variable	NID	X	Y	Z	TC	RC				
Type	I	F	F	F	F	F				
Default	none	0.	0.	0.	0.	0.				
Remarks					1	1				

<u>VARIABLE</u>	<u>DESCRIPTION</u>
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.

<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**.

*NODE

*NODE_RIGID_SURFACE

Purpose: Define a rigid node and its coordinates in the global coordinate system. These nodes are used to define rigid road surfaces and they have no degrees of freedom. The nodal points are used in the definition of the segments that define the rigid surface. See *CONTACT_RIGID_SURFACE. The nodal point ID must be unique relative to other nodes defined in the *NODE section.

Card Format (I8,3E16.0)

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

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

<u>VARIABLE</u>	<u>DESCRIPTION</u>
NID	Node number
X	x coordinate
Y	y coordinate
Z	z coordinate

***NODE_SCALAR**

Purpose: Define a scalar nodal point which has one degree-of-freedom. The scalar point ID must be unique relative to other nodes defined in the *NODE section.

Card Format (2I8)

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

Variable	NID	NDOF								
Type	I	I								
Default	none	0								
Remarks										

VARIABLE

DESCRIPTION

NID

Scalar node ID.

NDOF

Number of degrees-of-freedom
EQ.0: fully constrained
EQ.1: one degree-of-freedom

***NODE**

***PARAMETER**

Purpose: Define the numerical values of parameter names referenced throughout the input file. The parameter definitions, if used, should be placed at the beginning of the input file following *KEYWORD.

Define as many card as necessary.

Card 1 1 2 3 4 5 6 7 8

Variable	PRMR1	VAL1	PRMR2	VAL2	PRMR3	VAL3	PRMR4	VAL4
Type	A	I or F						
Default	none	none	none	none	none	none	none	none

Card 2,3,...

Variable	PRMRn	VALn	PRMRn+1	VALn+1		
Type	A	I or F	A	I or F				
Default	none	none	none	none				

VARIABLE

DESCRIPTION

PRMRn

Define the nth parameter in a field of 10. Within this field the first character must be either an "R" for a real number or an "I" for an integer. Lower or upper case for "I" or "R" is okay. Following the type designation, define the name of the parameter using up to, but not exceeding seven characters. For example, when defining a shell thickness named, "SHLTHK", both inputs "RSHLTHK" or "R SHLTHK" can be used and placed anywhere in the field of 10. When referencing SHLTHK in the the input file see Remark 1 below.

VALn

Define the numerical value of the n parameter as either a real or integer number consistent with preceding definition for PRMRn.

Remarks:

- Parameters can be referenced anywhere in the input by placing an "&" at the first column of its field followed by the name of the parameter without blanks..

***PARAMETER**

***PART**

Three keywords are used in this section.

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

***PART_ADAPTIVE_FAILURE**

***PART_MODES**

***PART_SENSOR**

***PART_MOVE**

*PART

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

For *OPTION1* the available choices are

<BLANK>

INERTIA

REPOSITION

For *OPTION2* the available choices are

<BLANK>

CONTACT

For *OPTION3* the available choices are

<BLANK>

PRINT

For *OPTION4* the available choices are

<BLANK>

ATTACHMENT_NODES

Options 1, 2, 3, and 4 may be specified in any order on the *PART card.

Purpose: Define parts, i.e., combine material information, section properties, hourglass type, thermal properties, and a flag for part adaptivity.

The **INERTIA** option allows the inertial properties and initial conditions to be defined rather than calculated from the finite element mesh. This applies to rigid bodies, see *MAT_RIGID, only. The **REPOSITION** option applies to deformable materials and is used to reposition deformable materials attached to rigid dummy components whose motion is controlled by either CAL3D or MADYMO. At the beginning of the calculation each component controlled by CAL3D/MADYMO is automatically repositioned to be consistent with the CAL3D/MADYMO input. However, deformable materials attached to these component will not be repositioned unless this option is used.

The **CONTACT** option allows part based contact parameters to be used with the automatic contact types a3, 4, a5, a10, 13, a13, 15 and 26, that is

***CONTACT_AUTOMATIC_SURFACE_TO_SURFACE**

***CONTACT_SINGLE_SURFACE,**

***CONTACT_AUTOMATIC_NODES_TO_SURFACE,**

***CONTACT_AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE,**

*CONTACT_AUTOMATIC_SINGLE_SURFACE,

*CONTACT_AIRBAG_SINGLE_SURFACE,

*CONTACT_ERODING_SINGLE_SURFACE,

*CONTACT_AUTOMATIC_GENERAL.

The default values to use for these contact parameters can be specified on the *CONTACT input 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	NODEID		
Type	F	F	F	F	I	I		

Card 4 1 2 3 4 5 6 7 8

Variable	IXX	IXY	IXZ	IYY	IYZ	IZZ		
Type	F	F	F	F	F	F		

Card 5 1 2 3 4 5 6 7 8

Variable	VTX	VTY	VTZ	VRX	VRY	VRZ		
Type	F	F	F	F	F	F		

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

Card 6 1 2 3 4 5 6 7 8

Variable	XL	YL	ZL	XLIP	YLIP	ZLIP	CID	
Type	F	F	F	F	F	F	I	
Remark	2	2	2	2	2	2	none	

An additional Card is required for the REPOSITION option.

Optional 1 2 3 4 5 6 7 8

Variable	CMSN	MDEP	MOVOPT					
Type	I	I	I					

Additional Card is required for the CONTACT option.

WARNING: If FS, FD, DC, and VC are specified they will not be used unless FS is set to a negative value (-1.0) in the *CONTACT section. These frictional coefficients apply only to contact types: SINGLE_SURFACE, AUTOMATIC_GENERAL, AUTOMATIC_SINGLE_SURFACE, AUTOMATIC_NODES_TO_..., AUTOMATIC_SURFACE_..., AUTOMATIC_ONE_WAY_..., and ERODING_SINGLE_SURFACE. Default values are input via *CONTROL_CONTACT input.

Optional 1 2 3 4 5 6 7 8

Variable	FS	FD	DC	VC	OPTT	SFT	SSF	
Type	F	F	F	F	F	F	F	

An additional Card is required for the PRINT option. This option applies to rigid bodies and provides a way to turn off ASCII output in files RBDOUT and MATSUM.

Optional 1 2 3 4 5 6 7 8

Variable	PRBF							
Type	I							

*PART

An additional Card is required for the ATTACHMENT_NODES option. All nodes are treated as attachment nodes if this option is not used. Attachment nodes apply to rigid bodies only. The motion of these nodes, which must belong to the rigid body, are updated each cycle. Other nodes in the rigid body are updated only for output purposes. Include all nodes in the attachment node set which interact with the structure through joints, contact, merged nodes, applied nodal point loads, and applied pressure. Include all nodes in the attachment node set if their displacements, accelerations, and velocities are to be written into an ASCII output file. Body force loads are applied to the c.g. of the rigid body.

Optional 1 2 3 4 5 6 7 8

Variable	ANSID								
Type	I								

<u>VARIABLE</u>	<u>DESCRIPTION</u>
HEADING	Heading for the part
PID	Part identification
SECID	Section identification defined in the *SECTION section
MID	Material identification defined in the *MAT section
EOSID	Equation of state identification defined in the *EOS section. Nonzero only for solid elements using a an equation of state to compute pressure.
HGID	Hourglass/bulk viscosity identification defined in the *HOURLASS 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 and 3-D tetrahedrons.

<u>VARIABLE</u>	<u>DESCRIPTION</u>
TMID	Thermal material property identification defined in the *MAT_THERMAL Section. Thermal properties must be specified for all solid, shell, and thick shell parts if a thermal or coupled thermal structural/analysis is being performed. Beams and discrete elements are not considered in thermal analyses. EQ.0: defaults to MID
XC	x-coordinate of center of mass. If nodal point, NODEID, is defined XC, YC, and ZC are ignored and the coordinates of the nodal point, NODEID, are taken as the center of mass.
YC	y-coordinate of center of mass
ZC	z-coordinate of center of mass
TM	Translational mass
IRCS	Flag for inertia tensor reference coordinate system: EQ.0: global inertia tensor, EQ.1: local inertia tensor is given in a system defined by the orientation vectors.
NODEID	Nodal point defining the CG of the rigid body. This node should be included as an extra node for the rigid body; however, this is not a requirement. If this node is free, its motion will not be updated to correspond with the rigid body after the calculation begins.
IXX	I_{xx} , xx component of inertia tensor
IXY	I_{xy} , xy component of inertia tensor (see Remark 4)
IXZ	I_{xz} , xz component of inertia tensor (see Remark 4)
IYY	I_{yy} , yy component of inertia tensor
IYZ	I_{yz} , yz component of inertia tensor (see Remark 4)
IZZ	I_{zz} , zz component of inertia tensor
VTX	initial translational velocity of rigid body in x direction
VTY	initial translational velocity of rigid body in y direction
VTZ	initial translational velocity of rigid body in z direction
VRX	initial rotational velocity of rigid body about x axis
VRX	initial rotational velocity of rigid body about y axis
VRZ	initial rotational velocity of rigid body about z axis

*PART

<u>VARIABLE</u>	<u>DESCRIPTION</u>
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 $\mu_c = FD + (FS - FD)e^{-DC v_{rel} }$
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 $\mu_c = FD + (FS - FD)e^{-DC v_{rel} }$
VC	Coefficient for viscous friction. This is necessary to limit the friction force to a maximum. A limiting force is computed $F_{lim} = VC \cdot A_{cont}$. A_{cont} being the area of the segment contacted by the node in contact. The suggested value for VC is to use the yield stress in shear $VC = \frac{\sigma_o}{\sqrt{3}}$ where σ_o is the yield stress of the contacted material.

<u>VARIABLE</u>	<u>DESCRIPTION</u>
OPTT	Optional contact thickness. This applies to shells only.
SFT	Optional thickness scale factor for PART ID in automatic contact (scales true thickness). This option applies only to contact with shell elements. True thickness is the element thickness of the shell elements.
SSF	Scale factor on default slave penalty stiffness for this PART ID whenever it appears in the contact definition. If zero, SSF is taken as unity.
PRBF	Print flag for RBDOUT and MATSUM files. EQ.0: default is taken from the keyword *CONTROL_OUTPUT, EQ.1: write data into RBDOUT file only EQ.2: write data into MATSUM file only EQ.3: do not write data into RBDOUT and MATSUM
ANSID	Attachment node set ID. This option should be used very cautiously and applies only to rigid bodies. The attachment point nodes are updated each cycle whereas other nodes in the rigid body are updated only in the output databases. All loads seen by the rigid body must be applied through this nodal subset or directly to the center of gravity of the rigid body. If the rigid body is in contact this set must include all interacting nodes. EQ.0: All nodal updates are skipped for this rigid body. The null option can be used if the rigid body is fixed in space or if the rigid body does not interact with other parts, e.g., the rigid body is only used for some visual purpose.

Remarks:

1. HEADING default is standard material description, e.g. Material Type 1. In case of SMUG post processing place PSHELL (or PBAR, or PSOLID) in columns 1-8 and Property name in columns 34-41.
2. The local cartesian coordinate system is defined as described in *DEFINE_COORDINATE_VECTOR. The local z-axis vector is the vector cross product of the x axis and the in plane vector. The local y-axis vector is finally computed as the vector cross product of the z-axis vector and the x-axis vector. The local coordinate system defined by CID has the advantage that the local system can be defined by nodes in the rigid body which makes repositioning of the rigid body in a preprocessor much easier since the local system moves with the nodal points.
3. When 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. Note that the off-diagonal terms of the inertia tensor are opposite in sign from the products of inertia.
5. The initial velocity of the rigid body may be overwritten by the *INITIAL_VELOCITY card. See parameter IRIGID on this card.

*PART

*PART_ADAPTIVE_FAILURE

Purpose: This is an option for two-dimensional adaptivity to allow a part that is singly connected to split into two parts. This option is under development and will be generalized in the future to allow the splitting of parts that are multiply connected.

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	PID	T						
Type	I	F						

<u>VARIABLE</u>	<u>DESCRIPTION</u>
PID	Part ID
T	Thickness. When the thickness of the part reaches this minimum value the part is split into two parts. <i>The value for T should be on the order of the element thickness of a typical element.</i>

***PART_MODES**

Purpose: Define mode shapes for a flexible rigid body. Currently, flexible bodies cannot be merged into other flexible bodies or rigid bodies; however, interconnections to other rigid/flexible bodies can use the penalty joint option. The flexible rigid bodies are not implemented with the Lagrange multiplier joint option. The deformations are modeled using the modes shapes obtained experimentally or in a finite element analysis, e.g., NASTRAN.pch file or an LSTC eigout file. These modes should include both constraint and attachment modes. For stress recovery in flexible rigid bodies, use of linear element formulations is recommended. A lump mass matrix is assumed in the implementation. Also see the keyword control card: *CONTROL_RIGID.

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	PID	NMFB	FORM	ANSID	FORMAT	KMFLAG	NUPDF	SIGREC
Type	I	I	I	I	I	I	I	

Card 2

Variable	FILENAME
Type	C
Default	none

Define the following cards if and only if KMFLAG=1. Use as many cards as necessary to identify the NMFB kept modes. After NMFB modes are defined no further input is expected.

Cards 3, ... 1 2 3 4 5 6 7 8

Variable	MODE1	MODE2	MODE3	MODE4	MODE5	MODE6	MODE7	MODE8
Type	I	I	I	I	I	I	I	I
Default	none	nont	none	nont	none	nont	none	nont

*PART

Read optional modal damping cards here. A keyword card (with a "*" in column 1) terminates this input.

Card 1 2 3 4 5 6 7 8

Variable	MSTART	MSTOP	DAMPF					
Type	I	I	F					
Default	none	nont	none					

VARIABLE

DESCRIPTION

PID	Part identification. This part must be a rigid body.
NMFB	Number of kept modes in flexible body. The number of modes in the file, FILENAME, must equal or exceed NMFB. If KMFLAG=0 the first NMFB modes in the file are used.
FORM	Flexible body formulation. See remark 5 below. EQ.0: exact EQ.1: fast
ANSID	Attachment node set ID (optional).
FORMAT	Input format of modal information: EQ.0: NASTRAN.pch file. EQ.1: (not supported) EQ.2: NASTRAN.pch file (LS-DYNA binary version). The binary version of this file is automatically created if a NASTRAN.pch file is read. The name of the binary file is the name of the NASTRAN.pch file but with ".bin" appended. The binary file is smaller and can be read much faster. EQ.3: LS-DYNA d3eigv binary eigenvalue database (see *CONTROL_IMPLICIT_EIGENVALUE). EQ.4: LS-DYNA d3mode binary constraint/attachment mode database (see *CONTROL_IMPLICIT_MODE). EQ.5: Both d3eigv and d3mode databases are input. Database names must be "d3eigv" and "d3mode", and FILENAME below is ignored. NMFB above gives the total number of modes in both databases.
KMFLAG	Kept mode flag. Selects method for identifying modes to keep. EQ.0: the first NMFB modes in the file, FILENAME, are used. EQ.1: define NMFB kept modes with additional input.
NUPDF	Nodal update flag. If active, an attachment node set, ANSID, must be defined. EQ.0: all nodes of of the rigid part are updated each cycle.

<u>VARIABLE</u>	<u>DESCRIPTION</u>
	EQ.1: only attachment nodes are fully updated. All nodes in the body are output based on the rigid body motion without the addition of the modal displacements. For maximum benefit an attachment node set can also be defined with the PART_ATTACHMENT_NODES option. The same attachment node set ID should be used here.
SIGREC	Stress recovery flag. If active, attachment nodes should not be used. EQ.0: no stress recovery EQ.1: recover stresses.
FILENAME	The path and name of a file which contains the modes for this rigid body.
MODEn	Keep normal mode, MODEn.
MSTART	First mode for damping, $(1 \leq MSTART \leq NMFB)$.
MSTOP	Last mode for damping, MSTOP, $(1 \leq MSTOP \leq NMFB)$. All modes between MSTART and MSTOP inclusive are subject to the same modal damping coefficient, DAMPF.
DAMPF	Modal damping coefficient, ζ .

Remarks:

1. The format of the file which contains the normal modes follows the file formats of NASTRAN output for modal information.
2. The mode set typically combines both normal modes and attachment modes. The eigenvalues for the attachment modes are computed from the stiffness and mass matrices.
3. The part ID specified must be either a single rigid body or a master rigid body (see *CONSTRAINED_RIGID_BODIES) which can be made up of many rigid parts.
4. The modal damping is defined by the modal damping coefficient ζ ., where a value of 1.0 equals critical damping. For a one degree of freedom model system, the relationship between the damping and the damping coefficient is $c = 2\zeta\omega_n m$, where c is the damping, m is the mass, and ω_n is the natural frequency, $\sqrt{k/m}$.
5. There are two formulation options. The first is a formulation that contains all the terms of the flexible body equations, and its cost grows approximately as the square of the number of modes. The second formulation ignores most of the second order terms appearing in the exact equations and its cost grows linearly with the number of modes. Users are responsible for determining which formulation is appropriate for their problems. In general, if the angular velocities are small and if the deflections are small with respect to the geometry of the system it is safe to use the second (faster) formulation.

*PART

*PART_SENSOR

Purpose: Activate and deactivate parts, based on sensor defined in ELEMENT_SEATBELT_SENSOR. This option applies to discrete beam element only.

Define one card. Card Format (3I10)

Card 1 2 3 4 5 6 7 8

Variable	PID	SIDA	ACTIVE					
Type	I	I	I					
Default	0	0	0					

<u>VARIABLE</u>	<u>DESCRIPTION</u>
PID	Part ID, which is controlled by sensor
SIDA	Sensor ID to activate or deactivate part.
ACTIVE	Flag. If zero, the part is active from time zero until a signal is received by the part to deactivate.. If one, the part is inactive from time zero and becomes active when a signal is received by the part to activate. The history variables for inactive parts are initialized at time zero.

***PART_MOVE**

Purpose: Translate 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.

***RAIL**

Two keywords are defined in this section.

***RAIL_TRACK**

***RAIL_TRAIN**

*RAIL

*RAIL_TRACK

Purpose: Wheel-rail contact algorithm intended for railway applications but can also be used for other purposes. The wheel nodes (defined on *RAIL_TRAIN) represent the contact patch between wheel and rail. A penalty method is used to constrain the wheel nodes to slide along the track. A track consists of two rails, each of which is defined by a set of beam elements.

Card Format – Card 1 of 2

Card 1 1 2 3 4 5 6 7 8

Variable	ID	BSETID1	NORGN1	LCUR1	OSET1	SF1	GA1	
Type	I	I	I	I	F	F	F	
Default	none	None	None	None	0.0	1.0	0.0	

Card 2

Variable	BLANK	BSETID2	NORGN2	LCUR2	OSET2	SF2	GA2	
Type	-	I	I	I	F	F	F	
Default	-	None	None	None	0.0	1.0	0.0	

VARIABLE

DESCRIPTION

ID	Track ID
BSETID1,2	Beam set ID for rails 1 and 2 containing all beam elements that make up the rail, see *SET_BEAM.
NORGN1,2	Reference node at one end of each rail, used as the origin for the roughness curve. The train will move in a direction away from this node.
LCUR1,2	Loadcurve ID (see *DEFINE_CURVE) defining track roughness (vertical displacement from line of beam elements) of the rail as a function of distance from the reference node NORIGIN. Distance from reference node on x-axis of curve, roughness on y-axis. Default: no roughness.

<u>VARIABLE</u>	<u>DESCRIPTION</u>
OSET1,2	Origin of curve LCUR is shifted by distance OSET from the reference node.
SF1,2	Roughness values are scaled by SF. Default: 1.0.
GA1,2	Shear stiffness of rail per unit length (used to calculate local rail shear deformation within each beam element). GA = shear modulus x cross-sectional area. Default: local shear deformation is ignored.

Remarks:

*RAIL_TRACK and *RAIL_TRAIN were written by Arup to represent wheel-rail contact. They have been used to generate loading on models of bridges for vibration predictions, stress calculations and for estimating accelerations experienced by passengers. Other non-railway uses are possible: the algorithm causes the “train” nodes to follow the line defined by the “rail” beam elements and transfers forces between them. In some cases (especially vibration modelling), double precision versions of LS-DYNA may give superior results because of the small relative deflections between wheel and rail.

Track modelling

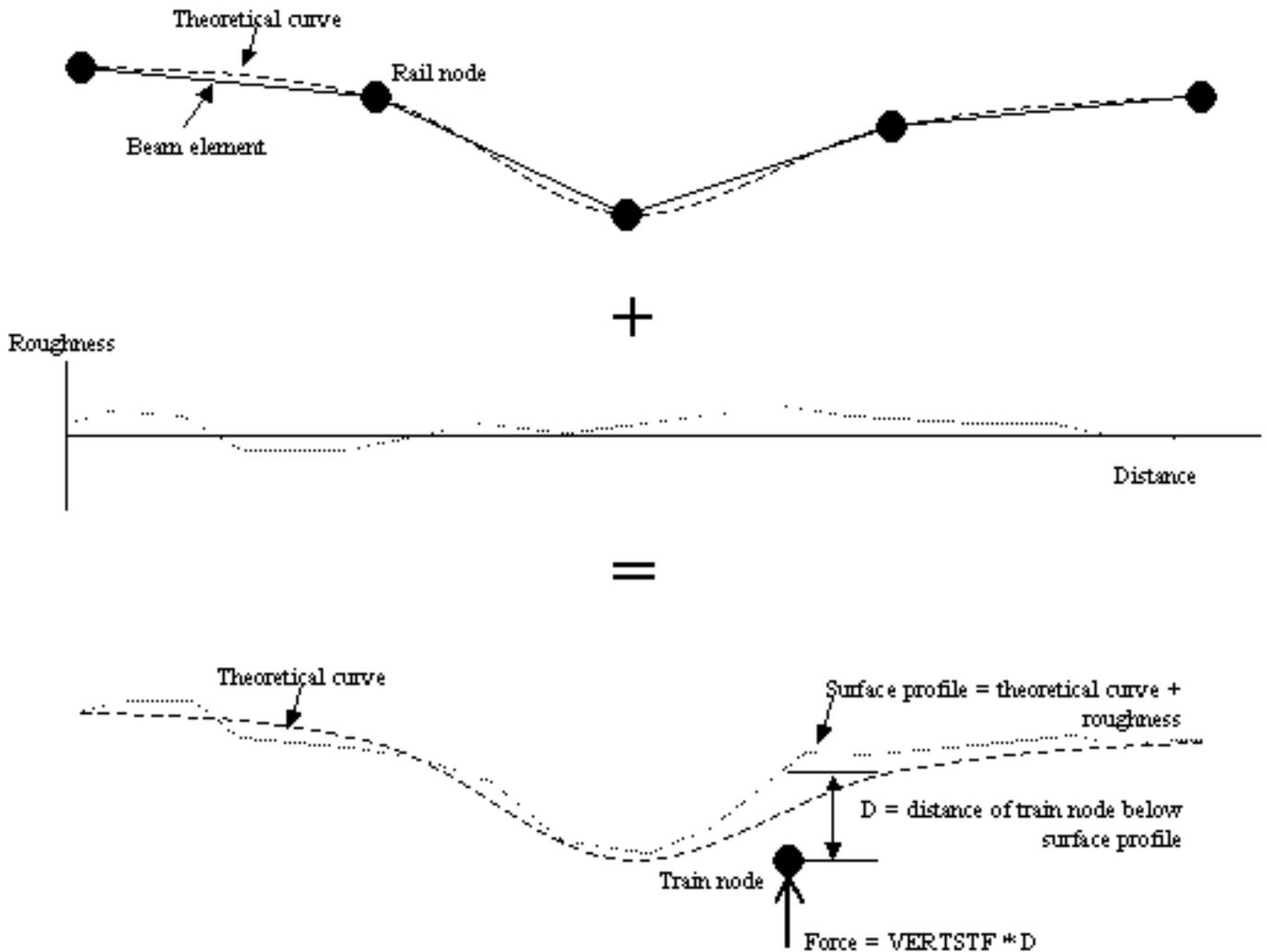
The rails of the track should be modelled by two parallel lines of beam elements. The track can be curved or straight and the rails can be modelled as deformable or rigid. If required, rail pads, sleepers and ballast may also be modelled – typically with spring, damper and beam elements. It is also possible to use this algorithm to control the motion of simple road vehicle models: beam element “rails” made of null material can be embedded in the road surface. It is recommended that the mesh size of the two rails should be similar: LS-DYNA calculates a local coordinate system for each train node based on the alignment of the currently contacted beam element and the nearest node on the other rail.

Because wheel-rail contact stiffness is generally very high, and wheel masses are large, small deviations from a straight line or smooth curve can lead to large transient forces. It is recommended that great care be taken in generating and checking the geometry for the track, especially where the track is curved. Some pre-processors write the coordinates with insufficient precision to the LS-DYNA input file, and this can cause unintended roughness in the geometry. For the same reason, if the line of the track were taken as straight between nodes, spurious forces would be generated when the wheel passes from one rail element to the next. This is avoided because the *RAIL algorithm calculates a theoretical curved centreline for the rail element to achieve continuity of slope from one element to the next. Where the length of the rail elements is similar to or shorter than the maximum section dimension, shear deformation may be significant and it is possible to include this in the theoretical centreline calculation to further reduce spurious forces at the element boundaries (inputs GA1, GA2).

Roughness (small deviations in the vertical profile from a perfect straight line) does exist in real life and is a principal source of vibration. *RAIL allows the roughness to be modelled by a loadcurve giving the vertical deviation (in length units) of the rail surface from the theoretical centreline of the beam elements as a function of distance along the track from the origin node of the rail. The roughness curve is optional. Ideally, roughness profiles measured from both rails of the same piece of track should be used so that the relationship between bump and roll modes is correctly captured.

Whether roughness is included or not, it is important to select as the origin nodes (NORIGIN1 and NORIGIN2) the nodes at the end of the rails away from which the train will be travelling. The train can start at any point along the rails but must travel away from the origin nodes.

*RAIL



Train modelling

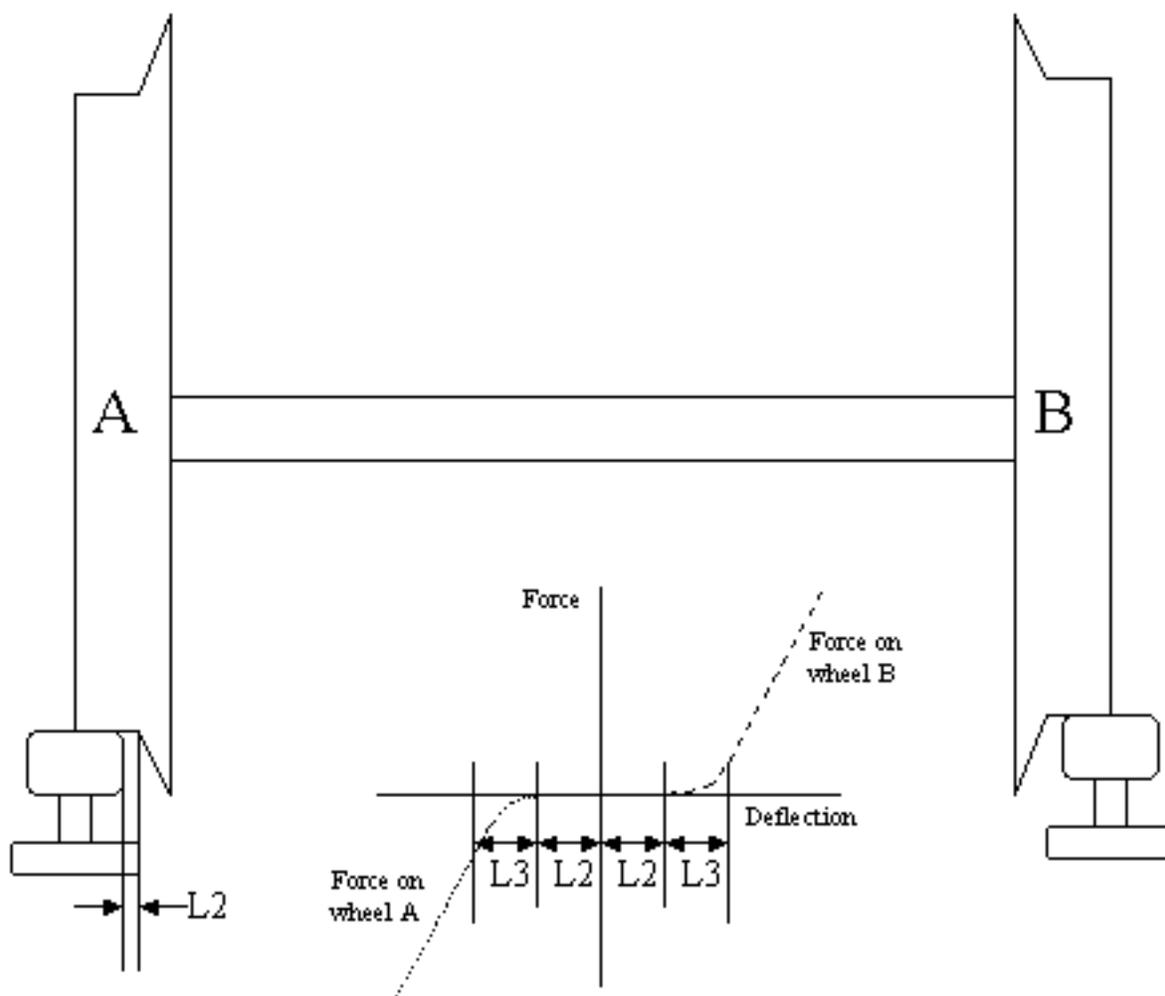
The vehicle models are typically modelled using spring, damper and rigid elements, or simply a point mass at each wheel position. Each node in the set referred to on *RAIL_TRAIN represents the contact patch of one wheel (note: not the centre of the wheel). These nodes should be initially on or near the line defined by either of the two rails. LS-DYNA will move the train nodes initially onto the rails to achieve the correct initial wheel-rail forces. If the results are viewed with magnified displacements, the initial movements can appear surprising.

Wheel roughness input is available. This will be applied in addition to track roughness. The input curve must continue for the total rolled distance – it is not assumed to repeat with each wheel rotation. This is to avoid problems associated with ensuring continuity between the start and end of the profile

around the wheel circumference, especially since the profiles might be generated from roughness spectra rather than taken directly from measured data.

Wheel-rail interface

The wheel-rail interface model is a simple penalty function designed to ensure that the train nodes follow the line of the track. It does not attempt to account for the shape of the rail profile. Vertical and lateral loads are treated independently. For this reason, the algorithm is not suitable for rail vehicle dynamics calculations.



Wheel-rail contact stiffness is input on *RAIL_TRAIN. For vertical loads, a linear force-deflection relationship is assumed in compression; no tensile force is generated (this corresponds to the train losing contact with the rail). Typical contact stiffness is 2MN/mm. Lateral deflections away from the theoretical centreline of the rail beams is also penalised by a linear force-deflection relationship. The lateral force is applied only to wheels on the side towards which the train has displaced (corresponding to wheel flanges that run inside the rails). Optionally, a “gap” can be defined (input parameter L2) such that the wheelset can drift laterally by L2 length units before any lateral force is generated. A further option is to allow smooth transition between “gap” and “contact” by means of a

*RAIL

transition distance (input parameter L3). Generally, with straight tracks a simple linear stiffness is sufficient. With curved tracks, a reasonable gap and transition distance should be defined to avoid unrealistic forces being generated in response to small inaccuracies in the distance between the rails. Gravity loading is expected, in order to maintain contact between rail and wheel. This is normally applied by an initial phase of dynamic relaxation. To help achieve convergence quickly, or in some cases avoid the need for dynamic relaxation altogether, the initial force expected on each train node can be input (parameter FINIT on *RAIL_TRAIN). LS-DYNA positions the nodes initially such that the vertical contact force will be FINIT at each node. If the suspension of the rail vehicles is modelled, it is recommended that the input includes carefully calculated precompression of the spring elements; if this is not done, achieving initial equilibrium under gravity loading can be very time consuming.

The *RAIL algorithm ensures that the train follows the rails, but does not provide forward motion. This is generally applied using *INITIAL_VELOCITY, or for straight tracks, *BOUNDARY_PRESCRIBED_MOTION.

Output

LS-DYNA generates an additional ASCII output file **train_force_n**, where *n* is an integer updated to avoid overwriting any existing files. The file contains the forces on each train node, output at the same time intervals as the binary time history file (DT on *DATABASE_BINARY_D3THDT).

Checking

It is recommended that track and train models be tested separately before adding the *RAIL cards. Check that the models respond stably to impulse forces and that they achieve equilibrium under gravity loading. The majority of problems we have encountered have been due to unstable behaviour of train or track. Often, these are first detected by the *RAIL algorithm and an error message will result.

***RAIL_TRAIN**

Purpose: Define train properties. A train is defined by a set of node in contact with a rail defined by *RAIL_TRACK.

Card Format – Card 1 of 2

Card 1 1 2 3 4 5 6 7 8

Variable	ID	NSETID	(omit)	FINIT	(omit)	TRID	LCUR	OFFS
Type	I	I	F	F	F	I	I	F
Default	none	None	0.0	0.0	0.0	0	None	0.0

Card 2

Variable	VERTSTF	LATSTF	V2	V3	L2	L3		
Type	F	F	F	F	F	F		
Default	0.0	0.0	0.0	0.0	0.0	0.0		

VARIABLE**DESCRIPTION**

ID	Train ID
NSETID	Node set ID containing all nodes that are in contact with rails.
(omit)	Unused variable – leave blank.
FINIT	Estimate of initial vertical force on each wheel (optional) – speeds up the process of initial settling down under gravity loading.
(omit)	Unused variable – leave blank.
TRID	ID of track for this train, see *RAIL_TRACK.
LCUR	Load curve ID (see *DEFINE_CURVE) containing wheel roughness (distance of wheel surface away from perfect circle) vs. distance travelled. The curve does not repeat with each rotation of the wheel – the last point should be at a greater distance than the train is expected to travel. Default: no wheel roughness.

*RAIL

<u>VARIABLE</u>	<u>DESCRIPTION</u>
OFFS	Offset distance used to generate different roughness curves for each wheel from the roughness curve LCUR. The curve is offset on the x axis by a different whole number multiple of OFFS for each wheel.
VERTSTF	Vertical stiffness of rail contact.
LATSTF	Lateral stiffness of rail contact.
V2,V3	Unused variables – leave blank.
L2	Lateral clearance from rail to wheel rim. Lateral force is applied to a wheel only when it has moved more than L2 away from the other rail, i.e. the wheel rims are assumed to be near the inner face of the rail.
L3	Further lateral distance before full lateral stiffness applies (force-deflection curve follows a parabola up to this point).

***RIGIDWALL**

Two keywords are used in this section to define rigid surfaces:

***RIGIDWALL_GEOMETRIC_OPTION_{OPTION}_{OPTION}**

***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.

Card Format for GEOMETRIC options:

- Cards 1 and 2 are required for all geometric shapes.
- Card 3 is required, but is dependent upon which shape is specified.
- Optional Card A is required if MOTION is specified.

Card 1 - Required for all shape types

Card 1 1 2 3 4 5 6 7 8

Variable	NSID	NSIDEX	BOXID					
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, n , originating on wall (tail) and terminating in space (head), see Figure 22.1.
YT	y-coordinate of tail of normal vector n
ZT	z-coordinate of tail of normal vector n
XH	x-coordinate of head of normal vector n
YH	y-coordinate of head of normal vector n
ZH	z-coordinate of head of normal vector n
FRIC	Interface friction: EQ.0.0: frictionless sliding after contact, EQ.1.0: stick condition after contact, 0.<FRIC<1.: Coulomb friction coefficient.

Card 3 - Required if FLAT is specified after the keyword.

A plane with a finite size or with an infinite size can be defined, see Figure 22.1. The vector **m** is computed as the vector cross product $\mathbf{n} \times \mathbf{l}$. The origin, which is the tail of the normal vector, is the corner point of the finite size plane.

Card 3 1 2 3 4 5 6 7 8

Variable	XHEV	YHEV	ZHEV	LENL	LENM			
Type	F	F	F	F	F			
Default	0.	0.	0.	infinity	infinity			

VARIABLE**DESCRIPTION**

XHEV	x-coordinate of head of edge vector l , see Figure 22.1.
YHEV	y-coordinate of head of edge vector l
ZHEV	z-coordinate of head of edge vector l
LENL	Length of l edge. A zero value defines an infinite size plane.
LENM	Length of m edge. A zero value defines an infinite size plane.

*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 22.1.

Card 3 1 2 3 4 5 6 7 8

Variable	XHEV	YHEV	ZHEV	LENL	LENM	LENP		
Type	F	F	F	F	F	F		
Default	none	0.	0.	infinity	infinity	infinity		

VARIABLE

DESCRIPTION

XHEV	x-coordinate of head of edge vector l , see Figure 22.1.
YHEV	y-coordinate of head of edge vector l
ZHEV	z-coordinate of head of edge vector l
LENL	Length of l edge. A zero value defines an infinite size plane.
LENM	Length of m edge. A zero value defines an infinite size plane.
LENP	Length of prism in the direction negative to n , see Figure 22.1.

Card 3 - Required if CYLINDER is specified after the keyword.

The tail of **n** specifies the top plane of the cylinder. The length is defined in the direction negative to **n**, see Figure 22.1.

Card 3 1 2 3 4 5 6 7 8

Variable	RADCYL	LENCYL						
Type	F	F						
Default	none	infinity						

VARIABLE

DESCRIPTION

RADCYL

Radius of cylinder

LENCYL

Length of cylinder, see Figure 22.1. Only if a value 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 22.1.

Card 3 1 2 3 4 5 6 7 8

Variable	RADSPH							
Type	F							
Default	0.							

VARIABLE

DESCRIPTION

RADSPH

Radius of sphere

Optional Card A - Required if MOTION is specified after the keyword.

Optional Card A	1	2	3	4	5	6	7	8
Variable	LCID	OPT	VX	VY	VZ			
Type	I	I	F	F	F			
Default	none	none	none	none	none			

VARIABLE

DESCRIPTION

LCID	Stonewall motion curve number, see *DEFINE_CURVE.
OPT	Type of motion: EQ.0: velocity specified, EQ.1: displacement specified.
VX	x-direction cosine of velocity/displacement vector
VY	y-direction cosine of velocity/displacement vector
VZ	z-direction cosine of velocity/displacement vector

*RIGIDWALL

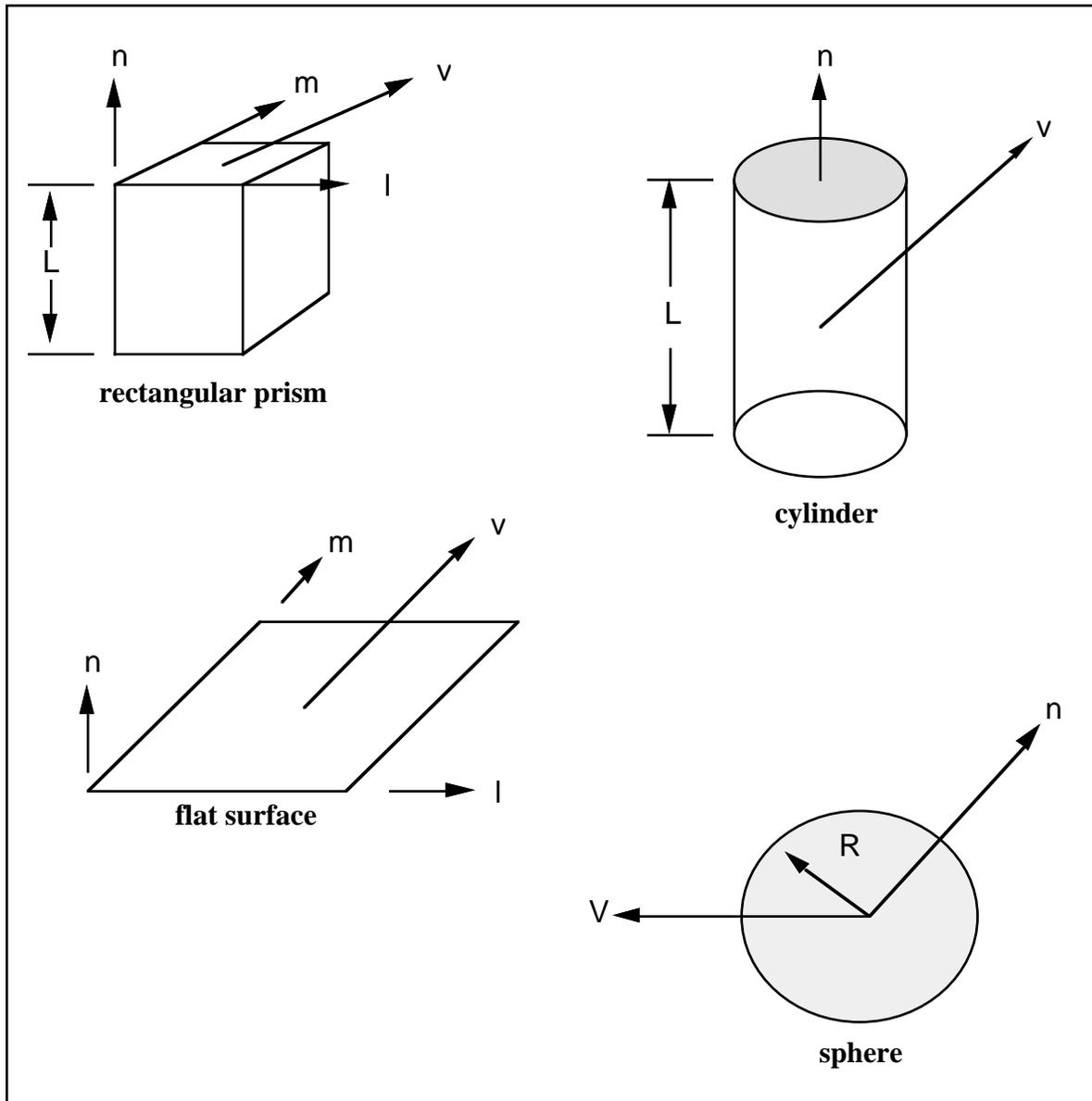


Figure 22.1. Vector n determines the orientation of the generalized stonewalls. For the prescribed motion options the wall can be moved in the direction V as shown.

*RIGIDWALL

*RIGIDWALL_PLANAR_{OPTION}_{OPTION}_{OPTION}

Available options include:

<BLANK>

ORTHO

FINITE

MOVING

FORCES

The ordering of the options in the input below must be observed but the ordering of the options on the command line is unimportant, i.e.; the **ORTHO** card is first, the **FINITE** definition card below must precede the **MOVING** definition card, and the **FORCES** definition card should be last. The **ORTHO** option does not apply if the **MOVING** option is used. If an ID number is specified the additional option is available:

ID

If active, the ID card is the first card following the keyword.

Purpose: Define planar rigid walls with either finite or infinite size (**FINITE**). Orthotropic friction can be defined (**ORTHO**). Also, the plane can possess a mass and an initial velocity (**MOVING**); otherwise, the wall is assumed to be stationary. The **FORCES** option allows the specification of segments on the rigid walls on which the contact forces are computed. In order to achieve a more physical reaction related to the force versus time curve, the SOFT value on the **FORCES** card can be specified.

ID Card Define if and only if ID option is active.

Card 1 1 2 3 4 5 6 7 8

Variable	RWID								
Type	I								
Default	none								

VARIABLE

DESCRIPTION

RWID

Rigid wall ID. Up to 8 characters can be used.

Card Format:

- Cards 1 and 2 are required.
- Optional Cards A and B are required if ORTHO is specified.
- Optional Card C is required if FINITE is specified.
- Optional Card D is required if MOVING is specified.
- Optional Card E is required if FORCES is specified.

Card 1 - Required.

Card 1 1 2 3 4 5 6 7 8

Variable	NSID	NSIDEX	BOXID	OFFSET				
Type	I	I	I	F				
Default	none	0	0	0.				

VARIABLE

DESCRIPTION

NSID	Nodal set ID containing slave nodes, see *SET_NODE_OPTION: EQ.0: all nodes are slave to rigid wall.
NSIDEX	Nodal set ID containing nodes that exempted as slave nodes, see *SET_NODE_OPTION.
BOXID	All nodes in box are included as slave nodes to rigid wall, see *DEFINE_BOX. If options NSID or NSIDEX are active then only the subset of nodes activated by these options are checked to see if they are within the box.
OFFSET	All nodes within a normal offset distance, OFFSET, to the rigid wall are included as slave nodes for the rigid wall. If options NSID, NSIDEX, or BOXID are active then only the subset of nodes activated by these options are checked to see if they are within the offset distance. This option applies to the PLANAR wall only.

*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, n , originating on wall (tail) and terminating in space (head), see Figure 22.3.
YT	y-coordinate of tail of normal vector n
ZT	z-coordinate of tail of normal vector n
XH	x-coordinate of head of normal vector n
YH	y-coordinate of head of normal vector n
ZH	z-coordinate of head of normal vector n
FRIC	Interface friction: EQ.0.0: frictionless sliding after contact, EQ.1.0: no sliding after contact, 0.<FRIC<1.: Coulomb friction coefficient. EQ.2.0: node is welded after contact with frictionless sliding. Welding occurs if and only if the normal value of the impact velocity exceeds the critical value specified by WVEL. EQ.3.0: node is welded after contact with no sliding. Welding occurs if and only if the normal value of the impact velocity exceeds the critical value specified by WVEL.
WVEL	Critical normal velocity at which nodes weld to wall (FRIC = 2 or 3).

Optional Cards A and B - Required if ORTHO is specified after the keyword.

See Figure 23.2 for the definition of orthotropic friction.

Optional Card A	1	2	3	4	5	6	7	8
Variable	SFRICA	SFRICB	DFRICA	DFRICB	DECAYA	DECAYB		
Type	F	F	F	F	F	F		
Default	0.	0.	0	0	0.	0.		

Optional Card B	1	2	3	4	5	6	7	8
Variable	NODE1	NODE2	D1	D2	D3			
Type	I	I	F	F	F			
Default	0.	0.	0	0	0.			

VARIABLE**DESCRIPTION**

SFRICA	Static friction coefficient in local a-direction, μ_{sa} , see Figure 22.2
SFRICB	Static friction coefficient in local b-direction, μ_{sb}
DFRICA	Dynamic friction coefficient in local a-direction, μ_{ka}
DFRICB	Dynamic friction coefficient in local b-direction, μ_{kb}
DECAYA	Decay constant in local a-direction, d_{va}
DECAYB	Decay constant in local b-direction, d_{vb}
NODE1	Node 1, alternative to definition with vector d below, see Figure 22.2. With the node definition the direction changes if the nodal pair rotates.
NODE2	Node 2

*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

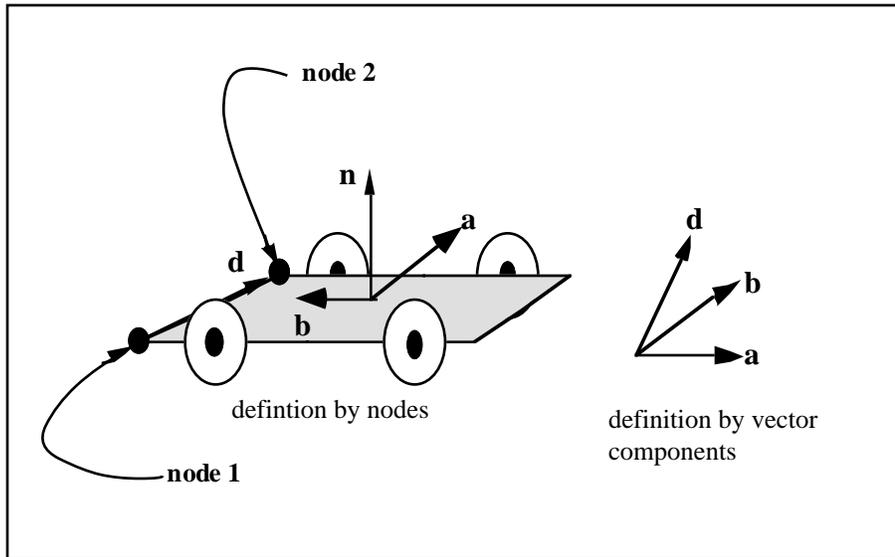


Figure 22.2. Definition of orthotropic friction vectors. The two methods of defining the vector, \mathbf{d} are shown. If vector \mathbf{d} is defined by nodes 1 and 2, the local coordinate system may rotate with the body which contains the nodes; otherwise, \mathbf{d} is fixed in space, thus on the 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 l , see Figure 22.3.
YHEV	y-coordinate of head of edge vector l
ZHEV	z-coordinate of head of edge vector l
LENL	Length of l edge
LENM	Length of m edge

*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.						

<u>VARIABLE</u>	<u>DESCRIPTION</u>
MASS	Total mass of stonewall
V0	Initial velocity of stonewall in direction of defining vector, n

Optional Card E - Required if FORCES is specified after the keyword.

This option allows the force distribution to be monitored on the plane. Also four points can be defined for visualization of the rigid wall. A shell or membrane element must be defined with these four points as the connectivity for viewing in LS-PREPOST.

Optional Card E	1	2	3	4	5	6	7	8
Variable	SOFT	SSID	N ₁	N ₂	N ₃	N ₄		
Type	I	I	I	I	I	I		
Default	0	0	0	0	0	0		
Remarks		1	2					

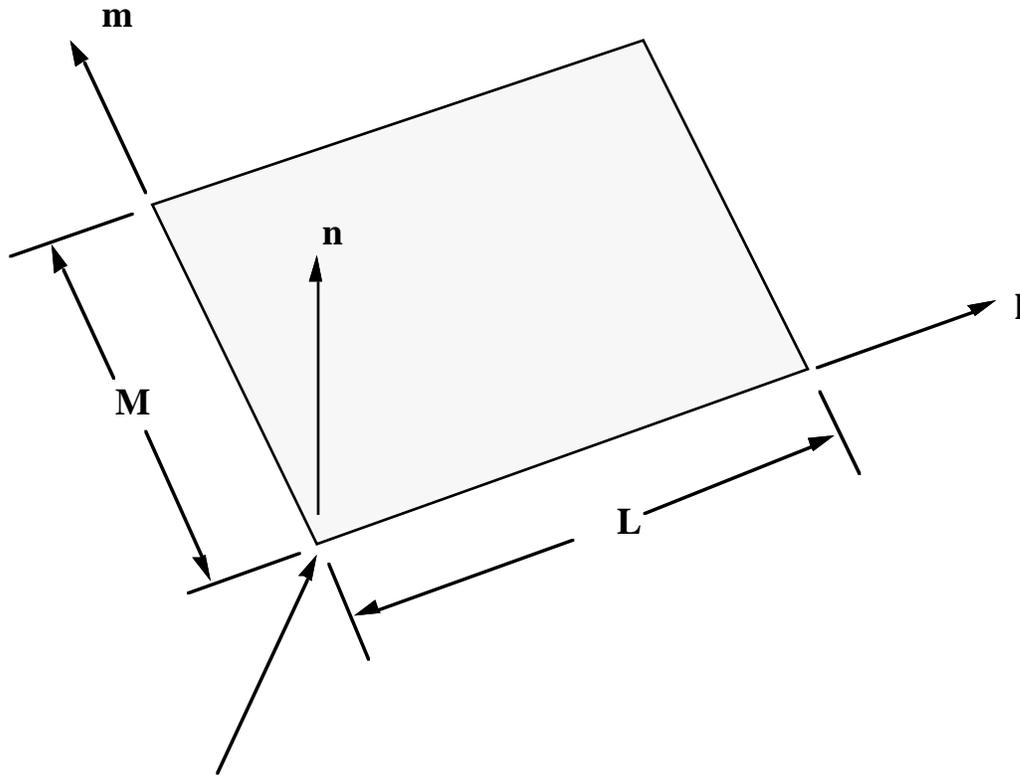
VARIABLE**DESCRIPTION**

SOFT	Number of cycles to zero relative velocity to reduce force spike
SSID	Segment set identification number for defining areas for force output, see *SET_SEGMENT and remark 1 below.
N1-N4	Optional node for visualization

Remarks:

- 1 The segment set defines areas for computing resultant forces. These segments translate with the moving stonewall and allow the forced distribution to be determined. The resultant forces are written in file "RWFORC."
2. These four nodes are for visualizing the movement of the wall, i.e., they move with the wall. To view the wall in LS-PREPOST it is necessary to define a single shell element with these four nodes as its connectivity. The single element must be deformable (non rigid) or else the segment will be treated as a rigid body and the nodes will have their motion modified independently of the stonewall.

*RIGIDWALL



Tail of normal vector is the origin and corner point if extent of stonewall is finite.

Figure 22.3. 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 L (LENL) and M (LENM). A zero value for either of these lengths indicates that the stonewall is infinite in that direction.

*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_POINT_SOURCE**

***SECTION_POINT_SOURCE_MIXTURE**

***SECTION_SEATBELT**

***SECTION_SHELL_{*OPTION*}**

***SECTION_SOLID_{*OPTION*}**

***SECTION_SPH**

***SECTION_TSHELL**

The location and order of these cards in the input file are arbitrary.

An additional option **_TITLE** may be appended to all the ***SECTION** keywords. If this option is used then an addition line is read for each section in 80a format which can be used to describe the section. At present LS-DYNA does not make use of the title. Inclusion of titles gives greater clarity to input decks.

*SECTION

*SECTION_BEAM

Purpose: Define cross sectional properties for beam, truss, discrete beam, and cable elements.

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	SECID	ELFORM	SHRF	QR/IRID	CST	SCoor	NSM	
Type	I	I	F	F	F	F	F	
Default	none	1	1.0	2.0	0.0	0.0	0.0	

Define the appropriate card format depending on the value of ELFORM (1-9) above.

Card 2

Integrated beam type 1	TS1	TS2	TT1	TT2	NSLOC	NTLOC		
Resultant 2,3	A	ISS	ITT	IRR	SA			
Integrated beam type 4,5	TS1	TS2	TT1	TT2				
Discrete 6	VOL	INER	CID	CA	OFFSET	RRCON	SRCON	TRCON
Scalar 6	VOL	INER	CID	DOFN1	DOFN2			
2D shells 7,8	TS1	TS2	TT1	TT2				
spotweld 9	TS1	TS2	TT1	TT2	PRINT			
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
SECID	Section ID. SECID is referenced on the *PART card and must be unique.
ELFORM	Element formulation options: EQ.1: Hughes-Liu with cross section integration (default), EQ.2: Belytschko-Schwer resultant beam (resultant), EQ.3: truss (resultant), see remark 2. EQ.4: Belytschko-Schwer full cross-section integration, EQ.5: Belytschko-Schwer tubular beam with cross-section integration, EQ.6: discrete beam/cable, EQ.7: 2D plane strain shell element (xy plane), EQ.8: 2D axisymmetric volume weighted shell element (xy plane), EQ.9: spotweld beam, see *MAT_SPOTWELD. 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 (circular only), EQ.2.0: arbitrary (user defined integration rule).
SCoor	Location of triad for tracking the rotation of the discrete beam element, see the parameter CID below. The force and moment resultants in the output databases are referenced to this triad. The flags -3.0, -1.0, 0.0, 1.0, and 3.0 are inactive if the option to update the local system is active in the CID definition. EQ.-3.0: beam node 1, the angular velocity of node 1 rotates triad, EQ.-2.0: beam node 1, the angular velocity of node 1 rotates triad but the r-axis is adjusted to lie along the line between the two beam nodal points. This option is not recommended for zero length discrete beams.,

*SECTION

<u>VARIABLE</u>	<u>DESCRIPTION</u>
	EQ.-1.0: beam node 1, the angular velocity of node 1 rotates triad, EQ. 0.0: centered between beam nodes 1 and 2, the average angular velocity of nodes 1 and 2 is used to rotate the triad, EQ.+1.0: beam node 2, the angular velocity of node 2 rotates triad. EQ.+2.0: beam node 2, the angular velocity of node 2 rotates triad. but the r-axis is adjusted to lie along the line between the two beam nodal points. This option is not recommended for zero length discrete beams. EQ.+3.0: beam node 2, the angular velocity of node 2 rotates triad. <i>If the magnitude of SC00R is less than or equal to unity then zero length discrete beams are assumed with infinitesimal separation between the nodes in the deformed state. For large separations or nonzero length beams set SC00R to 2 or 3.</i>
NSM	Nonstructural mass per unit length. This option applies to beam types 1-5 and does not apply to discrete, 2D, and spotweld beams, respectively.
TS1	Beam thickness (CST=0.0, 2.0) or outer diameter (CST = 1.0) in s direction at node n ₁ . Note that the thickness defined on the *ELEMENT_BEAM_THICKNESS card overrides the definition give here.
TS2	Beam thickness (CST=0.0, 2.0) or outer diameter (CST = 1.0) in s direction at node n ₂ .
TT1	Beam thickness (CST=0.0, 2.0) or inner diameter (CST = 1.0) in t direction at node n ₁ .
TT2	Beam thickness (CST=0.0, 2.0) or inner diameter (CST = 1.0) in t direction at node n ₂ .
NSLOC	Location of reference surface normal to s axis for Hughes-Liu beam elements only: EQ.1.0: side at s=1.0, EQ.0.0: center, EQ.-1.0: side at s = -1.0.
NTLOC	Location of reference surface normal to t axis for Hughes-Liu beam elements only: EQ.1.0: side at t =1.0, EQ.0.0: center, EQ.-1.0: side at t = -1.0.
A	Cross-sectional area. The definition on *ELEMENT_BEAM_THICKNESS overrides the value defined here, see Figure 23.1.
ISS	I _{ss} . The definition on *ELEMENT_BEAM_THICKNESS overrides the value defined here, see Figure 23.1.
ITT	I _{tt} . The definition on *ELEMENT_BEAM_THICKNESS overrides the value defined here, see Figure 23.1.

VARIABLE	DESCRIPTION
IRR	I_{rr} (J) polar inertia. The definition on *ELEMENT_BEAM_THICKNESS overrides the value defined here, see Figure 23.1. If IRR is zero, then IRR is reset to the sum of ISS+ITT as an approximation.
SA	Shear area. The definition on *ELEMENT_BEAM_THICKNESS overrides the value defined here, see Figure 23.1.
VOL	Volume of discrete beam. If the mass density of the material model for the discrete beam is set to unity, the magnitude of the lumped mass can be defined here instead. This lumped mass is partitioned to the two nodes of the beam element. The translational time step size for the type 6 beam is dependent on the volume, mass density, and the translational stiffness values, so it is important to define this parameter. Defining the volume is also essential for mass scaling if the type 6 beam controls the time step size.
INER	Mass moment of inertia for the six degree of freedom discrete beam. This lumped inertia is partitioned to the two nodes of the beam element. The rotational time step size for the type 6 beam is dependent on the lumped inertia and the rotational stiffness values, so it is important to define this parameter if the rotational springs are active. Defining the rotational inertia is also essential for mass scaling if the type 6 beam rotational stiffness controls the time step size.
CID	Coordinate system ID for orientation (material types 66-69, 93, 95, 97), see *DEFINE_COORDINATE_option. If CID=0, a default coordinate system is defined in the global system or on the third node of the beam, which is used for orientation. This option is not defined for material types than act between two nodal points, such as cable elements. The coordinate system rotates with the discrete beam, see SCOR above.
CA	Cable area, materials type ID 71, *MAT_CABLE.
OFFSET	Offset for cable. For a definition see materials type ID 71, *MAT_CABLE.
RRCON	r-rotational constraint for local coordinate system EQ.0.0: Coordinate ID rotates about r axis with nodes. EQ.1.0: Rotation is constrained about the r-axis
SRCON	s-rotational constraint for local coordinate system EQ.0.0: Coordinate ID rotates about s axis with nodes. EQ.1.0: Rotation is constrained about the s-axis
TRCON	t-rotational constraint for local coordinate system EQ.0.0: Coordinate ID rotates about t axis with nodes. EQ.1.0: Rotation is constrained about the t-axis
DOFN1	Active degree-of-freedom at node 1, a number between 1 to 6 where 1 in x-translation and 4 is x-rotation.
DOFN2	Active degree-of-freedom at node 2, a number between 1 to 6.

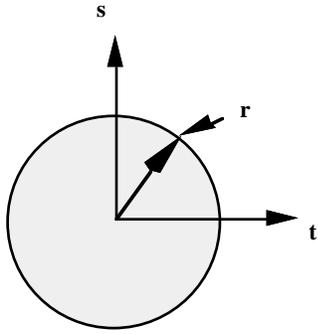
*SECTION

<u>VARIABLE</u>	<u>DESCRIPTION</u>
PRINT	Output spot force resultant from spotwelds. EQ.0.0: Data is output to SWFORC file. EQ.1.0: Output is suppressed.

Remarks:

1. For implicit calculations all of the beam element choices are implemented:
2. For the truss element, define the cross-sectional area, A, only.
3. The local coordinate system rotates as the nodal points that define the beam rotate. In some cases this may lead to unexpected results if the nodes undergo significant rotational motions. In the definition of the local coordinate system using *DEFINE_COORDINATE_NODES, if the option to update the system each cycle is active then this updated system is used. This latter technique seems to be more stable in some applications.

*SECTION

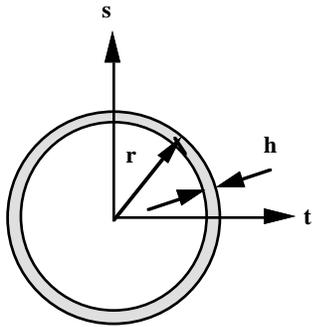


$$I_{tt} = I_{ss} = \frac{\pi r^4}{4}$$

$$J = \frac{\pi r^4}{2}$$

$$f_{tt} = f_{ss} = \frac{10}{9}$$

$$A = \pi r^2$$

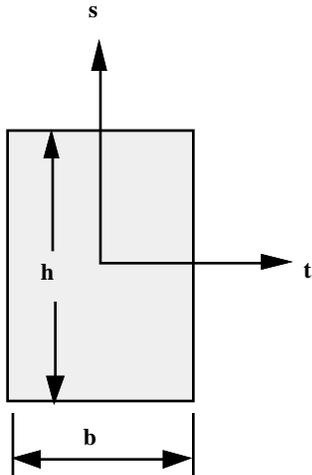


$$I_{tt} = I_{ss} \cong \pi r^3 h$$

$$J \cong 2 \pi r^3 h$$

$$f_{tt} = f_{ss} = 2$$

$$A \cong 2 \pi r h$$



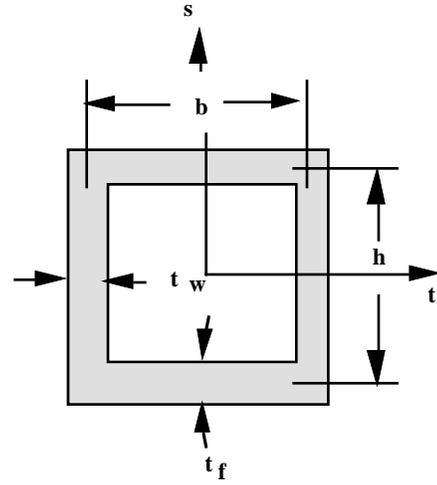
$$I_{tt} = \frac{bh^3}{12}$$

$$I_{ss} = \frac{b^3h}{12}$$

$$J \cong \left[\frac{1}{3} - 2.1 \left(\frac{b}{h} \right) \left(1 - \frac{b^4}{12h^4} \right) \right] hb^3$$

$$f_{tt} = f_{ss} = \frac{6}{5}$$

$$A = bh$$



$$I_{tt} \cong \left(\frac{h^2}{6} \right) (ht_w + 3bt_f)$$

$$I_{ss} \cong \left(\frac{b^2}{6} \right) (bt_f + 3ht_w)$$

$$J \cong \frac{2b^2h^2(t_w t_f)}{(bt_w + ht_f)}$$

$$f_{tt} = \frac{A}{[2(b + t_w)t_f]}$$

$$f_{ss} = \frac{A}{[2(h + t_f)t_w]}$$

$$A = 2(bt_f + ht_w)$$

$$\text{Shear Area} = \frac{A}{f} = \mu A$$

Figure 23.1. Properties of beam cross section for several common cross sections.

***SECTION_DISCRETE**

Purpose: Defined spring and damper elements for translation and rotation. These definitions must correspond with the material type selection for the elements, i.e., *MAT_SPRING_... and *MAT_DAMPER_...

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	SECID	DRO	KD	V0	CL	FD		
Type	I	I	F	F	F	F		

Card 2 1 2 3 4 5 6 7 8

Variable	CDL	TDL						
Type	F	F						

VARIABLE

DESCRIPTION

SECID	Section ID. SECID is referenced on the *PART card and must be unique.
DRO	Displacement/Rotation Option: EQ.0: the material describes a translational spring/damper, EQ.1: the material describes a torsional spring/damper.
KD	Dynamic magnification factor. See remarks 1 and 2 below.
V0	Test velocity
CL	Clearance. See remark 3 below.
FD	Failure deflection (twist for DRO=1). Negative for compression, positive for tension.
CDL	Deflection (twist for DRO=1) limit in compression. See remark 4 below.
TDL	Deflection (twist for DRO=1) limit in tension. See remark 4 below.

*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.

Constrained boundary conditions on the *NODE cards and the BOUNDARY_SPC cards must not be used for nodes of springs with deflection limits.

5. Discrete elements can be included in implicit applications.

*SECTION

*SECTION_POINT_SOURCE

Purpose: This command provides the inlet boundary condition for single gas in flow (inflation potential) via a set of point source(s). It also provides the inflator orifice geometry information. It requires 3 curves defining the inlet condition for the inflator gas coming into the tank or an airbag as input ($\bar{T}_{gas_corrected}(t)$, $v_r(t)$, and $vel(t)$). Please see also the *ALE_TANK_TEST card for additional information.

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	SECID	LCIDT	LCIDVOLR	LCIDVEL	NIDLC001	NIDLC002	NIDLC003	
Type	I	I	I	I	I	I	I	
Default	0	0	0	0	0	0	0	
Variable	NODEID	VECID	ORIFAREA					
Type	I	I	F					
Default	0	0	0.0					

<u>VARIABLE</u>	<u>DESCRIPTION</u>
SECID	Section ID number
LCIDT	Temperature load curve ID
LCIDVOLR	Relative volume load curve ID
LCIDVEL	Inlet flow velocity load curve ID
NIDLC001	The 1 st node ID defining a local coordinate (see remark 2).
NIDLC002	The 2 nd node ID defining a local coordinate (see remark 2).
NIDLC003	The 3 rd node ID defining a local coordinate (see remark 2).
NODEID	The node ID(s) defining the point source(s).
VECID	The vector ID defining the direction of flow at each point source.
ORIFAREA	The orifice area at each point source.

Remarks:

1. In an airbag inflator tank test, the tank pressure data is measured. This pressure is used to derive $\dot{m}(t)$ and the estimated $\bar{T}_{gas}(t)$, usually via a lumped-parameter method, a system of conservation equations and EOS. Subsequently $\dot{m}(t)$ and $\bar{T}_{gas}(t)$ (stagnation temperature) are used as input to obtain $\bar{T}_{gas_corrected}(t)$ (static temperature), $v_r(t)$, and $vel(t)$. These 3 curves are then used to describe inflator gas inlet condition (see *ALE_TANK_TEST for more information).
2. In a car crash model, the inflator housing may get displaced during the impact. The 3 node IDs defines the local reference coordinate system to which the point sources are attached. These 3 reference nodes may be located on a rigid body which can translate and rotate as the inflator moves during the impact. This allows for the point sources to move in time. These reference nodes may be used as the point sources themselves.
3. If the *ALE_TANK_TEST card is present, please see remarks under that card.

Example:

Consider a tank test model consists of the inflator gas (PID 1) and the air inside the tank (PID 2). The 3 load curves define the thermodynamic and kinetic condition of the incoming gas. The nodes define the center of the orifice, and the vector the direction of flow at each orifice.

```
$... | .....1.... | .....2.... | .....3.... | .....4.... | .....5.... | .....6.... | .....7.... | .....8
*PART
inflator gas
$      PID      SECID      MID      EOSID      HGID      GRAV      ADPOPT
TMID
      1          1          1          0          0          0          0
0
*SECTION_POINT_SOURCE
$      SECID      LCIDT      LCIDVOLR      LCIDVEL      NIDLCOOR1      NIDLCOOR2      NIDLCOOR3
      1          3          4          5          0          0          0
$      NODEID      VECTID      AREA
      24485          3      15.066
      ...
      24557          3      15.066
*PART
air inside the tank
$      PID      SECID      MID      EOSID      HGID      GRAV      ADPOPT
TMID
      2          2          2          0          0          0          0
0 *SECTION_SOLID
$      SECID      ELFORM      AET
      2          11          0
*ALE_MULTI-MATERIAL_GROUP
$      SID      SIDTYPE
      1          1
      2          1
$... | .....1.... | .....2.... | .....3.... | .....4.... | .....5.... | .....6.... | .....7.... | .....8
```

*SECTION

*SECTION_POINT_SOURCE_MIXTURE

Purpose: This command provides the inlet boundary condition for multiple-gas mixture in-flow via a set of point source(s). It also provides the inflator orifice geometry information. This must be used in combination with the *MAT_GAS_MIXTURE and/or *INITIAL_GAS_MIXTURE card. This card is designed so that control volume inflator tank test data ($\dot{m}(t)$ and $\bar{T}_{gas}(t)$) may be used as input directly. (This option will be available in the near future, summer 2003).

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	SECID	LCIDT	Not Used	LCIDVEL	NIDLC001	NIDLC002	NIDLC003	
Type	I	I	I	I	I	I	I	
Default	0	0	0	0	0	0	0	
Variable	LCMDOT1	LCMDOT2	LCMDOT3	LCMDOT4	LCMDOT5	LCMDOT6	LCMDOT7	LCMDOT8
Type	I	I	I	I	I	I	I	I
Default	0	0	0	0	0	0	0	0
Variable	NODEID	VECID	ORIFAREA					
Type	I	I	F					
Default	0	0	0.0					

VARIABLE

DESCRIPTION

SECID	Section ID number
LCIDT	Inflator gas mixture average stagnation temperature load curve ID.
LCIDVOLR	Relative volume load curve ID.
LCIDVEL	Inflator gas mixture average velocity load curve ID.
NIDLC001	The 1 st node ID defining a local coordinate (See Remark 2).

NIDLCOO2	The 2 nd node ID defining a local coordinate (See Remark 2).
NIDLCOO3	The 3 rd node ID defining a local coordinate (See Remark 2).
LCMDOT1	The mass flow rate load curve ID of the 1 st gas in the mixture.
LCMDOTn	The mass flow rate load curve ID of the n th gas in the mixture.
LCMDOT8	The mass flow rate load curve ID of the 8 th gas in the mixture.
NODEID	The node ID(s) defining the point source(s).
VECID	The vector ID defining the direction of flow at each point source.
ORIFAREA	The orifice area at each point source.

Remarks:

1. Input from control volume analysis ($\dot{m}(t)$ and $\bar{T}_{gas}(t)$) may be used as direct input for ALE analysis. However, the user must also give a gas mixture average inlet velocity (if not available, an estimated curve would be adequate).
2. The gas mixture is assumed to have a uniform temperature ($\bar{T} \approx T_i$) and inlet velocity. They may each have a different inlet mass flow rate.
3. A brief review of concept used is presented. The total energy (e_T) is the sum of internal (e) and kinetic $\left(\frac{V^2}{2} \right)$ energies, all per unit mass.

$$e_T = e + \frac{V^2}{2}$$

$$C_v T_{stag} = C_v T + \frac{V^2}{2}$$

$$T_{stag} = T + \frac{V^2}{2C_v}$$

The distinction between stagnation and static temperatures is shown above. The gas mixture average internal energy per unit mass in terms of mixture species contribution is

$$e = \bar{C}_v \bar{T} = \sum_i C_{v_i} T_i \approx \left[\sum_i C_{v_i} \right] \bar{T}$$

If we can approximate $\bar{T} \approx T_i$, then gas mixture average static temperature is related to the mixture average internal energy per unit mass as following

*SECTION

$$\bar{T} = \frac{e}{\sum_i C_{v_i}}$$

The total mixture pressure is the sum of the partial pressures of the individual species.

$$\bar{p} = \sum_i p_i$$

The ideal gas EOS applies to each individual species (by default)

$$p_i = \rho_i (C_{p_i} - C_{v_i}) T_i$$

4. The ALE solver in LS-DYNA is dissipative (designed to deal with high gradient problems). The numerical approach used conserves momentum but not kinetic energy. Some energy may be lost during the advection step. This energy lost may result in pressure drop in the system. The amount of kinetic energy not accounted may be computed for each element. In *MAT_GAS_MIXTURE computation, the element kinetic energy is converted into internal energy, hence no kinetic energy loss. This is a simple, ad hoc, approach that is not rigorously derived for the whole system based on first principles. Therefore it is not guaranteed to apply universally to all scenarios. It is the user's responsibility to validate the model with data.
5. Since ideal gas is assumed, there is no need to define the EOS for the gases in the mixture.

Example 1:

Consider a tank test model without coupling which consists of:

- a background mesh with air (PID 1 = gas 1) initially inside that mesh (tank space), and
- the inflator gas mixture (PID 2 consisting of inflator gases 2, 3, and 4).

The mixture is represented by one AMMGID and the air by another AMMGID.

The tank internal space is simply modeled with an Eulerian mesh of the same volume. The Tank itself is not modeled thus no coupling is required. The inflator gases fill up this space mixing with the air initially inside the tank.

The background air (gas 1) is included in the gas mixture definition in this case because that air will participate in the mixing process. Only include in the mixture those gases that actually undergo mixing (gases 1, 2, 3 and 4). Note that for an airbag model, the "outside" air should not be included in the mixture (it should be defined independently) since it does not participate in the mixing inside the airbag. This is shown in the next example.

The nodes define the center of the orifices, and the vectors define the directions of flow at these orifices.

```
$...|...1...|...2...|...3...|...4...|...5...|...6...|...7...|...8
*PART
Tank background mesh, initially filled with air, allows gas mixture to flow in.
$   PID      SECID      MID      EOSID      HGID      GRAV      ADPOPT      TMID
   1         1         1         0         0         0         0         0
*SECTION_SOLID
$   SECID      ELFORM      AET
   1         11         0
$ The next card defines the properties of the gas species in the mixture.
*MAT_GAS_MIXTURE
$   MID
   1
$   Cv1      Cv2      Cv3      Cv4      Cv5      Cv6      Cv7      Cv8
  654.47    482.00    2038.30    774.64     0.0     0.0     0.0     0.0
$   Cp1      Cp2      Cp3      Cp4      Cp5      Cp6      Cp7      Cp8
  941.32    666.67    2500.00    1071.40     0.0     0.0     0.0     0.0
$ The next card specifies that gas 1 (background air) occupies PID 1 at time 0.
*INITIAL_GAS_MIXTURE
$   SID      STYPE      AMMGID      TEMP0
   1         1         1         293.00
$   RHO1      RHO2      RHO3      RHO4      RHO5      RHO6      RHO7      RHO8
  1.20E-9    0.0      0.0      0.0      0.0      0.0      0.0      0.0
*PART
The gas mixture (inlet) definition (no initial mesh required for this PID)
$   PID      SECID      MID      EOSID      HGID      GRAV      ADPOPT      TMID
   2         2         1         0         0         0         0         0
*SECTION_POINT_SOURCE_MIXTURE
$   SECID      LCIDT      NOTUSED      LCIDVEL      NIDLCOOR1      NIDLCOOR2      NIDLCOOR3
   2         1         0         5         0         0         0
$   LCMDOT1      LCMDOT2      LCMDOT3      LCMDOT4      LCMDOT5      LCMDOT6      LCMDOT7      LCMDOT8
   0         2         3         4         0         0         0         0
$   NODEID      VECTID      AREA
  24485         1         25.0
   ...
  24557         1         25.0
*ALE_MULTI-MATERIAL_GROUP
$   SID      SIDTYPE
   1         1
   2         1
*DEFINE_VECTOR
$   VECTID      XTAIL      YTAIL      ZTAIL      XHEAD      YHEAD      ZHEAD
   1         0.0      0.0      0.0      0.0      1.0      0.0
$...|...1...|...2...|...3...|...4...|...5...|...6...|...7...|...8
```

Example 2:

Consider a airbag inflation model which consists of:

- a background Eulerian mesh for air initially outside the airbag (PID 1)
- the inflator gas mixture (PID 2 consisting of inflator gases 1, 2, and 3).

The mixture is represented by one AMMGID and the air by another AMMGID.

The background air (PID 1) is NOT included in the gas mixture definition in this case because that air will NOT participate in the mixing process. Only include in the mixture those gases that actually undergo mixing (gases 1, 2, and 3). Gases 1, 2, and 3 in this example correspond to gases 2, 3, and 4 in example 1. Compare the air properties in PID 1 here to that of example 1. Note that the *INITIAL_GAS_MIXTURE card is not required to initialize the background mesh in this case.

*SECTION

```

$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|....8
*PART
Tank background mesh, initially filled with air, allows gas mixture to flow in.
$      PID      SECID      MID      EOSID      HGID      GRAV      ADPOPT      TMID
      1          1          1          0          0          0          0          0
*SECTION_SOLID
$      SECID      ELFORM      AET
      1          11          0
*MAT_NULL
$      MID      RHO      PCUT      MU      TEROD      CEROD      YM      PR
      1      1.20E-9      -1.0E-6      0.0      0.0      0.0      0.0      0.0
*EOS_IDEAL_GAS
$      EOSID      CV0      CP0      COEF1      COEF2      T0      RELVOL0
      1      654.47      941.32      0.0      0.0      293.00      1.0
$ The next card defines the properties of the gas species in the mixture.
*PART
The gas mixture (inlet) definition (no initial mesh required for this PID)
$      PID      SECID      MID      EOSID      HGID      GRAV      ADPOPT      TMID
      2          2          2          0          0          0          0          0
*SECTION_POINT_SOURCE_MIXTURE
$      SECID      LCIDT      NOTUSED      LCIDVEL      NIDLCOOR1      NIDLCOOR2      NIDLCOOR3
      2          1          0          5          0          0          0
$      LCMDOT1      LCMDOT2      LCMDOT3      LCMDOT4      LCMDOT5      LCMDOT6      LCMDOT7      LCMDOT8
      2          3          4          0          0          0          0          0
$      NODEID      VECTID      AREA
      24485          1          25.0
      ...
      24557          1          25.0
*MAT_GAS_MIXTURE
$      MID
      2
$      Cv1      Cv2      Cv3      Cv4      Cv5      Cv6      Cv7      Cv8
      482.00      2038.30      774.64      0.0      0.0      0.0      0.0      0.0
$      Cp1      Cp2      Cp3      Cp4      Cp5      Cp6      Cp7      Cp8
      666.67      2500.00      1071.40      0.0      0.0      0.0      0.0      0.0
$ The next card specifies that gas 1 (background air) occupies PID 1 at time 0.
*ALE_MULTI-MATERIAL_GROUP
$      SID      SIDTYPE
      1          1
      2          1
*DEFINE_VECTOR
$      VECTID      XTAIL      YTAIL      ZTAIL      XHEAD      YHEAD      ZHEAD
      1          0.0      0.0      0.0      0.0      1.0      0.0
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|....8

```


*SECTION

*SECTION_SHELL_{*OPTION*}

Options include:

<BLANK>

ALE

EFG

such that the keyword cards appear:

*SECTION_SHELL

*SECTION_SHELL_ALE

*SECTION_SHELL_EFG

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	0	1.0	2	0.0	0.0	0	1
Remarks		1						

Card 2 1 2 3 4 5 6 7 8

Variable	T1	T2	T3	T4	NLOC	MAREA		
Type	F	F	F	F	F	F		
Default	0.0	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

Optional Section Card for EFG option.

Also see *CONTROL_EFG.

Card 1 1 2 3 4 5 6 7 8

Variable	DX	DY						
Type	F	F						
Default	1.1	1.1						

VARIABLE

DESCRIPTION

SECID

Section ID. SECID is referenced on the *PART card and must be unique.

ELFORM

Element formulation options, see Remarks 1 and 2 below:

- EQ.1: Hughes-Liu,
- EQ.2: Belytschko-Tsay,
- EQ.3: BCIZ triangular shell,
- EQ.4: C₀ 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,

*SECTION

<u>VARIABLE</u>	<u>DESCRIPTION</u>
	EQ.10: Belytschko-Wong-Chiang, EQ.11: Fast (co-rotational) Hughes-Liu, EQ.12: Plane stress (x-y plane) , EQ.13: Plane strain (x-y plane) EQ.14: Axisymmetric solid (y-axis of symmetry) - area weighted, EQ.15: Axisymmetric solid (y-axis of symmetry) - volume weighted, EQ.16: Fully integrated shell element (very fast), EQ.17: Fully integrated DKT, triangular shell element , EQ.18: Fully integrated linear DK quadrilateral/triangular shell EQ.20: Fully integrated linear assumed strain C0 shell (See remarks). EQ.21: Fully integrated linear assumed strain C0 shell (5 DOF). EQ.22: Linear shear panel element (3 DOF per node, see remarks) EQ.31: 1 point Eulerian Navier-Stokes EQ.32: 8 point Eulerian Navier-Stokes EQ.43: Mesh-free plane strain formulation (x-y plane). EQ.44: Mesh-free axisymmetric solid formulation (y-axis of symmetry). EQ.99: Simplified linear element for time-domain vibration studies. See remark 5 below.

The type 18 element is only for linear static and normal modes. It can also be used for linear springback in sheet metal stamping.

Note that the 2D and 3D element types must not be mixed, and different types of 2D elements must not be used together. For example, 2D axisymmetric calculations can use either element types 14 or 15 but these element types must not be mixed together. Likewise, the plane strain element type must not be used with either the plane stress element or the axisymmetric element types. In 3D, the different shell elements types, i.e., 1-11 and 16, can be freely mixed together.

SHRF

Shear corection factor which scales the transverse shear stress. The shell formulations in LS-DYNA, with the exception of the BCIZ and DK elements, are based on a first order shear deformation theory that yields constant transverse shear strains which violates the condition of zero traction on the top and bottom surfaces of the shell. The shear correction factor is attempt to compensate for this error. A suggested value is 5/6 for isotropic materials. This value is incorrect for sandwich or laminated shells; consequently, laminated/sandwich shell theory is now used in some of the constitutive model.

<u>VARIABLE</u>	<u>DESCRIPTION</u>
NIP	<p>Number of through thickness integration points. Either Gauss (default) or Lobatto integration can be used. The flag for Lobatto integration can be set on the control card, *CONTROL_SHELL. The location of the Gauss and Lobatto integration points are tabulated below.</p> <ul style="list-style-type: none">EQ.0.0: set to 2 integration points for shell elements.EQ.1.0: 1 point (no bending)EQ.2.0: 2 pointEQ.3.0: 3 pointEQ.4.0: 4 pointEQ.5.0: 5 pointEQ.6.0: 6 pointEQ.7.0: 7 pointEQ.8.0: 8 pointEQ.9.0: 9 pointEQ.10.: 10 pointGT.10.: trapezoidal or user defined rule <p>Through thickness integration for the two-dimensional elements (options 12-15 above) is not meaningful; consequently, the default is equal to 1 integration point. Fully integrated two-dimensional elements are available for options 13 and 15 by setting NIP equal to a value of 4 corresponding to a 2 by 2 Gaussian quadrature. If NIP is 0 or 1 and the *MAT_SIMPLIFIED_JOHNSON_COOK model is used, then a resultant plasticity formulation is activated. NIP is always set to 1 if a constitutive model based on resultants is used.</p>
PROPT	<p>Printout option (**NOT ACTIVE**):</p> <ul style="list-style-type: none">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	<p>Quadrature rule or Integration rule ID, see *INTEGRATION_SHELL:</p> <ul style="list-style-type: none">LT.0.0: absolute value is specified rule number,EQ.0.0: Gauss/Lobatto (up to 10 points are permitted),EQ.1.0: trapezoidal, <i>not recommend for accuracy reasons</i>.
ICOMP	<p>Flag for orthotropic/anisotropic layered composite material model. This option applies to material types 22, 23, 33, 34, 36, 40, 41-50, 54-56, 58, 59, 103, 116, and 194.</p> <ul style="list-style-type: none">EQ.1: a material angle in degrees is defined for each through thickness integration point. Thus, each layer has one integration point.
SETYP	<p>2D solid element type: Defined for ELFORM 13, 14, and 15.</p> <ul style="list-style-type: none">EQ.1: LagrangianEQ.2: Eulerian (single material with voids)EQ.3: ALE
T1	<p>Shell thickness at node n_1, unless the thickness is defined on the *ELEMENT_SHELL_OPTION card.</p>
T2	<p>Shell thickness at node n_2, see comment for T1 above.</p>

*SECTION

<u>VARIABLE</u>	<u>DESCRIPTION</u>
T3	Shell thickness at node n ₃ , see comment for T1 above.
T4	Shell thickness at node n ₄ , see comment for T1 above.
NLOC	Location of reference surface (Hughes-Liu formulations #1 and #6 only. Must set IRNXX=2 in *CONTROL_SHELL): EQ. 1.0: top surface, EQ. 0.0: mid surface (default), EQ.-1.0: bottom surface.
MAREA	Non-structural mass per unit area. This is additional mass which comes from materials such as carpeting. This mass is not directly included in the time step calculation.
B1	β_1 , material angle at first integration point
B2	β_2 , material angle at second integration point
B3	β_3 , material angle at third integration point
.	.
B8	β_8 , material angle at eighth integration point
.	.
Bnip	β_{nip} , material angle at n ^{ip} th integration point
AFAC	Smoothing weight factor - Simple average: EQ.-1: turn smoothing off.
BFAC	Smoothing weight factor - Volume weighting
CFAC	Smoothing weight factor - Isoparametric
DFAC	Smoothing weight factor - Equipotential
EFAC	Smoothing weight factor - Equilibrium
START	Start time for smoothing
END	End time for smoothing
AAFAC	ALE advection factor
DX,DY	Normalized dilation parameters of the kernel function in X and Y directions. The normalized dilation parameters of the kernel function are introduced to provide the smoothness and compact support properties on the construction of the mesh-free shape functions. Values between 1.0 and 2.0 are recommended. Values smaller than 1.0 are not allowed. Larger values will increase the computation time and will sometimes result in a divergence problem. See Remark 6.

GAUSS INTEGRATION RULE					
NUMBER OF GAUSS POINT	1 POINT	2 POINT	3 POINT	4 POINT	5 POINT
# 1	.0	-.5773503	.0	-.8611363	.0
# 2		+.5773503	-.7745967	-.3399810	-.9061798
# 3			+.7745967	+.3399810	-.5384693
# 4				+.8622363	+.5384693
# 5					+.9061798
NUMBER OF GAUSS POINT	6 POINT	7 POINT	8 POINT	9 POINT	10 POINT
# 1	-.9324695	-.9491080	-.9702896	-.9681602	-.9739066
# 2	-.6612094	-.7415312	-.7966665	-.8360311	-.8650634
# 3	-.2386192	-.4058452	-.5255324	-.6133714	-.6794096
# 4	+.2386192	.0	-.1834346	-.3242534	-.4333954
# 5	+.6612094	+.4058452	+.1834346	0.0	-.1488743
# 6	+.9324695	+.7415312	+.5255324	+.3242534	+.1488743
# 7		+.9491080	+.7966665	+.6133714	+.4333954
# 8			+.9702896	+.8360311	+.6794096
# 9				+.9681602	+.8650634
# 10					+.9739066

Location of through thickness Gauss integration points. The coordinate is referenced to the shell midsurface at location 0. The inner surface of the shell is at -1 and the outer surface is at +1.

LOBATTO INTEGRATION RULE					
NUMBER OF INTEG. POINT	-	-	3 POINT	4 POINT	5 POINT
# 1			.0	-1.0	.0
# 2			-1.0	-.4472136	-1.0
# 3			+1.0	+.4472136	-.6546537
# 4				+1.0	+.6546537
# 5					+1.0
NUMBER OF INTEG. POINT	6 POINT	7 POINT	8 POINT	9 POINT	10 POINT
# 1	-1.0	-1.0	-1.0	-1.0	-1.0
# 2	-.7650553	-.8302239	-.8717401	-.8997580	-.9195339
# 3	-.2852315	-.4688488	-.5917002	-.6771863	-.7387739
# 4	+.2852315	.0	-.2092992	-.3631175	-.4779249
# 5	+.7650553	+.4688488	+.2092992	.0	-.1652790

*SECTION

# 6	+1.0	+ .8302239	+ .5917002	+ .3631175	+ .1652790
# 7		+1.0	+ .8717401	+ .6771863	+ .4779249
# 8			+1.0	+ .8997580	+ .7387739
# 9				+1.0	+ .9195339
# 10					+1.0

Location of through thickness Lobatto integration points. The coordinate is referenced to the shell midsurface at location 0. The inner surface of the shell is at -1 and the outer surface is at +1.

Remarks:

1. Element formulations 31 and 32 are used exclusively with the CFD option which requires ISOLTYP=4 on the *CONTROL_SOLUTION card. In this case, ELFORM=31 is used with INSOL=1 and ELFORM=32 is used with INSOL=3 on the *CONTROL_CFD_GENERAL card. Note that selection of the element formulation is automatic based on the value of INSOL for the CFD solver.
2. For implicit calculations the following element choices are implemented:
 - EQ.1: Hughes-Liu,
 - EQ.2: Belytschko-Tsay (default),
 - EQ.6: S/R Hughes-Liu ,
 - EQ.10: Belytschko-Wong-Chiang,
 - EQ.12: Plane stress (x-y plane) ,
 - EQ.13: Plane strain (x-y plane)
 - EQ.15: Axisymmetric solid (y-axis of symmetry) - volume weighted,
 - EQ.16: Fully integrated shell element ,
 - EQ.17: Fully integrated DKT, triangular shell element ,
 - EQ.18: Taylor 4-node quadrilateral and 3-node triangle (linear only)
 - EQ.20: Wilson 3 & 4-node DSE quadrilateral (linear only)
 - EQ.21: Fully integrated linear assumed strain C0 shell (5 DOF).
 - EQ.22: Linear shear panel element (3 DOF per node)
 - EQ.31: 1 point Eulerian Navier-Stokes
 - EQ.32: 8 point Eulerian Navier-Stokes

If another element formulation is requested, LS-DYNA will substitute one of the above in place of the one chosen.

3. The linear elements consist of an assembly of membrane and plate elements. The elements have six d.o.f. per node and can therefore be connected to beams, or used in complex shell surface intersections. All elements possess the required zero energy rigid body modes and have exact constant strain and curvature representation, i.e. they pass all the first order patch tests. In addition, the elements have behavior approaching linear bending (cubic displacement) in the plate-bending configuration.
 - a. The membrane component of all elements is based on an 8-node/6-node isoparametric mother element which incorporates nodal in-plane rotations through cubic displacement constraints of the sides [Taylor, 1987; Wilson, 2000].

- b. The plate component of element 18 is based on the Discrete Kirchhoff Quadrilateral (DKQ) [Batoz, 1982]. Because the Kirchhoff assumption is enforced, the DKQ is transverse shear rigid and can only be used for thin shells. No transverse shear stress information is available. The triangle is based on a degeneration of the DKQ. This element sometimes gives slightly lower eigenvalues when compared with element type 20.
 - c. The plate component of element 20 is based on the 8-node serendipity element. At the mid-side, the parallel rotations and transverse displacements are constrained and the normal rotations are condensed to yield a 4-node element. The element is based on thick plate theory and is recommended for thick and thin plates.
 - d. The quadrilateral elements contain a warpage correction using rigid links.
 - e. The membrane component of element 18 has a zero energy mode associated with the in-plane rotations. This is automatically suppressed in a non-flat shell by the plate stiffness of the adjacent elements. Element 20 has no spurious zero energy modes.
4. The linear shear panel element resists tangential in plane shearing along the four edges and can only be used with the elastic material constants of *MAT_ELASTIC. Membrane forces and out-of-plane loads are not resisted.
 5. Element type 99 is intended for vibration studies carried out in the time domain. These models may have very large numbers of elements and may be run for relatively long durations. The purpose of this element is to achieve substantial CPU savings. This is achieved by imposing strict limitations on the range of applicability, thereby simplifying the calculations:
 - Elements must be rectangular; all edges must be parallel to the global X, Y or Z axis;
 - Small displacement, small strain, negligible rigid body rotation;
 - Elastic material only

If these conditions are satisfied, the performance of the element is similar to the fully integrated shell (ELFORM=16) but at less CPU cost than the default Belyschko-Tsay shell element (ELFORM=2). Single element torsion and in-plane bending modes are included; meshing guidelines are the same as for fully integrated shell elements.

No damping is included in the element formulation (e.g. volumetric damping). It is strongly recommended that damping be applied, e.g. *DAMPING_PART_MASS or *DAMPING_FREQUENCY_RANGE.

6. The current mesh-free formulation uses the finite element quadrilateral elements as the background mesh to identify the mesh-free part in the computation. The automatic sorting of finite element triangular and quadrilateral elements as the background mesh for the mesh-free computation will be updated later on.

***SECTION_SOLID_{OPTION}**

Options include:

<BLANK>

ALE**EFG**

such that the keyword cards appear:

SECTION_SOLID**SECTION_SOLID_ALE*****SECTION_SOLID_EFG**

Purpose: Define section properties for solid continuum and fluid elements.

Card 1 define for all options

Card 1 1 2 3 4 5 6 7 8

Variable	SECID	ELFORM	AET					
Type	I	I	I					
Remark		1, 2						

Card 2 define only for the ALE option.

Also see *ALE_SMOOTHING for the smoothing definition.

Cards 2 1 2 3 4 5 6 7 8

Variable	AFAC	BFAC	CFAC	DFAC	START	END	AAFAC	
Type	F	F	F	F	F	F	F	

AFAC	Smoothing weight factor - Simple average: EQ.-1: turn smoothing off.
BFAC	Smoothing weight factor - Volume weighting
CFAC	Smoothing weight factor - Isoparametric
DFAC	Smoothing weight factor - Equipotential
START	Start time for smoothing
END	End time for smoothing
AAFAC	ALE advection factor
DX, DY, DZ	Normalized dilation parameters of the kernel function in X, Y and Z directions. The normalized dilation parameters of the kernel function are introduced to provide the smoothness and compact support properties on the construction of the mesh-free shape functions. Values between 1.0 and 1.5 are recommended. Values smaller than 1.0 are not allowed. Larger values will increase the computation time and will sometimes result in a divergence problem. See Remark 7.

*SECTION

Remarks:

1. Element formulations 31 and 32 are used exclusively with the CFD option which requires ISOLTYP=4 on the *CONTROL_SOLUTION card. In this case, ELFORM=31 is used with INSOL=1 and ELFORM=32 is used with INSOL=3 on the *CONTROL_CFD_GENERAL card. Note that selection of the element formulation is automatic based on the value of INSOL for the CFD solver.
2. The keyword *CONTROL_SOLID activates automatic sorting of tetrahedron and pentahedron elements into type 10 and 15 element formulation, respectively. These latter elements are far more stable than the degenerate solid element. The sorting is performed internally and is transparent to the user.
3. For implicit calculations the following element choices are implemented:
 - EQ.1: constant stress solid element,
 - EQ.2: fully integrated S/R solid. See remark 5 below,
 - EQ.3: fully integrated 8 node solid with rotational DOFs,
 - EQ.4: fully integrated S/R 4 node tetrahedron with rotational DOFs,
 - EQ.10: 1 point tetrahedron.
 - EQ.15: 2 point pentahedron element.
 - EQ.16: 5 point 10 noded tetrahedron
 - EQ.18: 8 point enhanced strain solid element for linear statics only,
 - EQ.31: 1 point Eulerian Navier-Stokes
 - EQ.32: 8 point Eulerian Navier-Stokes

If another element formulation is requested, LS-DYNA will substitute, when possible, one of the above in place of the one chosen. The type 1 element, constant stress, is generally much more accurate than the type 2 element, the selective reduced integrated element for implicit problems.

4. Element formulations 0 and 9, applicable only to *MAT_MODIFIED_HONEYCOMB, behave essentially as nonlinear springs so as to permit severe distortions sometimes seen in honeycomb materials. In formulation 0, the local coordinate system follows the element rotation whereas in formulation 9, the local coordinate system is based on axes passing through the centroids of the element faces. Formulation 0 is preferred for severe shear deformation where the barrier is fixed in space. If the barrier is attached to a moving body, which can rotate, then formulation 9 is usually preferred.
5. The selective reduced integrated solid element, element type 2, assumes that pressure is constant throughout the element to avoid pressure locking during nearly incompressible flow. However, if the element aspect ratios are poor, shear locking will lead to an excessively stiff response. A better choice, given poor aspect ratios, is the one point solid element which work well for implicit and explicit calculations. For linear statics, the type 18 enhanced strain element works well with poor aspect ratios. Please note that highly distorted elements should always be avoided since excessive stiffness will still be observed even in the enhanced strain formulations.
6. Element type 99 is intended for vibration studies carried out in the time domain. These models may have very large numbers of elements and may be run for relatively long durations. The purpose of this element is to achieve substantial CPU savings. This is achieved by imposing strict limitations on the range of applicability, thereby simplifying the calculations:
 - Elements must be cuboid; all edges must parallel to the global X, Y or Z axis;

*SECTION

*SECTION_SPH

Purpose: Define section properties for SPH particles.

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	SECID	CSLH	HMIN	HMAX	SPHINI	DEATH	START	
Type	I	F	F	F	F	F	F	
Default	none	1.2	0.2	2.0	0.0	1.e20	0.0	

<u>VARIABLE</u>	<u>DESCRIPTION</u>
SECID	Section ID. SECID is referenced on the *PART card and must be unique.
CSLH	Constant applied to the smoothing length of the particles. The default value applies for most problems. Values between 1.05 and 1.3 are acceptable. Taking a value less than 1 is inadmissible. Values larger than 1.3 will increase the computational time. The default value is recommended.
HMIN	Scale factor for the minimum smoothing length (See Remark 1)
HMAX	Scale factor for the maximum smoothing length (See Remark 1)
SPHINI	Optional initial smoothing length (overrides true smoothing length). This option applies to avoid LS-DYNA to calculate the smoothing length during initialization. In this case, the variable CSLH doesn't apply.
DEATH	Time imposed SPH approximation is stopped.
START	Time imposed SPH approximation is activated.

Remarks:

1. The SPH processor in LS-DYNA uses a variable smoothing length. LS-DYNA computes the initial smoothing length, h_0 , for each SPH part by taking the maximum of the minimum distance between every particle. Every particle has its own smoothing length which varies in time according to the following equation:

$$\frac{d}{dt}(h(t)) = h(t)div(v)$$

$h(t)$ is the smoothing length, $div(v)$ is the divergence of the flow. The smoothing length increases when particles separate from each other and reduces when the concentration of

particles is important. It varies to keep the same number of particles in the neighborhood. The smoothing length varies between the minimum and maximum values

$$HMIN * h_0 < h(t) < HMAX * h_0$$

Defining a value of 1 for HMIN and 1 for HMAX will result in a constant smoothing length in time and space.

2. SPH is implemented for explicit applications.

*SECTION

*SECTION_TSHLL

Purpose: Define section properties for thick shell elements.

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	SECID	ELFORM	SHRF	NIP	PROPT	QR	ICOMP	
Type	I	I	F	F	F	F	I	
Default	none	1	1.0	2	1	0	0	

Optional Section Cards if ICOMP=1 define NIP angles putting 8 on each card.

Cards 2,3,.. 1 2 3 4 5 6 7 8

Variable	B1	B2	B3	B4	B5	B6	B7	B8
Type	F	F	F	F	F	F	F	F

VARIABLE

DESCRIPTION

SECID Section ID. SECID is referenced on the *PART card and must be unique.

ELFORM Element formulation:
 EQ.1: one point reduced integration (default),
 EQ.2: selective reduced 2×2 in plane integration.
 EQ.3: assumed strain 2×2 in plane integration, see remark below.

SHRF Shear factor. A value of 5/6 is recommended.

NIP Number of through shell thickness integration points:
 EQ.0: set to 2 integration points.

PROPT Printout option:
 EQ.1.0: average resultants and fiber lengths,
 EQ.2.0: resultants at plan points and fiber lengths,
 EQ.3.0: resultants, stresses at all points, fiber lengths.

<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	β_1 , material angle at first integration point. The same procedure for determining material directions is use for thick shells that is used for the 4 node quadrilateral shell.
B2	β_2 , material angle at second integration point
B3	β_3 , material angle at third integration point
.	.
.	.
.	.
B8	β_8 , material angle at eighth integration point
.	.
Bnip	β_{nip} , material angle at niph integration point

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

Remarks:

1. Thick shell formulation type 3 uses a full three-dimensional stress update rather than the two-dimensional plane stress update of types 1 and 2. The type 3 element is distortion sensitive and should not be used in situations where the elements are badly shaped. With element types 1 and 2 a single element through the thickness will capture bending response, but with element type 3 two are recommended to avoid excessive softness.
2. These elements are available for implicit applications.

*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_MULTI-MATERIAL_GROUP_LIST**

***SET_NODE_{OPTION}**

***SET_PART_{OPTION}**

***SET_SEGMENT_{OPTION}**

***SET_SHELL_{OPTION}**

***SET_SOLID_{OPTION}**

***SET_TSHELL_{OPTION}**

An additional option **_TITLE** may be appended to all the ***SET** keywords. If this option is used then an addition line is read for each section in 80a format which can be used to describe the set. At present LS-DYNA does not make use of the title. Inclusion of titles gives greater clarity to input decks.

The **GENERAL** option is available for set definitions. In this option, the commands are executed in the order defined. For example, the delete option cannot delete a node or element unless the node or element was previously added via a command such as **BOX** or **ALL**.

*SET

*SET_BEAM_{*OPTION*}

Available options include:

<BLANK>

GENERATE

GENERAL

The last option, GENERATE, will generate a block of beam element ID's between a starting ID and an ending ID. An arbitrary number of blocks can be specified to define the set.

Purpose: Define a set of beam elements.

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	SID							
Type	I							
Default	none							

Cards 2, 3, 4, ... (*OPTION*=none) (The next “*” card terminates the input.)

 1 2 3 4 5 6 7 8

Variable	K1	K2	K3	K4	K5	K6	K7	K8
Type	I	I	I	I	I	I	I	I

Cards 2, 3, 4, ... (*OPTION*=GENERATE) (The next “*” card terminates the input.)

 1 2 3 4 5 6 7 8

Variable	B1BEG	B1END	B2BEG	B2END	B3BEG	B3END	B4BEG	B4END
Type	I	I	I	I	I	I	I	I

Cards 2, 3, 4, ... (OPTION=GENERAL) (The next “*” card terminates the input.) This set is a combination of a series of options: ALL, ELEM, DELEM, PART, DPART, BOX, and DBOX.

	1	2	3	4	5	6	7	8
Variable	OPTION	E1	E2	E3	E4	E5	E6	E7
Type	A	I	I	I	I	I	I	I

<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.
OPTION	Option for GENERAL. See table below.
E1,...,E7	Specified entity. Each card must have the option specified. See table below.

*SET

OPTION	ENTITY (define up to 7)	FUNCTION
ALL		All beam elements will be included in the set.
ELEM	e1, e2, e3, e4, e5, e6, e7	Elements e1, e2, e3, ... will be included.
DELEM	e1, e2, e3, e4, e5, e6, e7	Elements e1, e2, e3, ... previously added will be excluded.
PART	p1, p2, p3, p4, p5, p6, p7	Elements of parts p1, p2, p3, ... will be included.
DPART	p1, p2, p3, p4, p5, p6, p7	Elements of parts p1, p2, p3, ... previously added will be excluded.
BOX	b1, b2, b3, b4, b5, b6, b7	Elements inside boxes b1, b2, ... will be included.
DBOX	b1, b2, b3, b4, b5, b6, b7	Elements inside boxes b1, b2, ... previously added will be excluded.

***SET_DISCRETE_{OPTION}**

Available options include:

<BLANK>

GENERATE

GENERAL

The last option, GENERATE, will generate a block of discrete element ID's between a starting ID and an ending ID. An arbitrary number of blocks can be specified to define the set.

Purpose: Define a set of discrete elements.

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable	SID							
Type	I							
Default	none							

Cards 2, 3, 4, ... (OPTION=none) (The next "*" card terminates the input.)

1 2 3 4 5 6 7 8

Variable	K1	K2	K3	K4	K5	K6	K7	K8
Type	I	I	I	I	I	I	I	I

*SET

Cards 2, 3, 4, ... (OPTION=GENERATE) (The next “*” card terminates the input.)

1 2 3 4 5 6 7 8

Variable	B1BEG	B1END	B2BEG	B2END	B3BEG	B3END	B4BEG	B4END
Type	I	I	I	I	I	I	I	I

**Cards 2, 3, 4, ... (OPTION=GENERAL) (The next “*” card terminates the input.)
This set is a combination of a series of options: ALL, ELEM, DELEM, PART, DPART, BOX, and DBOX.**

1 2 3 4 5 6 7 8

Variable	OPTION	E1	E2	E3	E4	E5	E6	E7
Type	A	I	I	I	I	I	I	I

<u>VARIABLE</u>	<u>DESCRIPTION</u>
SID	Set ID
K1	First discrete element
K2	Second discrete element
.	.
.	.
KNUM	Last discrete element
BNBEG	First discrete element ID in block N.
BNEND	Last discrete element ID in block N. All defined ID's between and including BNBEG to BNEND are added to the set. These sets are generated after all input is read so that gaps in the element numbering are not a problem. BNBEG and BNEND may simply be limits on the ID's and not element ID's.
OPTION	Option for GENERAL. See table below.
E1, ..., E7	Specified entity. Each card must have the option specified. See table below.

OPTION	ENTITY (define up to 7)	FUNCTION
ALL		All discrete elements will be included in the set.
ELEM	e1, e2, e3, e4, e5, e6, e7	Elements e1, e2, e3, ... will be included.
DELEM	e1, e2, e3, e4, e5, e6, e7	Elements e1, e2, e3, ... previously added will be excluded.
PART	p1, p2, p3, p4, p5, p6, p7	Elements of parts p1, p2, p3, ... will be included.
DPART	p1, p2, p3, p4, p5, p6, p7	Elements of parts p1, p2, p3, ... previously added will be excluded.
BOX	b1, b2, b3, b4, b5, b6, b7	Elements inside boxes b1, b2, ... will be included.
DBOX	b1, b2, b3, b4, b5, b6, b7	Elements inside boxes b1, b2, ... previously added will be excluded.

*SET

*SET_MULTI-MATERIAL_GROUP_LIST

Purpose: This command defines an ALE multi-material set ID (AMMSID) which contains a collection of one or more ALE multi-material group ID(s) (AMMGID). This provides a means for selecting any specific ALE multi-material(s). Application includes, for example, a selection of any particular fluid(s) to be coupled to in a fluid-structure interaction.

Card Format

1 2 3 4 5 6 7 8

Variable	AMMSID							
Type	I							
Default	0							
Variable	AMMGID1	AMMGID2	AMMGID3	AMMGID4	AMMGID5	AMMGID6	AMMGID7	AMMGID8
Type	I	I	I	I	I	I	I	I
Default	0	0	0	0	0	0	0	0

VARIABLE

DESCRIPTION

AMMSID	An ALE multi-material set ID (AMMSID) which contains a collection of one or more ALE multi-material group ID(s) (AMMGID).
AMMGID1	The 1st ALE multi-material group ID (AMMGID=1) defined by the 1st line of the *ALE_MULTI-MATERIAL_GROUP card.
...	...
AMMGID8	The 8th ALE multi-material group ID (AMMGID=8) defined by the 8th line of the *ALE_MULTI-MATERIAL_GROUP card.

Remark:

1. Please refer to an example in the *CONSTRAINED_LAGRANGE_IN_SOLID section.

***SET_NODE_{OPTION}**

Available options include:

<BLANK>

LIST

COLUMN

LIST_GENERATE

GENERAL

The option, LIST_GENERATE, will generate a block of node ID's between a starting nodal ID number and an ending nodal ID number. An arbitrary number of blocks can be specified to define the set.

Purpose: Define a nodal set with some identical or unique attributes.

Card Format

	1	2	3	4	5	6	7	8
Variable	SID	DA1	DA2	DA3	DA4			
Type	I	F	F	F	F			
Default	none	0.	0.	0.	0.			
Remark		1	1	1	1			

Cards 2, 3, 4, ... (OPTION=LIST) (The next "*" card terminates the input.)

	1	2	3	4	5	6	7	8
Variable	NID1	NID2	NID3	NID4	NID5	NID6	NID7	NID8
Type	I	I	I	I	I	I	I	I

*SET

Cards 2, 3, 4, ... (*OPTION=COLUMN*) (The next “*” card terminates the input.)

	1	2	3	4	5	6	7	8
Variable	NID	A1	A2	A3	A4			
Type	I	F	F	F	F			
Remark		2	2	2	2			

Cards 2, 3, 4, ... (*OPTION=LIST_GENERATE*) (The next “*” card terminates the input.)

	1	2	3	4	5	6	7	8
Variable	B1BEG	B1END	B2BEG	B2END	B3BEG	B3END	B4BEG	B4END
Type	I	I	I	I	I	I	I	I

Cards 2, 3, 4, ... (*OPTION=GENERAL*) (The next “*” card terminates the input.)
This set is a combination of a series of options: ALL, NODE, DNODE, PART, DPART, BOX, and DBOX.

	1	2	3	4	5	6	7	8
Variable	OPTION	E1	E2	E3	E4	E5	E6	E7
Type	A	I	I	I	I	I	I	I

VARIABLE

DESCRIPTION

SID	Set identification. All node sets should have a unique set ID.
DA1	First nodal attribute default value, see remark 1 below.
DA2	Second nodal attribute default value
DA3	Third nodal attribute default value

VARIABLE	DESCRIPTION
DA4	Fourth nodal attribute default value
NIDN	Node ID n
NID	Nodal ID
A1	First nodal attribute, see remark 2 below.
A2	Second nodal attribute
A3	Third nodal attribute
A4	Fourth nodal attribute
BNBEG	First node ID in block N.
BNEND	Last node ID in block N. All defined ID's between and including BNBEG to BNEND are added to the set. These sets are generated after all input is read so that gaps in the node numbering are not a problem. BNBEG and BNEND may simply be limits on the ID's and not nodal ID's.
OPTION	Option for GENERAL. See table below.
E1,...,E7	Specified entity. Each card must have the option specified. See table below.

OPTION	ENTITY (define up to 7)	FUNCTION
ALL		All nodes will be included in the set.
NODE	n1, n2, n3, n4, n5, n6, n7	Nodes n1, n2, n3, ... will be included.
DNODE	n1, n2, n3, n4, n5, n6, n7	Nodes n1, n2, n3, ... previously added will be excluded.
PART	p1, p2, p3, p4, p5, p6, p7	Nodes of parts p1, p2, p3, ... will be included.
DPART	p1, p2, p3, p4, p5, p6, p7	Nodes of parts p1, p2, p3, ... previously added will be excluded.
BOX	b1, b2, b3, b4, b5, b6, b7	Nodes inside boxes b1, b2, b3, ... will be included.
DBOX	b1, b2, b3, b4, b5, b6, b7	Nodes inside boxes b1, b2, b3, ... previously added will be excluded.

*SET

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:

<BLANK>

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_{OPTION}

Available options include:

<BLANK>

GENERAL

Purpose: Define a set of quadrilateral and triangular segments with optional identical or unique attributes.

Card Format

1 2 3 4 5 6 7 8

Variable	SID	DA1	DA2	DA3	DA4			
Type	I	F	F	F	F			
Default	none	0.	0.	0.	0.			
Remarks		1	1	1	1			

Cards 2, 3, 4, ... (No option is specified) (The next “*” card terminates the input.)

1 2 3 4 5 6 7 8

Variable	N1	N2	N3	N4	A1	A2	A3	A4
Type	I	I	I	I	F	F	F	F
Remarks				2	3	3	3	3

**Cards 2, 3, 4, ... (OPTION=GENERAL) (The next “*” card terminates the input.)
This set is a combination of a series of options listed in the table defined below.**

	1	2	3	4	5	6	7	8
Variable	OPTION	E1	E2	E3	E4	E5	E6	E7
Type	A	I	I	I	I or F	I or F	I or F	I or F

VARIABLE

DESCRIPTION

SID	Set ID. All segment sets should have a unique set ID.
DA1	First segment attribute default value, see remark 1 below.
DA2	Second segment attribute default value
DA3	Third segment attribute default value
DA4	Fourth segment attribute default value
N1	Nodal point n ₁
N2	Nodal point n ₂
N3	Nodal point n ₃
N4	Nodal point n ₄ , see remark 2 below.
A1	First segment attribute, see remark 3 below.
A2	Second segment attribute
A3	Third segment attribute
A4	Fourth segment attribute
NFLS	Normal failure stress
SFLS	Shear failure stress. Failure criterion:
OPTION	Option for GENERAL. See table below.
E1,...,E7	Specified entity. Each card must have an option specified. See table below.

*SET

FORMAT (A10,3I10, 4F10.0)		
OPTION	ENTITIES + ATTRIBUTES	FUNCTION
BOX	b1, b2, b3, a1, a2, a3, a4	Generate segments inside box ID bi, i=1,,2,3. For shell elements one segment per shell is generated. For solid elements only those segments wrapping the solid part and pointing outward from the part will be generated.
BOX_SHELL	b1, b2, b3, a1, a2, a3, a4	Generate segments inside box ID bi, i=1,,2,3. The segments are only generated for shell elements. One segment per shell is generated.
BOX_SLDIO	b1, b2, b3, a1, a2, a3, a4	Generate segments inside box ID bi, i=1,,2,3. Both exterior segments and inter-element segments are generated.
BOX_SOLID	b1, b2, b3, a1, a2, a3, a4	Generate segments inside box ID bi, i=1,,2,3. The segments are only generated for exterior solid elements
PART	p1, p2, p3, a1, a2, a3, a4	Generate segments of parts p1, p2, p3 with attributes a1-a4. For shell elements one segment per shell is generated. For solid elements only those segments wrapping the solid part and pointing outward from the part will be generated.
PART_IO	p1, p2, p3, a1, a2, a3, a4	Generate segments of parts p1, p2, p3 with attributes a1-a4. Same as the PART option above except that inter-element segments inside parts will be generated as well. This option is sometimes useful for single surface contact of solid elements to prevent negative volumes caused by inversion.

FORMAT (A10,7I10)		
DBOX	b1, b2, b3, b4, b5, b6, b7	Segments inside boxes b1, b2, ... previously added will be excluded.
DBOX_SHELL	b1, b2, b3, b4, b5, b6, b7	Shell related segments inside boxes b1, b2, ... previously added will be excluded.
DBOX_SOLID	b1, b2, b3, b4, b5, b6, b7	Solid related segments inside boxes b1, b2, ... previously added will be excluded.
DPART	p1, p2, p3, p4, p5, p6, p7	Segments of parts p1, p2, p3, ... previously added will be excluded.
DSEG	n1, n2, n3, n4	Segments with node ID's n1,n2, n3, and n4 previously added will be deleted. The numbering sequence is irrelevant.
SEG	n1, n2, n3, n4	Create segment with node ID's n1,n2, n3, and n4.t.

Remarks:

1. Segment attributes can be assigned for some input types. For example, for the contact options, the attributes for the SLAVE surface are:

DA1=NFLS Normal failure stress, *CONTACT_TIEBREAK_SURFACE_contact only,

DA2=SFLS Shear failure stress, *CONTACT_TIEBREAK_SURFACE_contact only,

DA3=FSF Coulomb friction scale factor,

DA4=VSF Viscous friction scale factor,

and the attributes for the MASTER surface are:

DA1=FSF Coulomb friction scale factor,

DA2=VSF Viscous friction scale factor.

For airbags, see *AIRBAG, a time delay, DA1=T1, can be defined before pressure begins to act on a segment along with a time delay, DA2=T2, before full pressure is applied to the segment, (default T2=T1), and for the constraint option,

2. To define a triangular segment make n4 equal to n3.
3. The default segment attributes can be overridden on these cards, otherwise, A1=DA1, etc.

*SET

*SET_SHELL_{OPTION}

Available options include:

<BLANK>

LIST

COLUMN

LIST_GENERATE

GENERAL

The last option will generate a block of shell ID's between a starting shell ID number and an ending ID number. An arbitrary number of blocks can be specified to define the shell set.

Purpose: Define a set of shell elements with optional identical or unique attributes.

Card Format

	1	2	3	4	5	6	7	8
Variable	SID	DA1	DA2	DA3	DA4			
Type	I	F	F	F	F			
Default	none	0.	0.	0.	0.			
Remarks		1	1	1	1			

Card 2, 3, 4, ... (OPTION=LIST) (The next "*" card terminates the input.)

	1	2	3	4	5	6	7	8
Variable	EID1	EID2	EID3	EID4	EID5	EID6	EID7	EID8
Type	I	I	I	I	I	I	I	I
Remarks	2	2	2	2	2	2	2	2

Card 2, 3, 4, ... (OPTION=COLUMN) (The next “*” card terminates the input.)

	1	2	3	4	5	6	7	8
Variable	EID	A1	A2	A3	A4			
Type	I	F	F	F	F			
Remarks		3	3	3	3			

Cards 2, 3, 4, ... (OPTION=LIST_GENERATE) (The next “*” card terminates the input.)

	1	2	3	4	5	6	7	8
Variable	B1BEG	B1END	B2BEG	B2END	B3BEG	B3END	B4BEG	B4END
Type	I	I	I	I	I	I	I	I

Cards 2, 3, 4, ... (OPTION=GENERAL) (The next “*” card terminates the input.)
This set is a combination of a series of options: ALL, ELEM, DELEM, PART, DPART, BOX, and DBOX.

	1	2	3	4	5	6	7	8
Variable	OPTION	E1	E2	E3	E4	E5	E6	E7
Type	A	I	I	I	I	I	I	I

VARIABLE

DESCRIPTION

SID	Set ID. All shell sets should have a unique set ID.
DA1	First attribute default value, see remark 1.
DA2	Second attribute default value
DA3	Third attribute default value
DA4	Fourth attribute default value

*SET

VARIABLE	DESCRIPTION
EID1	First shell element ID, see remark 2.
EID2	Second shell element ID
.	.
.	.
EID	Element ID
A1	First attribute
A2	Second attribute
A3	Third attribute
A4	Fourth attribute
BNBEG	First shell ID in shell block N.
BNEND	Last shell ID in block N. All defined ID's between and including BNBEG to BNEND are added to the set. These sets are generated after all input is read so that gaps in the element numbering are not a problem. BNBEG and BNEND may simply be limits on the ID's and not element ID's.
OPTION	Option for GENERAL. See table below.
E1,...,E7	Specified entity. Each card must have the option specified. See table below.

OPTION	ENTITY (define up to 7)	FUNCTION
ALL		All shell elements will be included in the set.
ELEM	e1, e2, e3, e4, e5, e6, e7	Elements e1, e2, e3, ... will be included.
DELEM	e1, e2, e3, e4, e5, e6, e7	Elements e1, e2, e3, ... previously added will be excluded.
PART	p1, p2, p3, p4, p5, p6, p7	Elements of parts p1, p2, p3, ... will be included.
DPART	p1, p2, p3, p4, p5, p6, p7	Elements of parts p1, p2, p3, ... previously added will be excluded.
BOX	b1, b2, b3, b4, b5, b6, b7	Elements inside boxes b1, b2, ... will be included.
DBOX	b1, b2, b3, b4, b5, b6, b7	Elements inside boxes b1, b2, ... previously added will be excluded.

Remarks:

1. Shell attributes can be assigned for some input types. For example, for the contact options, the attributes for the SLAVE surface are:

DA1=NFLS Normal failure stress, *CONTACT_TIEBREAK_SURFACE_contact only,

DA2=SFLS Shear failure stress, *CONTACT_TIEBREAK_SURFACE_contact only,

DA3=FSF Coulomb friction scale factor,

DA4=VSF Viscous friction scale factor,

and the attributes for the MASTER surface are:

DA1=FSF Coulomb friction scale factor,

DA2=VSF Viscous friction scale factor.

2. The default attributes are taken.
3. The default shell attributes can be overridden on these cards; otherwise, A1=DA1, etc.

*SET

***SET_SOLID_{OPTION}**

Available options include:

<BLANK>

GENERATE

GENERAL

The last option, GENERATE, will generate a block of solid element ID's between a starting ID and an ending ID. An arbitrary number of blocks can be specified to define the set.

Purpose: Define a set of solid elements.

Card Format

1 2 3 4 5 6 7 8

Variable	SID							
Type	I							
Default	none							

Cards 2, 3, 4, ... (OPTION=none) (The next “*” card terminates the input.)

1 2 3 4 5 6 7 8

Variable	K1	K2	K3	K4	K5	K6	K7	K8
Type	I	I	I	I	I	I	I	I

Cards 2, 3, 4, ... (OPTION=GENERATE) (The next “*” card terminates the input.)

1 2 3 4 5 6 7 8

Variable	B1BEG	B1END	B2BEG	B2END	B3BEG	B3END	B4BEG	B4END
Type	I	I	I	I	I	I	I	I

Cards 2, 3, 4, ... (OPTION=GENERAL) (The next “*” card terminates the input.)
This set is a combination of a series of options: ALL, ELEM, DELEM, PART, DPART, BOX, and DBOX.

1 2 3 4 5 6 7 8

Variable	OPTION	E1	E2	E3	E4	E5	E6	E7
Type	A	I	I	I	I	I	I	I

VARIABLE

DESCRIPTION

SID	Set ID. All solid sets should have a unique set ID.
K1	First element ID
K2	Second element ID
.	.
.	.
K8	Eighth element ID
BNBEG	First solid element ID in block N.
BNEND	Last solid element ID in block N. All defined ID's between and including BNBEG to BNEND are added to the set. These sets are generated after all input is read so that gaps in the element numbering are not a problem. BNBEG and BNEND may simply be limits on the ID's and not element ID's.
OPTION	Option for GENERAL. See table below.
E1,...,E7	Specified entity. Each card must have the option specified. See table below.

*SET

OPTION	ENTITY (define up to 7)	FUNCTION
ALL		All solid elements will be included in the set.
ELEM	e1, e2, e3, e4, e5, e6, e7	Elements e1, e2, e3, ... will be included.
DELEM	e1, e2, e3, e4, e5, e6, e7	Elements e1, e2, e3, ... previously added will be excluded.
PART	p1, p2, p3, p4, p5, p6, p7	Elements of parts p1, p2, p3, ... will be included.
DPART	p1, p2, p3, p4, p5, p6, p7	Elements of parts p1, p2, p3, ...previously added will be excluded.
BOX	b1, b2, b3, b4, b5, b6, b7	Elements inside boxes b1, b2, ... will be included.
DBOX	b1, b2, b3, b4, b5, b6, b7	Elements inside boxes b1, b2, ... previously added will be excluded.

***SET_TSHELL_{OPTION}**

Available options include:

<BLANK>

GENERATE

GENERAL

The last option, GENERATE, will generate a block of thick shell element ID's between a starting ID and an ending ID. An arbitrary number of blocks can be specified to define the set.

Purpose: Define a set of thick shell elements.

Card Format

1 2 3 4 5 6 7 8

Variable	SID							
Type	I							
Default	none							

Cards 2, 3, 4, ... (OPTION=none) (The next "*" card terminates the input.)

1 2 3 4 5 6 7 8

Variable	K1	K2	K3	K4	K5	K6	K7	K8
Type	I	I	I	I	I	I	I	I

*SET

Cards 2, 3, 4, ... (*OPTION=GENERATE*) (The next “*” card terminates the input.)

1 2 3 4 5 6 7 8

Variable	B1BEG	B1END	B2BEG	B2END	B3BEG	B3END	B4BEG	B4END
Type	I	I	I	I	I	I	I	I

Cards 2, 3, 4, ... (*OPTION=GENERAL*) (The next “*” card terminates the input.)
This set is a combination of a series of options: ALL, ELEM, DELEM, PART, DPART, BOX, and DBOX.

1 2 3 4 5 6 7 8

Variable	OPTION	E1	E2	E3	E4	E5	E6	E7
Type	A	I	I	I	I	I	I	I

<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.
OPTION	Option for GENERAL. See table below.
E1,...,E7	Specified entity. Each card must have the option specified. See table below.

OPTION	ENTITY (define up to 7)	FUNCTION
ALL		All thick shell elements will be included in the set.
ELEM	e1, e2, e3, e4, e5, e6, e7	Elements e1, e2, e3, ... will be included.
DELEM	e1, e2, e3, e4, e5, e6, e7	Elements e1, e2, e3, ... previously added will be excluded.
PART	p1, p2, p3, p4, p5, p6, p7	Elements of parts p1, p2, p3, ... will be included.
DPART	p1, p2, p3, p4, p5, p6, p7	Elements of parts p1, p2, p3, ... previously added will be excluded.
BOX	b1, b2, b3, b4, b5, b6, b7	Elements inside boxes b1, b2, ... will be included.
DBOX	b1, b2, b3, b4, b5, b6, b7	Elements inside boxes b1, b2, ... previously added will be excluded.

***SET**

***TERMINATION**

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

***TERMINATION_BODY**

***TERMINATION_CONTACT**

***TERMINATION_CURVE**

***TERMINATION_NODE**

*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.

*TERMINATION

*TERMINATION_CURVE

Purpose: Terminate the calculation when the load curve value returns to zero. This termination can be used with the contact option *CONTACT_AUTO_MOVE. In this latter option, the load curve is modified to account for the movement of the master surface.

Card Format

	1	2	3	4	5	6	7	8
Variable	LCID	ATIME						
Type	I	F						
Default	none	Remark 1		-				

VARIABLE

DESCRIPTION

LCID

Load curve ID governing termination.

ATIME

Activation time. After this time the load curve is checked. If zero, see remark 1 below.

Remarks:

1. If ATIME=0.0, termination will occur after the load curve value becomes nonzero and then returns to zero.

*TERMINATION

*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**

***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:

Version

ANSYS Keyword

LS-DYNA Keyword

All	N,Type,NODE,Val1,Val2,Val3	*NODE
All	EN,Type,I1,I2,I3,I4,I5,I6,I7,I8	*ELEMENT
All	MPDATA, R5.0, LENGTH, Lab, MAT, STLOC, VAL1, VAL2, VAL3	*MAT_ELASTIC

*TRANSLATE

<u>Version</u>	<u>ANSYS Keyword</u>	<u>LS-DYNA Keyword</u>
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
MASTER	2411	*NODE
5	780	*ELEMENT
MASTER	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>
MASTER	2430	*PART&*SECTION
5	755	*BOUNDARY_SPC_NODE
MASTER	791	
	time variation set.le.0.0	*BOUNDARY_SPC_NODE
	time variation set.gt.0.0	*BOUNDARY_PRESCRIBED_ MOTION_NODE
MASTER	790	
	load type.eq.1	*LOAD_NODE

*TRANSLATE_NASTRAN

Purpose: Provide a convenient route to read in NASTRAN input deck as part of the LS-DYNA keyword input. This keyword can appear more than once anywhere in the input. Also, see remarks below. **NOTE: The *TRANSLATE_NASTRAN command has been superseded by the *INCLUDE_NASTRAN command.** The following parameters are supported by the new command.

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.
- Current **NASTRAN** only supports shell element with constant thickness T.
For further explanation see *PART and *SECTION_SOLID.

PSOLID	PID	MID	SCID	EOSID	HGID				
Type	I	I	I	I	I				

- The **THRU** command for SPC, SPC1 is not supported in the current translation.
- For RBE2 keyword, if any of the rotational DOF (4,5,6) appears in the constraint, LS-DYNA will treat it as nodal rigid body constraint. Otherwise, LS-DYNA will use nodal constraints to treat this RBE2.

***USER**

***USER_INTERFACE_OPTION**

Available options include:

CONTROL

FRICTION

Purpose: Define user defined input and allocate storage for user defined subroutines for the contact algorithms. See also *CONTROL_CONTACT. The **CONTROL** option above allows the user to take information from the contact interface for further action, e.g., stopping the analysis. A sample user subroutine is provided in Appendix D.

The **FRICTION** option may be used to modify the Coulomb friction coefficients according to contact information or to use a friction coefficient database. A sample subroutine for treating the friction in contact is provided in Appendix E.

Card Format

	1	2	3	4	5	6	7	8
Variable	IFID	NOC	NOCI					
Type	I	I	I					
Default	none	none	none					

*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							

VARIABLE

DESCRIPTION

PARM_n

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=D3DUMP*n*

where *D3DUMPn* (or whatever name is chosen for the family member) is the *n* th restart file from the last run where the data is taken. LS-DYNA automatically detects that a small input deck is used since the I=*restartinput* file may contain the keywords:

***CHANGE_OPTION**

***CONTROL_DYNAMIC_RELAXATION**

***CONTROL_SHELL**

***CONTROL_TERMINATION**

*RESTART

***CONTROL_TIMESTEP**

***DAMPING_GLOBAL**

***DATABASE_OPTION**

***DATABASE_BINARY_OPTION**

***DELETE_OPTION**

***INTERFACE_SPRINGBACK**

***RIGID_DEFORMABLE_OPTION**

***STRESS_INITIALIZATION_{OPTION}**

***TERMINATION_OPTION**

***TITLE**

***KEYWORD** (see INTRODUCTION, Execution Syntax)

***CONTROL_CPU**

***DEFINE_OPTION**

***SET_OPTION**

i.e., the keyword ***STRESS_INITIALIZATION** may not be used in the small restart. The user has to take care that nonphysical modifications to the input deck are avoided; otherwise, complete nonsense may be the result.

- c) If many modifications are desired a so called full restart may be the appropriate choice. Then the keyword ***STRESS_INITIALIZATION** has to be provided in the input. As also outlined in the INTRODUCTION, Restart Analysis, either all parts can be initialized with the restart data or some selection of parts can be made for the stress initialization. See ***STRESS_INITIALIZATION**. In a full deck restart, deleted elements in this section will be deleted in the full deck automatically even though they are defined. Likewise, if it is necessary to change the velocity field, that must also be performed in this section using the **CHANGE_VELOCITY_....** options. The velocity field in the full deck part of the input is ignored.

*CHANGE_OPTION

Available options are:

BOUNDARY_CONDITION
CONTACT_SMALL_PENETRATION
CURVE_DEFINITION
RIGID_BODY_CONSTRAINT
RIGID_BODY_STOPPER
STATUS_REPORT_FREQUENCY
THERMAL_PARAMETERS
VELOCITY
VELOCITY_NODE
VELOCITY_RIGID_BODY
VELOCITY_ZERO

Purpose: Change some solution options.

*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.

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

DESCRIPTIONID n Contact ID for surface number n .

The **CURVE_DEFINITION** option allows a load curve to be redefined. *The new load curve must contain the same number of points as the curve it replaces.* The curve should be defined in the DEFINE_CURVE section of this manual. This input terminates when the next "*" card is encountered. Any offsets and scale factors are ignored.

Card Format

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

VARIABLE

DESCRIPTION

LCID

Load curve ID

*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 ²⁸						

VARIABLE

DESCRIPTION

PID	Part ID of master rigid body, see *PART.
LCMAX	Load curve ID defining the maximum coordinate as a function of time: EQ.0: no limitation of the maximum displacement. New curves can be defined by the *DEFINE_CURVE within the present restart deck.
LCMIN	Load curve ID defining the minimum coordinate as a function of time: EQ.0: no limitation of the minimum displacement. New curves can be defined by the *DEFINE_CURVE within the present restart deck.
PSIDMX	Optional part set ID of rigid bodies that are slaved in the maximum coordinate direction to the master rigid body. This option requires additional input by the *SET_PART definition.

*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 **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.
TOL	Non-linear convergence tolerance: EQ.0: No change.

*RESTART

The **VELOCITY_NODE** and the **VELOCITY_NODE_ONLY** options allows the velocity of nodal points to be changed at restart. Termination of this input is when the next "*" card is read. Undefined nodes will have their nodal velocities reset to zero if a *CHANGE_VELOCITY_NODE definition is encountered in the restart deck. However, if any of the *CHANGE_VELOCITY or CHANGE_VELOCITY_NODE definitions have _ONLY appended, then only the specified nodes will have their nodal velocities modified.

Card 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.
3. If both *CHANGE_VELOCITY and *CHANGE_VELOCITY_ZERO cards are defined then all velocities will be reset to zero.

*RESTART

The **VELOCITY** and **VELOCITY_ONLY** options allows a new velocity field to be imposed at restart. Termination of this input is when the next "*" card is read. Undefined nodes will have their nodal velocities reset to zero if a *CHANGE_VELOCITY definition is encountered in the restart deck. However, if any of the *CHANGE_VELOCITY definitions have _ONLY appended, then only the specified nodes will have their nodal velocities modified.

Card 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.

<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.	

<u>VARIABLE</u>	<u>DESCRIPTION</u>
PID	Part ID of rigid body.
VX	Translational velocity in x-direction.
VY	Translational velocity in y-direction.
VZ	Translational velocity in z-direction.
VXR	Rotational velocity about the x-axis.
VYR	Rotational velocity about the y-axis.
VZR	Rotational velocity about the z-axis.

Remarks:

1. Rotational velocities are defined about the center of mass of the rigid body.
2. Rigid bodies not defined in this section will not have their velocities modified.

The **VELOCITY_ZERO** option resets the velocities to zero at the start of the restart. Only the *CHANGE_VELOCITY_ZERO card is required for this option without any further input.

***CONTROL_DYNAMIC_RELAXATION**

Purpose: Define controls for dynamic relaxation.

Card Format

	1	2	3	4	5	6	7	8
Variable	NRCYCK	DRTOL	DRFCTR	DRTERM	TSSFDR	IRELAL	EDTTL	IDRFLG
Type	I	F	F	F	F	I	F	I
Default	250	0.001	0.995	infinity	TSSFAC	0	0.0	0
Remarks	1	1	1	1	1			1

VARIABLE**DESCRIPTION**

NRCYCK	Number of iterations between convergence checks, for dynamic relaxation option (default = 250).
DRTOL	Convergence tolerance for dynamic relaxation option (default = 0.001).
DRFCTR	Dynamic relaxation factor (default = .995).
DRTERM	Optional termination time for dynamic relaxation. Termination occurs at this time or when convergence is attained (default = infinity).
TSSFDR	Scale factor for computed time step during dynamic relaxation. If zero, the value is set to TSSFAC defined on *CONTROL_TERMINATION. After converging, the scale factor is reset to TSSFAC.
IRELAL	Automatic control for dynamic relaxation option based on algorithm of Papadrakakis [Papadrakakis 1981].
EDTTL	Convergence tolerance on automatic control of dynamic relaxation.
IDRFLG	Dynamic relaxation flag for stress initialization: EQ.0: not active, EQ.1: dynamic relaxation is activated.

*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_SHELL**

Purpose: Change failure parameters NFAIL1 and NFAIL2 if necessary. These parameters must be nonzero in the initial run.

Card Format

Card 1 1 2 3 4 5 6 7 8

Variable								
Type								

Card 2 1 2 3 4 5 6 7 8

Variable						NFAIL1	NFAIL4	
Type						I	I	

VARIABLE

DESCRIPTION

NFAIL1

Flag to check for highly distorted under-integrated shell elements, print a message, and delete the element or terminate. Generally, this flag is not needed for one point elements that do not use the warping stiffness. A distorted element is one where a negative jacobian exist within the domain of the shell, not just at integratiion points. The checks are made away from the integration points to enable the bad elements to be deleted before an instability leading to an error termination occurs. This test will increase CPU requirements for one point elements.

EQ.1: print message and delete element.

EQ.2: print message, write D3DUMP file, and terminate

GT.2: print message and delete element. When NFAIL1 elements are deleted then write D3DUMP file and terminate. These NFAIL1 failed elements also include all shell elements that failed for other reasons than distortion. Before the D3DUMP file is writen, NFAIL1 is doubled, so the run can immediately be continued if desired.

NFAIL4

Flag to check for highly distorted fully-integrated shell elements, print a message, and delete the element or terminate. Generally, this flag is recommended. A distorted element is one where a negative jacobian exist within the domain of the shell, not just at integratiion points. The checks are made away from the integration points to enable the bad elements to be deleted before an instability leading to an error termination occurs.

EQ.1: print message and delete element.

EQ.2: print message, write D3DUMP file, and terminate

***RESTART**

GT.2: print message and delete element. When NFAIL4 elements are deleted then write D3DUMP file and terminate. These NFAIL4 failed elements also include all shell elements that failed for other reasons than distortion. Before the D3DUMP file is written, NFAIL4 is doubled, so the run can immediately be continued if desired.

***CONTROL_TERMINATION**

Purpose: Stop the job.

Card 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.

SECFORC	Cross section forces.
RWFORC	Wall forces.
NODOUT	Nodal point data.
ELOUT	Element data.
GLSTAT	Global data.
DEFORC	Discrete elements.
MATSUM	Material energies.
NCFORC	Nodal interface forces.
RCFORC	Resultant interface forces.
DEFGEO	Deformed geometry file
SPCFORC	Set dt for spc reaction forces.
SWFORC	Nodal constraint reaction forces (spotwelds and rivets).
ABSTAT	Set dt for airbag statistics.
NODFOR	Set dt for nodal force groups.
BNDOUT	Boundary condition forces and energy
RBDOUT	Set dt for rigid body data.
GCEOUT	Set dt for geometric contact entities.
SLEOUT	Set dt for sliding interface energy.
JNTFORC	Set dt for joint force file.
SBTOUT	Set dt for seat belt output file.
AVSFLT	Set dt for AVS database.
MOVIE	Set dt for MOVIE.
MPGS	Set dt for MPGS.
TPRINT	Set dt for thermal file.

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

<u>VARIABLE</u>	<u>DESCRIPTION</u>
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.					

VARIABLE

DESCRIPTION

NID Node ID

*RESTART

<u>VARIABLE</u>	<u>DESCRIPTION</u>
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	Flag to delete or activate nodal rigid bodies. If nodal rigid bodies or generalized, weld definitions are active in the deformable bodies that are switched to rigid, then the definitions should be deleted to avoid instabilities: EQ.0: no change, EQ.1: delete, EQ.2: activate.
NCSF	Flag to delete or activate nodal constraint set. If nodal constraint/spotweld definitions are active in the deformable bodies that are switched to rigid, then the definitions should be deleted to avoid instabilities: EQ.0: no change, EQ.1: delete, EQ.2: activate.
RWF	Flag to delete or activate rigid walls: EQ.0: no change, EQ.1: delete, EQ.2: activate.
DTMAX	Maximum permitted time step size after restart.

*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 keyword 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

- Abbo, A.J., and S.W. Sloan, "A Smooth Hyperbolic Approximation to the Mohr-Coulomb Yield Criterion," *Computers and Structures*, Vol. 54, No. 1, (1995).
- 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).
- Auricchio, F., R.L. Taylor and J. Lubliner, "Shape-memory alloys: macromodelling and numerical simulations of the superelastic behavior", *Computer Methods in Applied Mechanics and Engineering*, vol. 146, pages 281-312, 1997
- Auricchio, F. and R.L. Taylor, "Shape-memory alloys: modelling and numerical simulations of the finite-strain superelastic behavior", *Computer Methods in Applied Mechanics and Engineering*, vol. 143, pages 175-194, 1997
- Bahler AS: The series elastic element of mammalian skeletal muscle. *Am J Physiol* 213:1560-1564, 1967.
- Baker, E.L., "An Explosives Products Thermodynamic Equation of State Appropriate for Material Acceleration and Overdriven Detonation: Theoretical Background and Formulation," Technical Report ARAED-TR-911013, U.S. Army Armament Research, Development and Engineering Center, Picatinny 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, Picatinny 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).

REFERENCES

- 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).
- Bathe, K.-J. and Dvorkin, E.N. A four node plate bending element based on Mindlin-Reissner plate theory and a mixed interpolation, *Int. J. Num. Meth. Eng.*, 21, 367-383 (1985).
- Batoz, J.L. and Ben Tahar, M. Evaluation of a new quadrilateral thin plate bending element, *Int. J. Num. Meth. Eng.*, 18, 1644-1677 (1982).
- 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^0 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).

REFERENCES

- 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.
- Batoz, J.-L. and M. Ben Tahar, Evaluation of a new quadrilateral thin plate bending element, *International Journal for Numerical Methods in Engineering*, 18, (1982), 1655-1677.
- Broadhouse, B.J., "The Winfrith Concrete Model in LS-DYNA3D," Report: SPD/D(95)363, Structural Performance Department, AEA Technology, Winfrith Technology Centre, U.K. (1995).
- Brode, H.L., "Height of Burst Effects at High Overpressure," RAND, RM-6301-DASA, DASA 2506, (1970).
- 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).
- Christensen, R.M. "A Nonlinear Theory of Viscoelasticity for Application to Elastomers," *Journal of Applied Mechanics*, Volume 47, American Society of Mechanical Engineers, pages 762-768, December 1980.
- Chu, C.C. and A. Needleman, "Void Nucleation effects in Biaxially Stretched Sheets", *Journal of Engineering Materials and Technology*, (1977).
- Chung, K. and K. Shah, "Finite Element Simulation of Sheet Metal Forming for Planar Anisotropic Metals," *Int. J. of Plasticity*, 8, 453-476, (1992).

REFERENCES

- 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).
- Deshpande, V.S. and N.A. Fleck, "Isotropic Models for Metallic Foams," Journal of the Mechanics and Physics of Solids, 48, 1253-1283, (2000).
- Dick, R.E., and W.H. Harris, "Full Automated Rezoning of Evolving Geometry Problems," Numerical Methods in Industrial Forming Processes, Chenot, Wood, and Zienkiewicz, Editors, Balkema, 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.
- Faßnacht, W., "Simulation der Rißbildung in Aluminiumgußbauteilen," Dissertation, Technische Universität Darmstadt, (1999).
- Feucht, M., "Ein gradientenabhängiges Gursonmodell zur Beschreibung duktiler Schädigung mit Entfestigung," Dissertation, Technische Universität Darmstadt, (1998).
- Flanagan, D.P. and T. Belytschko, "A Uniform Strain Hexahedron and Quadrilateral and Orthogonal Hourglass Control," Int. J. Numer. Meths. Eng., 17, 679-706 (1981).
- Fung, Y.C., Biomechanics, Springer, New York, 1993.
- 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).

REFERENCES

- Goudreau, G.L. and J.O. Hallquist, "Recent Developments in Large Scale Finite Element Lagrangian Hydrocode Technology," *J. Comp. Meths. Appl. Mechs. Eng.*, 30 (1982).
- Goldak, J., Chakravarti, A., and Bibby, M., "A New Finite Element Model for Welding Heat Sources," *Metallurgical Transactions B*, vol. 15B, pp. 299-305, June, 1984
- Govindjee, S., Kay, J.G., and Simo, J.C. [1994], "Anisotropic Modeling and Numerical Simulation of Brittle Damage in Concrete," Report No. UCB/SEMM-94/18, Department of Civil Engineering, University of California, Berkeley, CA 94720.
- Govindjee, S., Kay, J.G., and Simo, J.C. [1995], "Anisotropic Modeling and Numerical Simulation of Brittle Damage in Concrete," *Int. J. Numer. Meth. Engng*, **38**, 3611-3633.
- Graefe, H., W. Krummheuer, and V. Siejak, "Computer Simulation of Static Deployment Tests for Airbags, Air Permeability of Uncoated Fabrics and Steady State Measurements of the Rate of Volume Flow Through Airbags," SAE Technical Paper Series, 901750, Passenger Car Meeting and Exposition, Dearborn, Michigan, September 17-20, 1990.
- Gran, J.K. and P.E. Senseny, "Compression Bending of Scale-Model Reinforced-Concrete Walls," *ASCE Journal of Engineering Mechanics*, Volume 122, Number 7, pages 660-668, July 1996.
- Guccione, J., A. McCulloch, and L. Waldman, "Passive Material Properties of Intact Ventricular Myocardium Determined from a Cylindrical Model," published in the *ASME Journal of Biomechanical Engineering*, Vol. 113, pages 42-55, (1991).
- Gurson, A.L., "Plastic Flow and Fracture Behavior of Ductile Materials Incorporating Void Nucleation, Growth, and Interaction", Ph.D. Thesis, Brown University, (1975).
- Gurson, A.L., "Continuum Theory of Ductile Rupture by Void Nucleation and Growth: Part I - Yield Criteria and Flow Rules for Porous Ductile Media", *J. of Eng. Materials and Technology*, (1977).
- Hallquist, J.O., "Preliminary User's Manuals for DYNA3D and DYNAP (Nonlinear Dynamic Analysis of Solids in Three Dimension)," University of California, Lawrence Livermore National Laboratory, Rept. UCID-17268 (1976) and Rev. 1 (1979).[a]
- Hallquist, J.O., "A Procedure for the Solution of Finite Deformation Contact-Impact Problems by the Finite Element Method," University of California, Lawrence Livermore National Laboratory, Rept. UCRL-52066 (1976).
- Hallquist, J.O., "A Numerical Procedure for Three-Dimensional Impact Problems," *American Society of Civil Engineering*, Preprint 2956 (1977).
- Hallquist, J.O., "A Numerical Treatment of Sliding Interfaces and Impact," in: K.C. Park and D.K. Gartling (eds.) *Computational Techniques for Interface Problems*, AMD Vol. 30, ASME, New York (1978).
- Hallquist, J.O., "NIKE2D: An Implicit, Finite-Element Code for Analyzing the Static and Dynamic Response of Two-Dimensional Solids," University of California, Lawrence Livermore National Laboratory, Rept. UCRL-52678 (1979).[b]

REFERENCES

- Hallquist, J.O., "User's Manual for DYNA2D – An Explicit Two-Dimensional Hydrodynamic Finite Element Code with Interactive Rezoning," University of California, Lawrence Livermore National Laboratory, Rept. UCID-18756 (1980).
- Hallquist, J.O., "User's Manual for DYNA3D and DYNAP (Nonlinear Dynamic Analysis of Solids in Three Dimensions)," University of California, Lawrence Livermore National Laboratory, Rept. UCID-19156 (1981).[a]
- Hallquist, J. O., "NIKE3D: An Implicit, Finite-Deformation, Finite-Element Code for Analyzing the Static and Dynamic Response of Three-Dimensional Solids," University of California, Lawrence Livermore National Laboratory, Rept. UCID-18822 (1981).[b]
- Hallquist, J.O., "DYNA3D User's Manual (Nonlinear Dynamic Analysis of Solids in Three Dimensions)," University of California, Lawrence Livermore National Laboratory, Rept. UCID-19156 (1982; Rev. 1: 1984; Rev. 2: 1986).
- Hallquist, J.O., "Theoretical Manual for DYNA3D," University of California, Lawrence Livermore National Laboratory, Rept. UCID-19501 (March, 1983).
- Hallquist, J.O., "DYNA3D User's Manual (Nonlinear Dynamic Analysis of Solids in Three Dimensions)," University of California, Lawrence Livermore National Laboratory, Rept. UCID-19156 (1988, Rev. 4).
- Hallquist, J.O., "LS-DYNA User's Manual (Nonlinear Dynamic Analysis of Solids in Three Dimensions)," Livermore Software Technology Corporation, Rept. 1007 (1990).
- Hallquist, J.O., D.J. Benson, and G.L. Goudreau, "Implementation of a Modified Hughes-Liu Shell into a Fully Vectorized Explicit Finite Element Code," Proceedings of the International Symposium on Finite Element Methods for Nonlinear Problems, University of Trondheim, Trondheim, Norway (1985).
- Hallquist, J.O. and D.J. Benson, "A Comparison of an Implicit and Explicit Implementation of the Hughes-Liu Shell," Finite Element Methods for Plate and Shell Structures, T.J.R. Hughes and E. Hinton, Editors, 394-431, Pineridge Press Int., 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.

REFERENCES

- Hill, R., "Aspects of Invariance in Solid Mechanics," *Advances in Applied Mechanics*, Vol. 18, pp. 1-75 (1978).
- 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).
- Holmquist, T.J., G.R. Johnson, and W.H. Cook, "A Computational Constitutive Model for Concrete Subjected to Large Strains, High Strain Rates, and High Pressures", *Proceedings 14th International Symposium on Ballistics*, Quebec, Canada, pp. 591-600, (1993).
- Hopperstad, O.S. and Remseth, S., "A return Mapping Algorithm for a Class of Cyclic Plasticity Models", *International Journal for Numerical Methods in Engineering*, Vol. 38, pp. 549-564, (1995).
- 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).
- Ibrahimbegovic, A. and Wilson, E.L. A unified formulation for triangular and quadrilateral flat shell finite elements with six nodal degrees of freedom, *Comm. Applied Num. Meth*, 7, 1-9 (1991).
- Johnson, G.C. and D.J. Bammann, "A discussion of stress rates in finite deformation problems," *Int. J. Solids Struct*, 20, 725-737 (1984).
- Johnson, G.R. and W.H. Cook, "A Constitutive Model and Data for Metals Subjected to Large Strains, High Strain Rates and High Temperatures." Presented at the Seventh International Symposium on Ballistics, The Hague, The Netherlands, April 1983.
- Johnson, G.R. and T.J. Holmquist, "An Improved Computational Model for Brittle Materials" in *High-Pressure Science and Technology - 1993 American Institute of Physics Conference Proceedings* 309 (c 1994) pp.981-984 ISBN 1-56396-219-5.
- Holmquist, T.J., G.R. Johnson, and W.H. Cook, "A Computational Constitutive Model for Concrete Subjected to Large Strains, High Strain Rates, and High Pressures, pp. 591-600, (1993).
- Jones, R.M., *Mechanics of Composite Materials*, Hemisphere Publishing Corporation, New York, (1975).

REFERENCES

- 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).
- Lewis, B.A., "Developing and Implementing a Road Side Safety Soil Model into LS-DYNA," FHWA Research and Development Turner-Fairbank Highway Research Center, (1999).
- 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., Lubliner, J., and Taylor, R.L., "A Constitutive Model for Anisotropic Damage in Fiber-Composites," *Mechanics of Materials*, Vol. 20, pp. 125-152 (1995).
- Matzenmiller, A. and J. K. 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 Verlag, (1991).
- Murray, Y.D., "Users Manual for Transversely Isotropic Wood Model" APTEK, Inc., Technical Report to the FHWA (to be published) 2002.
- Murray, Y.D. and Lewis, B.A., "Numerical Simulation of Damage in Concrete" APTEK, Inc., Technical Report DNA-TR-94-190, Contract DNA 001-91-C-0075, Defense Nuclear Agency, Alexandria VA 22310.
- Nagararajah, Reinhorn, & Constantinou, "Nonlinear Dynamic Analysis of 3-D Base-Isolated Structures", *Journal of Structural Engineering* Vol 117 No 7 July (1991)
- Neilsen, M.K., H.S. Morgan, and R.D. Krieg, "A Phenomenological Constitutive Model for Low Density Polyurethane Foams," Rept. SAND86-2927, Sandia National Laboratories, Albuquerque, N.M., (1987)
- Nusholtz, G., W. Fong, and J. Wu, "Air Bag Wind Blast Phenomena Evaluation," Experimental Techniques, Nov.-Dec. (1991).
- Nusholtz, G., D. Wang, and E.B. Wylie, "Air Bag Momentum Force Including Aspiration," Preprint, Chrysler Corporation, (1996).

REFERENCES

- Nusholz, private communication, (1996).
- Park, Y.J., Wen, Y.K, and Ang, A.H-S, "Random Vibration of Hysteretic Systems Under Bi-directional Ground Motions", Earthquake Engineering and Structural Dynamics, Vol. 14, pp. 543-557 (1986).
- Puso, M.A. and Weiss, J.A., "Finite Element Implementation of Anisotropic Quasilinear Viscoelasticity Using a Discrete Spectrum Approximation", ASME J. Biomech. Engng., 120, 62-70 (1998).
- 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).
- Quapp, K.M. and Weiss, J.A., "Material Characterization of Human Medial Collateral Ligament", ASME J. Biomech Engng., 120, 757-763 (1998)
- Reyes, A., O.S. Hopperstad, T. Berstad, and M. Langseth, "Implementation of a Material Model for Aluminium Foam in LS-DYNA", Report R-01-02, Restricted, Department of Structural Engineering, Norwegian University of Science and Technology, (2002).
- Randers-Pehrson, G. and K. A. Bannister, "Airblast Loading Model for DYNA2D and DYNA3D," Army Research Laboratory, Rept. ARL-TR-1310, publicly released with unlimited distribution, (1997).
- Rupp, A., Grubisic, V., and Buxbaum, O., Ermittlung ertragbarer Beanspruchungen am Schweisspunkt auf Basis der ubertragbaren Schnittgrossen, FAT Schriftenreihe 111, Frankfurt (1994).
- 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.
- Schwer, L.E. and Y.D. Murray, "A Three-Invariant Smooth Cap Model with Mixed Hardening," International Journal for Numerical and Analytical Methods in Geomechanics, Volume 18, pp. 657-688, 1994.
- Schwer, L.E., "A Viscoplastic Augmentation of the Smooth Cap Model," Nuclear Engineering and Design, Vol. 150, pp. 215-223, 1994.
- Schwer, L.E., "Demonstration of the Continuous Surface Cap Model with Damage: Concrete Unconfined Compression Test Calibration," LS-DYNA Geomaterial Modeling Short Course Notes, July , 2001.
- Sheppard, S.D., Estimations of Fatigue Propagation Life in Resistance Spot Welds, ASTM STP 1211, pp. 169-185, (1993).

REFERENCES

- 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).
- Stouffer and Dame, *Inelastic Deformation of Metals*, Wiley, (1996).
- Stout, M.G., D.E. Helling, T.L. Martin, and G.R. Canova, Int. J. Plasticity, Vol. 1, pp. 163-174, 1985.
- Taylor, L.M. and D.P. Flanagan, "PRONTO3D A Three-Dimensional Transient Solid Dynamics Program," Sandia Report: SAND87-1912, UC-32, (1989).
- Taylor, R.L. Finite element analysis of linear shell problems, in *Whiteman, J.R. (ed.), Proceedings of the Mathematics in Finite Elements and Applications*, Academic Press, New York, 191-203, (1987).
- Taylor, R.L. and Simo, J.C. Bending and membrane elements for the analysis of thick and thin shells, *Proc. of NUMETA Conference*, Swansea (1985).
- Tsai, S.W. and E.M. Wu, "A General Theory of Strength for Anisotropic Materials," J. Composite Materials, 5, 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).

REFERENCES

- Wang, J.T., "An Analytical Model for an Airbag with a Hybrid Inflator", AMD-Vol. 210, BED-Vol. 30, ASME, pp 467-497, (1995).
- Weiss, J.A., Maker, B.N. and Govindjee, S., "Finite Element Implementation of Incompressible, Transversely Isotropic Hyperelasticity", Comp. Meth. Appl. Mech. Eng., 135, 107-128 (1996).
- Wen, T.K. "Method for Random Vibration of Hysteretic Systems", J. Engrg. Mech., ASCE, Vol. 102, No. EM2, Proc. Paper 12073, pp.249-263 (1976).
- Wilson, E.L. *Three Dimensional Static and Dynamic Analysis of Structures*, Computers and Structures, Inc., Berkeley CA, (2000).
- Whirley, R. G., and J. O. Hallquist, "DYNA3D, A Nonlinear, Explicit, Three-Dimensional Finite Element Code for Solid and Structural Mechanics-Users Manual," Report No.UCRL-MA-107254 , Lawrence Livermore National Laboratory, (1991).
- Whirley, R. G., and G.A. Henshall, "Creep Deformation Structural Analysis Using An Efficient Numerical Algorithm," IJNME, Vol. 35, pp. 1427-1442, (1992).
- Wilkins, M.L., 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, Report UCRL-51079, (1973).
- Worswick, M.J., and Xavier Lalbin, Private communication, Livermore, California, (1999).
- Yen, C.F. and Caiazzo, A, "Innovative Processing of Multifunctional Composite Armor for Ground Vehicles," ARL-CR-484, U.S. Army Research Laboratory, Aberdeen Proving Ground, MD, (2001).
- Zajac F.E., "Muscle and tendon: Properties, models, scaling, and application to biomechanics and motor control, "CRC Critical Reviews in Biomedical Engineering 17(4):359-411, (1989).
- Zayas, V.A., Low, S.S. and Mahin, S.A., "A Simple Pendulum Technique for Achieving Seismic Isolation", J. Earthquake Spectra, Vol. 6, No. 2, pp. 317-334 (1990).
- Zhang, S., Approximate Stress Intensity Factors and Notch Stresses for Common Spot-Welded Specimens, Welding Research Supplement, pp. 173s-179s, (1999).

REFERENCES

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.

Thermal effects can be included if nodal temperatures are available through either thermal coupling or one of the keyword options such as *LOAD_THERMAL_LOAD_CURVE. The last argument in the calling sequence to the user subroutine is the current temperature which is assumed to be uniform over the element.

A sample subroutine is provided below for treating an elastic material.

```

SUBROUTINE UMAT41 (CM, EPS, SIG, HISV, DT1, CAPA, ETYPE, TIME, TEMP)
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 TEMP=CURRENT TEMPERATURE
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 RBU(3-6) TOTAL ROTATIONS ABOUT THE LOCAL XYZ AXES
C RBV(1-3) VELOCITIES IN THE LOCAL XYZ DIRECTIONS
C RBV(3-6) ROTATIONAL VELOCITIES ABOUT THE LOCAL XYZ AXES
C RBA(1-3) ACCELERATIONS IN THE LOCAL XYZ DIRECTIONS
C RBA(3-6) ROTATIONAL ACCELERATIONS ABOUT THE LOCAL XYZ AXES
C TIME IS THE CURRENT TIME
C DT1 IS TIME STEP SIZE AT N-1/2
C DT2 IS TIME STEP SIZE AT N+1/2
C PARAM IS USER DEFINED INPUT PARAMETERS (MAX 25)
C HIST IS USER DEFINED HISTORY VARIABLES (MAX 25)
C ITRNON IS FLAG TO TURN ON THE AIRBAG INFLATION
C RBUG,RBVG,RBAG, ARE SIMILAR TO RBU,RBV,RBA BUT ARE DEFINED
C GLOBALLY.
C
C THE USER SUBROUTINE SETS THE VARIABLE ITRNON TO:
C
C ITRNON=0 BAG IS NOT INFLATED
C ITRNON=1 BAG INFLATION BEGINS AND THIS SUBROUTINE IN NOT
C CALLED AGAIN
C
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 called each time step and does not need any control card to operate.

```

SUBROUTINE UCTRL1 (NUMNP, NDOF, TIME, DT1, DT2, PRTC, PLTC, FRCI, PRTO,
. PLTO, FRCO, VT, VR, AT, AR, UT, UR, XMST, XMSR, IRBODY, RBDYN, USRHV,
. MESSAG, TOTALM, CYCL, IDRINT)
C*****
C | LIVERMORE SOFTWARE TECHNOLOGY CORPORATION (LSTC) |
C | ----- |
C | COPYRIGHT 1987, 1988, 1989 JOHN O. HALLQUIST, LSTC |
C | ALL RIGHTS RESERVED |
C*****
C
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 OF FREEDOM PER NODE
C TIME=CURRENT SOLUTION TIME
C PRTC=OUTPUT INTERVAL FOR LS-DYNA TIME HISTORY DATA
C PLTC=OUTPUT INTERVAL FOR LS-DYNA STATE DATA
C FRCI=OUTPUT INTERVAL FOR LS-DYNA INTERFACE FORCE DATA
C PRTO=OUTPUT TIME FOR TIME HISTORY FILE
C PLTO=OUTPUT TIME FOR STATE DATA
C FRCO=OUTPUT TIME FOR FORCE DATA
C VT(3,NUMNP) =NODAL TRANSLATIONAL VELOCITY VECTOR
C VR(3,NUMNP) =NODAL ROTATIONAL VELOCITY VECTOR. THIS ARRAY
C IS DEFINED IF AND ONLY IF NDOF=6
C AT(3,NUMNP) =NODAL TRANSLATIONAL ACCELERATION VECTOR
C AR(3,NUMNP) =NODAL ROTATIONAL ACCELERATION VECTOR. THIS
C ARRAY IS DEFINED IF AND ONLY IF NDOF=6
C UT(3,NUMNP) =NODAL TRANSLATIONAL DISPLACEMENT VECTOR
C UR(3,NUMNP) =NODAL ROTATIONAL DISPLACEMENT VECTOR. THIS ARRAY
C IS DEFINED IF AND ONLY IF NDOF=6
C XMST(NUMNP) =RECIPROCAL OF NODAL TRANSLATIONAL MASSES
C XMSR(NUMNP) =RECIPROCAL OF NODAL ROTATIONAL MASSES. THIS
C ARRAY IS DEFINED IF AND ONLY IF NDOF=6
C IRBODY =FLAG FOR RIGID BODY NODAL POINTS
C IF DEFORMABLE NODE THEN SET TO 1.0
C IF RIGID BODY NODE THEN SET TO 0.0
C DEFINED IF AN ONLY IF RIGID BODY ARE PRESENT
C I.E., IRBODY.NE.0 IF NO RIGID BODY ARE PRESENT
C USRHV (LENHV) =USER DEFINED HISTORY VARIABLES THAT ARE STORED

```

```

C          IN THE RESTART FILE.  LENHV=100+U*NUMMAT WHERE
C          NUMMAT IS THE # OF MATERIALS IN THE PROBLEM.
C          ARRAY USRHV IS UPDATED ONLY IN THIS SUBROUTINE.
C          MESSAG      =FLAG FOR DYNA3D WHICH MAY BE SET TO:
C                      'SW1.' LS-DYNA TERMINATES WITH RESTART FILE
C                      'SW3.' LS-DYNA WRITES A RESTART FILE
C                      'SW4.' LS-DYNA WRITES A PLOT STATE
C          TOTALM      =TOTAL MASS IN PROBLEM
C          CYCLE      =CYCLE NUMBER
C          IDRINT      =FLAG FOR DYNAMIC RELAXATION PHASE
C                      .NE.0:  DYNAMIC RELAXATION IN PROGRESS
C                      .EQ.0:  SOLUTION PHASE
C
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
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
C DIMENSION VT(3,*),VR(3,*),AT(3,*),AR(3,*),UT(3,*),UR(3,*),
XNST(*),XMSR(*),RBDYN(*),USRHV(*)
C
C SAMPLE MOMENTUM AND KINETIC ENERGY CALCULATIONS
C
C REMOVE ALL COMMENTS IN COLUMN 1 BELOW TO ACTIVATE
CC
CC
CC INITIALIZE KINETIC ENERGY, XKE, AND X,Y,Z MOMENTUMS.
CC
C XKE=2.*SWKENG+2.*ENRBDY
C XM=SWXMOM+RBDYX
C YM=SWYMOM+RBDYY
C ZM=SWZMOM+RBDYZ
CC
C NUMNP2=NUMNP
C IF (NDOF.EQ.6) THEN
C NUMNP2=NUMNP+NUMNP
C ENDIF
C PRINT *,NDOF
C IF (IRBODY.EQ.0) THEN
CC
CC
CC NO RIGID BODIES PRESENT
CC
CC NOTE IN BLANK COMMENT VR FOLLOWS VT. THIS FACT IS USED BELOW.
C DO 10 N=1,NUMNP2
C XMSN=1./XNST(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./XNST(N)
C VN1=RBDYN(N)*VT(1,N)
C VN2=RBDYN(N)*VT(2,N)
C VN3=RBDYN(N)*VT(3,N)
C XM=XM+XMSN*VN1
C YM=YM+XMSN*VN2
C ZM=ZM+XMSN*VN3
C XKE=XKE+XMSN*(VN1*VN1+VN2*VN2+VN3*VN3)
C 20 CONTINUE
C IF (NDOF.EQ.6) THEN
C DO 30 N=1,NUMNP

```

```
C      XMSN=1./XMSR(N)
C      VN1=RBDYN(N)*VR(1,N)
C      VN2=RBDYN(N)*VR(2,N)
C      VN3=RBDYN(N)*VR(3,N)
C      XM=XM+XMSN*VN1
C      YM=YM+XMSN*VN2
C      ZM=ZM+XMSN*VN3
C      XKE=XKE+XMSN*(VN1*VN1+VN2*VN2+VN3*VN3)
C 30  CONTINUE
C      ENDIF
CC
C      ENDIF
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
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(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)
20 CONTINUE
C      IF (XMAX+THKS.LT.XMIN-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
C   INTEGER CYCLE
C   CHARACTER*(*) SIDE
C   DIMENSION UA(*), MASTRS(4), XCM(4), YCM(4), ZCM(4)
C
C
C   USER SUBROUTINE FOR INTERFACE FRICTION CONTROL
C
C   NOTE: LS-DYNA USES AN INTERNAL NUMBERING SYSTEM TO
C   ACCOMODATE ARBITRARY NODE NUMBERING. TO ACCESS
C   INFORMATION FOR USER NODE N, ADDRESS ARRAY LOCATION M,
C   M=LQF(N,1). TO OBTAIN USER NODE NUMBER, N,
C   CORRESPONDING TO ARRAY ADDRESS M, SET N=LQFINV(M,1)
C
C   ARGUMENTS:
C   NSI           =NUMBER OF SLIDING INTERFACE
C   TIME          =CURRENT SOLUTION TIME
C   CYCLE         =CYCLE NUMBER
C   DT2          =TIME STEPS SIZE AT N+1/2
C   NSLAVE       =SLAVE NODE NUMBER IN LS-DYNA INTERNAL
C               NUMBERING
C   AREAS        =SLAVE NODE AREA (INTERFACE TYPES 5&10 ONLY)
C   XS           =X-COORDINATE SLAVE NODE (PROJECTED)
C   YS           =Y-COORDINATE SLAVE NODE (PROJECTED)
C   ZS           =Z-COORDINATE SLAVE NODE (PROJECTED)
C   MSN          =MASTER SEGMENT NUMBER
C   MASTRS(4)    =MASTER SEGMENT NODE IN LS-DYNA INTERNAL
C               NUMBERING
C   AREAM        =MASTER SEGMENT NUMBER
C   XCM(4)       =X-COORDINATES MASTER SURFACE (PROJECTED)
C   YCM(4)       =Y-COORDINATES MASTER SURFACE (PROJECTED)
C   ZCM(4)       =Z-COORDINATES MASTER SURFACE (PROJECTED)
C   STFSN        =SLAVE NODE PENALTY STIFFNESS
C   STFMS        =MASTER SEGMENT PENALTY STIFFNESS
C   FORCEN       =NORMAL FORCE
C   RVX, RVY, RVZ, =RELATIVE X, Y, Z-VELOCITY BETWEEN SLAVE NODE AND
C               MASTER SEGMENT

```

```
C*****
C   THE FOLLOWING VALUES ARE TO BE SET BY USER
C
C   FRIC1           =STATIC FRICTION COEFFICIENT
C   FRIC2           =DYNAMIC FRICTION COEFFICIENT
C   FRIC3           =DECAY CONSTANT
C   FRIC4           =VISCOUS FRICTION COEFFICIENT (SETTING FRIC4=0
C                   TURNS THIS OPTION OFF)
C
C*****
C   NINPUT          =NUMBER OF VARIABLES INPUT INTO UA
C   UA(*)           =USERS' ARRAY, FIRST NINPUT LOCATIONS
C                   DEFINED BY USER.  THE LENGTH OF THIS
C                   ARRAY IS DEFINED ON CONTROL CARD 15.
C                   THIS ARRAY IS UNIQUE TO INTERFACE NSI.
C
C   SIDE            ='MASTER' FOR FIRST PASS.  THE MASTER
C                   SURFACE IS THE SURFACE DESIGNATED IN THE
C                   INPUT.
C                   ='SLAVE' FOR SECOND PASS AFTER SLAVE AND
C                   MASTER SURFACES HAVE BE SWITCHED FOR
C                   THE TYPE 3 SYMMETRIC INTERFACE TREATMENT
C
C*****
C
C   RETURN
C   END
```

APPENDIX F: Occupant Simulation Including the Coupling to Programs CAL3D and MADYMO**INTRODUCTION**

LS-DYNA is coupled to occupant simulation codes to generate solutions in automotive crashworthiness that include occupants interacting with the automotive structure. In such applications LS-DYNA provides the simulation of the structural and deformable aspects of the model and the OSP (Occupant Simulation Program) simulates the motion of the occupant. There is some overlap between the two programs which provides flexibility in the modeling approach. For example, both the OSP and LS-DYNA have the capability of modeling seat belts and other deformable restraints. The advantage of using the OSP is related to the considerable databases and expertise that have been developed in the past for simulating dummy behavior using these programs.

The development of the interface provided LSTC a number of possible approaches. The approach selected is consistent with the LSTC philosophy of providing the most flexible and useful interface possible. This is important because the field of non-linear mechanics is evolving rapidly and techniques which are used today are frequently rendered obsolete by improved methodologies and lower cost computing which allows more rigorous techniques to be used. This does make the learning somewhat more difficult as there is not any single procedure for performing a coupling.

One characteristic of LS-DYNA is the large number of capabilities, particularly those associated with rigid bodies. This creates both an opportunity and a difficulty: LSDYNA3D has many ways approximating different aspects of problems, but they are frequently not obvious to users without considerable experience. Therefore, in this Appendix we emphasize modeling methods rather than simply listing capabilities.

THE LS-DYNA/OCCUPANT SIMULATION PROGRAM LINK

Coupling between the OSP and LS-DYNA is performed by combining the programs into a single executable. In the case of CAL3D, LS-DYNA calls CAL3D as a subroutine, but in the case of MADYMO, LS-DYNA is called as a subroutine. The two programs are then integrated in parallel with the results being passed between the two until a user defined termination time is reached.

The OSP and LS-DYNA have different approaches to the time integration schemes. The OSP time integrators are based on accurate implicit integrators which are valid for large time steps which are on the order of a millisecond for the particular applications of interest here. An iterative solution is used to insure that the problem remains in equilibrium. The implicit integrators are extremely good for smoothly varying loads, however, sharp nonlinear pulses can introduce considerable error. An automatic time step size control which decreases the time step size quickly restores the accuracy for such events. The LS-DYNA time integrator is based on an explicit central difference scheme. Stability requires that the time step size be less than the highest frequency in the system. For a coarse airbag mesh, this number is on the order of 100 microseconds while an actual car crash simulation is on the order of 1 microsecond. The smallest LS-DYNA models have at least 1,000 elements. Experience indicates that the cost of a single LS-DYNA time step for a small model is at least as great as the cost of a time step in the OSP. Therefore, in the coupling, the LS-DYNA time step is used to control the entire simulation including the OSP part. This approach has negligible cost penalties and avoids questions of stability and accuracy that would result by using a subcycling scheme between the two programs. Optionally, a subcycling scheme can be used, however, the results of the analysis have to be checked with care.

LS-DYNA has a highly developed rigid body capability which is used in different parts of automobile crash simulation. In particular, components such as the engine are routinely modeled with rigid bodies. These rigid bodies have been modified so that they form the basis of the coupling procedure in LS-DYNA to the OSP.

In LS-DYNA, the geometry of a model is broken down into nodal points which identify positions in space. These nodes are then connected by elements so that the volume of a structure is identified. Each element has a “material” associated with it. If the element is deformable, then the material will specify its characteristics such as density and Young’s Modulus. A crash model can consist of 100 or more separate materials which are each assigned a “material number,” and each material number has an associated “material type” which determines if it is elastic, plastic, viscoelastic, orthotropic, etc.

The material type may also specify that it is a rigid body. In this case, all elements of the same material number are treated as a single rigid body. These elements are integrated to determine the mass, centroid and moments of inertia for the group. This group is then treated as a rigid body with six degrees-of-freedom including three translations and three rotations. The positions of the rigid bodies are updated in LS-DYNA by a time integrator which works together with the central difference time integration.

There is an additional flag which specifies that the LS-DYNA rigid body is coupled to an OSP rigid body. This flag can be found in the description of the rigid body material *MAT_RIGID (formerly material type 20). In coupled updates, the OSP rigid body time integrator takes over control of the LS-DYNA rigid body and the normal LS-DYNA updates are bypassed. The time integration procedure is then as follows:

1. At the beginning of a step, LS-DYNA determines the locations and updates the positions of all of the rigid bodies which are coupled to the OSP. This information is obtained from common block information in the OSP.
2. Using the information on rigid body locations, LS-DYNA proceeds to update the stresses and history variables of all of the deformable structures and computes the resultant forces acting on all rigid bodies.
3. The resultant forces are stored into an OSP common block along with the current time step. Control is then returned to the OSP so that the step can be completed by the OSP determining the new positions of the rigid bodies based on the applied forces.

At the end of the calculation LS-DYNA terminates normally, closing its files, and then control is returned to OSP which will also terminate normally. The termination time for the coupled run is taken as the minimum of the termination time provided to LS-DYNA and the termination time provided to the OSP.

The executable for the coupling with MADYMO currently needs to be specially created at each site. TNO provides all of the appropriate load modules with their libraries, and the appropriate load modules for LS-DYNA may be obtained by the corporate contact point at the LS-DYNA distributor. A complete executable must then be made by linking the two libraries. A revised password file must be obtained from TNO prior to running the coupled code. Coupling with CAL3D requires special on-site modification of the client’s CAL3D version to eliminate conflicting I/O unit numbers and to ensure that the common block lengths between the codes are consistent. LSTC does not distribute or support CAL3D.

To make the coupled program run, an input deck must be provided to both the OSP and LS-DYNA. The two input decks must be provided in the same set of consistent units. This can potentially require a major conversion to either the OSP input or the LS-DYNA input. With two legitimate and consistent input decks, the coupled program should run to completion with no problems. Additional inputs are required to make the models interact between the OSP and LS-DYNA portions of the run.

The simplest form of a coupled simulation is simply to include a single body in an OSP run. No special modifications are needed to the OSP input deck for use in the coupled simulation. Ellipsoids and planes in the OSP are usually attached to “segments” which correspond to LS-DYNA “rigid bodies.” Because the coupling procedure works on the basis of shared information on LS-DYNA rigid bodies with the OSP segments, the ellipsoids/planes listed in the OSP section must correspond to the segments which are to be coupled. These ellipsoids and planes may be actual geometry which is used for contact, or they may be simply artificial shapes to permit the data transfer between the OSP and LS-DYNA.

DUMMY MODELING

The dummy is typically modeled entirely within the OSP. The coupling of the dummy into LS-DYNA requires the creation of a separate LS-DYNA rigid body material for each segment of the OSP. The easiest way to create a mesh for the model is to set the LS-DYNA rigid body coupling option to 2.0. This causes LS-DYNA to search all of the ellipsoids connected to the appropriate segment and generate meshes which are then slaved to the OSP dummy. Thus, with minimal input, a complete dummy may be generated and the kinematics may be traced in LS-DYNA and displayed in the LS-DYNA post-processor, LS-PREPOST

Once the basic dummy coupling has been accomplished, the deformable finite element structure can be added. Assuming that an ellipsoid is available for the steering wheel, a flat airbag can be added in the proper location. One or more nodes must be attached to the steering wheel. This is done by identifying the attached nodes as “Extra Nodes for Rigid Body” which is input in LS-DYNA by **CONSTRAINED_EXTRA_NODES*_Option. The nodes are slaved to the LS-DYNA material which has been coupled to the MADYMO steering wheel model. Contact must now be identified between the airbag and the steering wheel, the windshield, and the various body parts which may be affected. This requires the use of one geometric contact entity (see **CONTACT_ENTITY*) for each plane or ellipsoid which may interact with the airbag. A control volume specifying inflation properties for the airbag must be specified (see **AIRBAG_OPTION*) to complete the model.

AIRBAG MODELING

Modeling of airbags is accomplished by use of shell or membrane elements in conjunction with a control volume (see **AIRBAG_OPTION*) and possibly a single surface contact algorithm to eliminate interpenetrations during the inflation phase (see **CONTACT_OPTION*). The contact types showing an “a” in front are most suited for airbag analysis. Current recommended material types for the airbags are:

**MAT_ELASTIC* = Type 1. Elastic

**MAT_COMPOSITE_DAMAGE* = Type 22. Layered orthotropic elastic for composites

**MAT_FABRIC* = Type 34. Fabric model for folded airbags

Model 34 is a “fabric” model which can be used for flat bags. As a user option this model may or may not support compression.

The elements which can be used are as follows:

Belytschko-Tsay quadrilateral with 1 point quadrature. This element behaves rather well for folded and unfolded cases with only a small tendency to hourglass. The element tends to be a little stiff. Stiffness form hourglass control is recommended.

Belytschko-Tsay membrane. This model is softer than the normal Belytschko-Tsay element and can hourglass quite badly. Stiffness form hourglass is recommended. As a better option, the fully integrated Belytschko-Tsay membrane element can be chosen.

C0 Triangular element. The C0 triangle is very good for flat bag inflation and has no tendency to hourglass.

The best choice is a specially developed airbag membrane element with quadrilateral shape. This is an automatic choice when the fabric material is used.

As an airbag inflates, a considerable amount of energy is transferred to the surrounding air. This energy transfer decreases the kinetic energy of the bag as it inflates. In the control volume logic, this is simulated either by using either a mass weighted damping option or a back pressure on the bag based on a stagnation pressure. In both cases, the energy that is absorbed is a function of the fabric velocity relative to a rigid body velocity for the bag. For the mass weighted case, the damping force on a node is proportional to the mass times the damping factor times the velocity vector. This is quite effective in maintaining a stable system, but has little physical justification. The latter approach using the stagnation pressure method estimates the pressure needed to accelerate the surrounding air to the speed of the fabric. The formula for this is:

$$P = Area \times \alpha \times \left((\vec{V}_i - \vec{V}_{cg}) \cdot \hat{n} \right)^2$$

This formula accomplishes a similar function and has a physical justification. Values of the damping factor, α , are limited to the range of 0 to 1, but a value of 0.1 or less is more likely to be a good value.

KNEE BOLSTER

The knee-to-knee bolster interactions are characterized by the stiffness of the knee being comparable to that of the knee bolster. Therefore, modeling the knee as a rigid body may produce large errors in the interaction forces. Calibrated force-deflection curves could be determined, but they would have no predictive value for slight changes to knee bolster designs. For this reason, a more accurate modeling of the compliance of the knee bolster and the knee is required.

The knee can be modeled as a combined rigid/deformable body. The rigid body is coupled to the OSP. Overlaying the rigid body are brick elements which model the “skin” that exists over the knees of the dummy. These brick elements use material type 6 (*MAT_VISCOELASTIC) which is a viscoelastic model that does a reasonable job of approximating the hysteretic behavior of rubbers. The inner layer of the brick elements is attached to the rigid body through the *CONSTRAINED_EXTRA_NODES Option. Between the knee bolster is a SURFACE-TO-SURFACE contact definition.

COMMON ERRORS

1. **Improper airbag inflation or no inflation.**

The most common problem is inconsistency in the units used for the input constants. An inflation load curve must also be specified. The normals for the airbag segments must all be consistent and facing outwards. If a negative volume results, this can sometimes be quickly cured by using the “flip” flag on the control volume definition to force inward facing normals to face outwards.

2. **Excessive airbag distortions.**

Check the material constants. Triangular elements should have less distortion problems than quadrilaterals. Overlapped elements at time zero can cause locking to occur in the contact leading to excessive distortions. The considerable energy input to the bag will create numerical noise and some damping is recommended to avoid problems.

3. **The dummy passes through the airbag.**

A most likely problem is that the contacts are improperly defined. Another possibility is that the models were developed in an incompatible unit system. The extra check for penetration flag if set to 1 on the contact control cards variable PENCHK in the *CONTACT_... definitions may sometimes cause nodes to be prematurely released due to the softness of the penalties. In this case the flag should be turned off.

4. **The OSP fails to converge.**

This may occur when excessively large forces are passed to the OSP. First, check that unit systems are consistent and then look for improperly defined contacts in the LS-DYNA input.

5. **Time step approaches zero.**

This is almost always in the airbag. If elastic or orthotropic (*MAT_ELASTIC or *MAT_COMPOSITE material 1 or 22) is being used, then switch to fabric material *MAT_FABRIC which is less time step size sensitive and use the fully integrated membrane element. Increasing the damping in the control volume usually helps considerably. Also, check for “cuts” in the airbag where nodes are not merged. These can allow elements to deform freely and cut the time step to zero.

APPENDIX 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.
CX	Rotate slice plane at zmin about x axis.
CY	Rotate slice plane at zmin about y axis.
CZ	Rotate slice plane at zmin about z axis.
DIF	Change diffused light level for material; DIF <mat #, -1 for all> <value>.
DISTANCE	Set distance of model from viewer; DIST <value in normalized model dimensions>.

DMATERIALS	Delete display of material in subsequent views; DMAT <ALL or list of numbers>.
DRAW	Display outside edges of model.
DSCALE	Scale current displacement from initial shape.
DYN	After using TAURUS command will reset display to read current DYNA3D state data.
ELPLT	Set or unset element numbering in subsequent views.
END	Delete display and return to execution.
ESCAPE	Escapes from menu pad mode.
EXECUTE	Return to execution and keep display active.
FCL	Fix or unfix current contour levels.
FOV	Set display field of view angle; FOV <value in degrees>.
FRINGE	View with colored contour fringes; fringe <component #> <list mat #>; see TAURUS manual.
GETFRAME	Display a saved frame; GETF <frame #>.
HARDWARE	Hardware mode; workstation hardware calls are used to draw, move and color model; repeat command to reset to normal mode.
HELP	
HZB	Switch on or off hardware zbuffer for a subsequent view, draw or contour command; rotations and translations will be in hardware.
LIMIT	Set range of node numbers subsequent views; limit <first node #> <last node #>.
MAT	Re-enable display of deleted materials mat <all or list of numbers>.
MENU	Button menu pad mode.
MOTION	Motion of model through mouse movement or use of a dial box. The left button down enables translation in the plane, middle button rotation about axes in the plane; and with right button down in the out of plane axis; left and middle button down quit this mode.
MOV	Drag picked part to new position set with mouse.

NDPLT	Set or unset node numbering in subsequent views.
NOFRAME	Set and unset drawing of a frame around the picture.
PAUSE	Animation display pause in seconds
PHS2 or THISTORY	Time history plotting phase. Similar to LS-TAURUS.
PICK	Get element information with mouse.
POST	Enable or disable postscript mode on the PC and eps file is written as picture is drawn; remove eofs and initgraphics for eps use.
QUIT	Same as execute.
RANGE	Set fix range for contour levels; range <minvalue> <maxvalue>.
RAX	Reflect model about xy plane; restore command will switch-off reflections.
RAY	Reflect model about yz plane; restore command will switch-off reflections.
RAZ	Reflect model about zx plane, restore command will switch-off reflections.
RESTORE	Restores model to original position, also switches off element and node numbers, slice capper, reflections and cut model.
RETURN	Exit.
RGB	Change color red green blue element <mat #> <red> <green> <blue>.
RX	Rotate model about x axis.
RY	Rotate model about y axis.
RZ	Rotate model about z axis.
SAVE	Set or unset saving of display for animation.
SEQUENCE	Periodic plot during execution; SEQ <# of cycles> <commands> EXE.
SHR	Shrink element facets towards centroids in subsequent views, shrink <value>.
SIP	Select shell integration point for contour; SIP <#>.

SLICE	Slice model a z-minimum plane; slice <value in normalized model dimension> this feature is removed after using restore. Slice enables internal details for brick elements to be used to generate new polygons on the slice plane.
SNORMAL	Set or unset display of shell direction normals to indicate topology order.
SPOT	Draw node numbers on model spot <first #> <last # for range>.
TAURUS	LS-DYNA database, TAU <state #>, or state <state #>, reads LS-TAURUS file to extract previous state data.
TRIAD	Set or unset display of axis triad.
TSHELL	Set or unset shell element thickness simulation in subsequent views.
TV	Change display type.
TX	Translates model along x axis.
TY	Translates model along y axis.
TZ	Translates model along z axis.
V	Display model using painters algorithm.
VECTOR v or d	View with vector arrows of velocity or displacement; <v> or <d>.
ZB	Switch on or off zbuffer algorithm for subsequent view; or draw commands.
ZIN	Zoom in using mouse to set display size and position.
ZMA	Set position of zmax plane; ZMAX <value in normalized model dimensions>.
ZMI	Set position of zmin plane; ZMIN <value in normalized model dimensions>.
ZOUT	Zoom out using mouse to set displays size expansion and position.

APPENDIX 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-DYNA are directly called by this driver, the behavior of the constitutive model is precisely that which can be expected in actual applications. In the current implementation the constitutive subroutines for both shell elements and solid elements can be examined.

INPUT DEFINITION

The material model driver is invoked by setting the total number of beam, shell, and solid elements to zero in a standard LS-DYNA input file. The number of material model definitions should be set to one, the number of load curves should be nine, and the termination time to the desired length of the driver run. The complete state dump interval is interpreted as the time step to be used in the material model driver run. Plotting information is saved for every step of a driver run and sufficient memory is allocated to save this information in core for the interactive plotting phase.

The input deck consists only of the TITLE card, the CONTROL cards, one MATERIAL DEFINITION, and NINE LOAD CURVES describing the strain path should be defined. These nine curves define the time history of the displacement gradient components shown in Table 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 c ₁ c ₂	Plot component c ₁ versus c ₂ .
ECOMP	<p>Display component numbers on the graphics display:</p> <ul style="list-style-type: none"> 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 $\ln(v/v_0)$, 18 relative volume, 19 $v_0/v - 1.0$, 20 1st history variable, 21 2nd history variable. <p>Adding 100 or 400 to component numbers 1-16 yields strains and strain rates, respectively.</p>
FILE name	Change pampers filename to name for printing.
GRID	Graphics displays will be overlaid by a grid of orthogonal lines.
NOGRID	Graphics displays will not be overlaid by a grid of orthogonal lines.

OSCL	Scale all ordinate data by f. Default is f=1.
OSET omin omax	Set min and max values on ordinate to omin and omax, respectively. If omin=omax=0, scaling is automatic.
PRINT	Print plotted time history data into file "pampers." Only data plotted after this command is printed. File name can be changed with the "file" command.
QUIT, END, T	Exit the material model driver program.
RDLC m n r ₁ z ₁ ... r _n z _n	Redefine load curve m using n coordinate pairs (r ₁ ,z ₁), (r ₂ ,z ₂), ..., (r _n ,z _n).
TIME c	Plot component c versus time.
TV n	Use terminal output device type n. LS-DYNA provides a list of available devices.

Presently, the material model drive is implemented for solid and shell element material models. The driver does not yet support material models for beam elements.

APPENDIX 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-DYNA.

Linking is performed via names. To these names surfaces are attached, which in turn can be linked together from many files externally to LS-DYNA. Thus, arbitrary surfaces can be provided by a preprocessor and then can be written to various files. The so called VDA file given on the LS-DYNA execution line via `V=vda` contains references to all other files. It also contains several other parameters affecting the treatment in the contact subroutines; see below.

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

```
file afile bfile {
```

```
    alias definitions
```

```
}
```

```
    alias definitions
```

```
    followed by optional runtime parameters and a final end statement.
```

The file, **afile**, is optional, and if given must be the name of an ASCII input file formatted in accordance with the VDA Surface Interface Definitions as defined by the German automobile and automotive supply industry. **bfile** is required, and is the name of a binary VDA file. In a first run **afile** is given and **bfile** is created. In any further run, if the definitions have not changed, **afile** can be dropped and only **bfile** is needed. The purpose of **bfile** is that it allows for much faster initialization if the same VDA surfaces are to be used in a future LS-DYNA run.

If **afile** is given, **bfile** will always be created or overwritten. The alias definitions are used for linking to LS-DYNA and between the various surface definitions in the files defined by **afile** and **bfile**.

The alias definitions are of the form

```
alias name { e11 e12 ... eln }
```

where **name** is any string of up to 12 characters, and e11,...,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
```

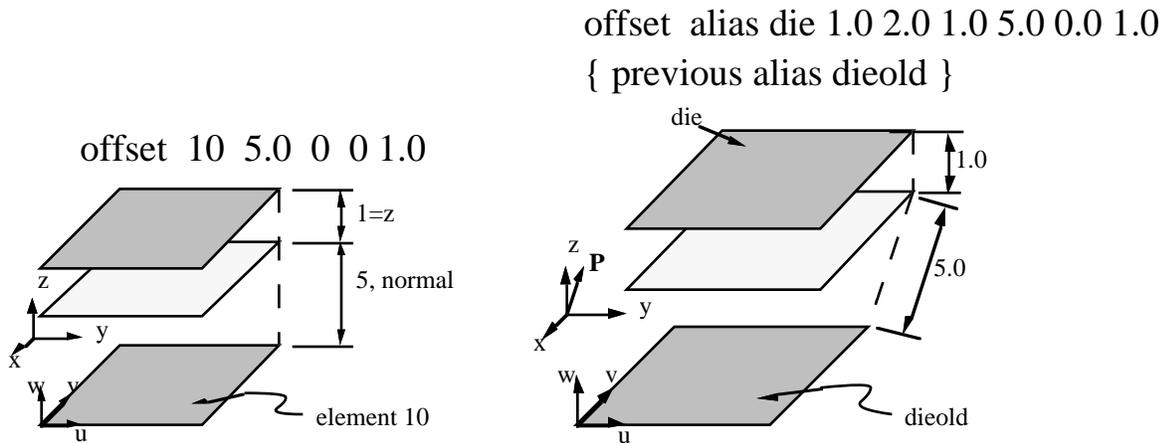
where **normal** is the amount to offset the surface along the normal direction, and **x,y,z** are the translations to be applied. The default normal direction is given by the cross product of the local u and v directions on the VDA surface, taken in that order. **normal** can be negative.

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 xc yc zc normal x y z { previous alias_name }
```

where **normal**, **x**, **y**, and **z** are defined as in the offset keyword. A reference point **xc**, **yc**, and **zc** defines a point in space which determines the normal direction to the VDA surface, which is a vector from the origin to P(xc,yc,zc). 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 bound, all VDA surfaces must have the global search performed for them. Hence, this parameter should be set to the maximum number of surfaces that any point can be expected to be near at one time (the largest number of surfaces that come together at one point). Setting **ntrack** higher will require more memory but result in faster execution. If **ntrack** is too low, performance may be unacceptably slow. The default value is 4.0.

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

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

iterate [8] Maximum number of surface iterations allowed. Since points being tracked are checked every cycle, if convergence fails it will be tried again next cycle, so setting this parameter high does not necessarily help much. On the other hand, a point converging to a crease in the VDA surface (a crease between patches with discontinuous derivative, for example) may bounce back and forth between patches up to this many times, without actually moving. Hence, this value should not be too large. The default value is 8.

el_size [t mx mn]

Controls the generation of elements where:

t =surface tolerance for mesh generation,

mx=maximum element size to generate,

mn=minimum element size to generate.

The default values are [0.25 100. 1.0]

aspect [s1 s2]

Controls the generation of elements where:

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

s2=maximum aspect ratio for any generated element.

The default values are [1.5 4.0]

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

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

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

somepatch [n] Like onepatch, but generates an element for 1 out of every [n] patches.

Example for file V=**vda**. It contains the following data:

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

Explanation:

vda1 This file contains the surfaces/face elements sur0001,sur0003, fce0006, and sur0008.

alias die face	Combines the surface/face elements sur0001, sur0003, and the offsetted surface 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 Δt	Reset the node and element data dump interval Δt .
PRTI Δt	Reset the node and element printout interval Δt .
TERM t	Reset the termination to t.

Graphics Window Controls

ESET n	Center picture at element n with a Δr by Δz window. This window is set until it is released by the unfix command or reset with another window.
FF	Encircle picture with reference grid with tickmarks. Default grid is plotted along bottom and left side of picture.
FIX	Set the display to its current window. This window is set until it is reset by the “GSET”, “FSET”, or “SETF” commands or released by the “UNFIX” command.
FSET n Δr Δz	Center display at node n with a rectangular $\Delta r \times \Delta z$ window. This window is set until it is reset with or the “UNFIX” command is typed.
GSET r z Δl	Center display picture at point (r,z) with square window of width Δl . This window is set until it is reset or the “UNFIX” command is typed.
GRID	Overlay graphics displays with a grid of orthogonal lines.
NOGRID	Do not overlay graphics displays with a grid of orthogonal lines (default).
SETF r z Δr Δz	Center display at point (r,z) with a rectangular $\Delta r \times \Delta z$ window. This window is set until it is reset or the “UNFIX” command is typed.
UNFIX	Release current display window set by the “FIX”, “GSET”, “FSET” or “SETF” commands.
UZ a b Δl	Zoom in at point (a,b) with window Δl where a, b, and Δl are numbers between 0 and 1. The picture is assumed to lie in a unit square.
UZG	Cover currently displayed picture with a 10 by 10 square grid to aid in zooming with the unity zoom, “UZ”, command.
UZOU a b Δl	Zoom out at point (a,b) with window Δl where a, b, and Δl are numbers between 0 and 1. The current window is scaled by the factor $1/\Delta l$. The picture is assumed to lie in a unit square.

Z r z Δl	Zoom in at point (r,z) with window Δl .
ZOUT r z Δl	Zoom out at point (r,z) with window Δl . The window is enlarged by the ratio of the current window and Δl . The cursor may be used to zoom out via the cursor command DZOU and entering two points with the cursor to define the window. The ratio of the current window with the specified window determines the picture size reduction. An alternative cursor command, DZZO, may be used and only needs one point to be entered at the location where the reduction (2 \times) is expected.

Graphics Window Controls for x versus y plots

The following commands apply to line plots, interface plots, etc.

ASCL f_a	Scale all abscissa data by f_a . The default is $f_a = 1$.
ASET amin amax	Set minimum and maximum values on abscissa to amin and amax, respectively. If amin=amax=0.0 (default) LS-DYNA determines the minimum and maximum values.
OSCL f_o	Scale all ordinate data by f_o . The default is $f_o = 1$.
OSET omin omax	Set minimum and maximum values on ordinate to omin and omax, respectively. If omin=omax=0.0 (default) LS-DYNA determines the minimum and maximum values.
SMOOTH n	Smooth a data curve by replacing each data point by the average of the 2n adjacent points. The default is n=0.

Mesh Display Options

ELPLT	Plot element numbers on mesh of material n.
FSOFF	Turn off the "FSON" command.
FSON	Plot only free surfaces and slideline interfaces with "O" command. (Must be used before "O" command.)
G	View mesh.
GO	View mesh right of centerline and outline left of centerline.
GS	View mesh and solid fill elements to identify materials by color.
M n	Material n is to be rezoned.
MNOFF	Do not plot material numbers with the "O", "G", and "GO" commands (default).

MNON	Plot material numbers with “O”, “G”, and “GO” commands.
NDPLT	Plot node numbers on mesh of material n.
O	Plot outlines of all material.
RPHA	Reflect mesh, contour, fringe, etc., plots about horizontal axis. Retyping “RPHA” turns this option off.
RPVA	Reflect mesh, contour, fringe, etc., plots about vertical axis. Retyping “RPVA” turns this option off.
TN r z Δ	Type node numbers and coordinates of all nodes within window ($r \pm \Delta/2$, $z \pm \Delta/2$).
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.
BLN 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 ₁ l ₁ ... f _n l _n	Delete n element blocks consisting of element numbers f ₁ to l ₁ , f ₂ to l ₂ ... , and f _n to l _n inclusive. These elements will be inactive when the calculation resume.
DE e ₁ e ₂	Delete elements e ₁ to e ₂ .
DMB n m ₁ m ₂ ... m _n	Delete n material blocks consisting of all elements with material numbers m ₁ , m ₂ ,..., and m _n . These materials will be inactive when the calculations resume.
DM n m ₁ m ₂ ... m _n	Delete n materials including m ₁ , m ₂ ,..., and m _n .
DZER k d incr nrow	Delete element row where k is the kept element, d is the deleted element, incr is the increment, and nrow is the number of elements in the row.
DZLN number n ₁ n ₂ n ₃ ...n _{last}	Delete nodal row where number is the number of nodes in the row and n ₁ , n ₂ , ... n _{last} are the ordered list of deleted nodes.

DZNR l j incr	Delete nodal row where l is the first node in the row, j is the last node in the row, and incr is the increment.
R	Restore original mesh.
S	Smooth mesh of material n. To smooth a subset of elements, a window can be set via the “GSET”, “FSET”, OR “SETF” commands. Only the elements lying within the window are smoothed.

Boundary Modifications

A	Display all slidelines. Slave sides are plotted as dashed lines.
B	Determine boundary nodes and sides of material n and display boundary with nodes and side numbers.
BD m n	Dekink boundary from boundary node m to boundary node n (counterclockwise).
BDS s	Dekink side s.
DSL n l ₁ l ₂ ...l _n	Delete n slidelines including slideline numbers l ₁ l ₂ ..., and l _n .
ER m n	Equal space in r-direction boundary nodes m to n (counterclockwise).
ERS s	Equal space in the r-direction boundary nodes on side s.
ES m n	Equal space along boundary, boundary nodes m to n (counterclockwise).
ESS s	Equal space along boundary, boundary nodes on side s.
EZ m n	Equal space in z-direction boundary nodes m to n (counterclockwise).
EZS s	Equal space in the z-direction boundary nodes on side s.
MC n	Check master nodes of slideline n and put any nodes that have penetrated through the slave surface back on the slave surface.
MD n	Dekink master side of slideline n. After using this command, the SC or MC command is sometimes advisable.
MN n	Display slideline n with master node numbers.
SC n	Check slave nodes of slideline n and put any nodes that have penetrated through the master surface back on the master surface.

SD n	Dekink slave side of slideline n; after using this command, the SC or MC command is sometimes advisable.
SLN m n	Equal space boundary nodes between nodes m to n on a straight line connecting node m to n.
SLNS n	Equal space boundary nodes along side n on a straight line connecting the corner nodes.
SN n	Display slideline n with slave node numbers.
VS m n r	Vary the spacing of boundary nodes m to n such that r is the ratio of the first segment length to the last segment length.
VSS s r	Vary the spacing of boundary nodes on side s such that r is the ratio of the first segment length to the last segment length.

MAZE Line Definitions

B	Determine boundary nodes and sides of material n and display boundary with nodes and side numbers. See command "M".
LD n k l	Line definition n for MAZE includes boundary nodes k to l
LDS n l	Line definition n for MAZE consists of side number l.
M n	Material n is active for the boundary command B.

Calculation Graphics Display Control Parameters

MOLP	Overlay the mesh on the contour, fringe, principal stress, and principal strain plots. Retyping "MOLP" turns this option off.
NLOC	Do not plot letters on contour lines.
NUMCON n	Plot n contour levels. The default is 9.
PLOC	Plot letters on contour lines to identify their levels (default).
RANGE r1 r2	Set the range of levels to be between r1 and r2 instead of in the range chosen automatically by LS-DYNA. To deactivate this command, type RANGE 0 0.

Calculation Graphics Display

CONTOUR c n m ₁ m ₂ ...m _n	Contour component number c on n materials including materials m ₁ , m ₂ , ..., m _n . If n is zero, only the outline of material m ₁ with contours is plotted. Component numbers are given in Table 1.
FRINGE c n m ₁ m ₂ ...m _n	Fringe component number c on n materials including m ₁ , m ₂ ,...,m _n . If n is zero, only the outline of material m ₁ with contours is plotted. Component numbers are given in Table 1.
IFD n	Begin definition of interface n. If interface n has been previously defined, this command has the effect of destroying the old definition.
IFN l m	Include boundary nodes l to m (counterclockwise) in the interface definition. This command must follow the “B” command.
IFP c m	Plot component c of interface m. Component numbers are given in Table 2.
IFS m	Include side m in the interface definition. Side m is defined for material n by the “B” command.
IFVA r _c z _c	Plot the angular location of the interface based on the center point (r _c ,z _c) along the abscissa. Positive angles are measured counterclockwise from the y axis.
IFVS	Plot the distance along the interface from the first interface node along the abscissa (default).
LINE c n m ₁ m ₂ ...m _n	Plot variation of component c along line defined with the “NLDF”, “PLDF”, “NSDF”, or the “NSSDF” commands given below. In determining variation, consider n materials including material number m ₁ , m ₂ ,...,m _n .
NCOL n	Number of colors in fringe plots is n. The default value for n is 6 which includes colors magenta, blue, cyan, green, yellow, and red. An alternative value for n is 5 which eliminates the minimum value magenta.
NLDF n n ₁ n ₂ ...n ₃	Define line for “LINE” command using n nodes including node numbers n ₁ , n ₂ ,...,n _n . This line moves with the nodes.
NSDF m	Define line for “LINE” command as side m. Side m is defined for material n by the “B” command.
NSSDF l m	Define line for “LINE” command and that includes boundary nodes l to m (counterclockwise) in the interface definitions. This command must follow the “B” command.
PLDF n r ₁ z ₁ ...r _n z _n	Define line for “LINE” command using n coordinate pairs (r ₁ ,z ₁), (r ₂ ,z ₂), ...(r _n ,z _n). This line is fixed in space.

PRIN c n m ₁ m ₂ ...m _n	<p>Plot lines of principal stress and strain in the yz plane on n materials including materials m₁, m₂,...,m_n. If n is zero, only the outline of material m₁ 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.</p>
PROFILE c n m ₁ m ₂ ...m _n	<p>Plot component c versus element number for n materials including materials m₁, m₂,...,m_n. If n is 0 then component c is plotted for all elements. Component numbers are given in Table 1.</p>
VECTOR c n m ₁ m ₂ ...m _n	<p>Make a vector plot of component c on n materials including materials m₁, m₂,...,m_n. If n is zero, only the outline of material m₁ with vectors is plotted. Component c may be set to "D" and "V" for vector plots of displacement and velocity, respectively.</p>

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, almansis strains, strain rates, extensions, and residual strain are obtained. Maximum and minimum principal stresses and strains are in the rz plane. The corresponding hoop quantities must be examined to determine the overall extremum. ij, jk, etc. normal components are normal to the ij, jk, etc side. The peak value database must be flagged on Control Card 4 in columns 6-10 or components 71-79 will not be available for plotting.

No.	Component
1	pressure
2	shear stress
3	normal force
4	tangential force
5	y-force
6	z-force

Table 2. Component numbers for interface variables. In axisymmetric geometries the force is per radian.

Cursor Commands

DBD a b	Use cursor to define points a and b on boundary. Dekink boundary starting at a, moving counterclockwise, and ending at b.
DCN a b	Use cursor to define points a and b. The node closest to point a will be moved to point b.
DCSN n a	Move nodal point n to point a defined by the cursor.
DCNM a b	Use cursor to define points a and b. The node at point a is given the coordinate at point b.
DER a b	Use cursor to define points a and b on boundary. Equal space nodes in r-direction along boundary starting at a, moving counterclockwise, and ending at b.
DES a b	Use cursor to define points a and b on boundary. Equal space nodes along boundary starting at a, moving counterclockwise, and ending at b.
DEZ a b	Use cursor to define points a and b on boundary. Equal space nodes in z-direction along boundary starting at a, moving counterclockwise, and ending at b.
DTE a b	Use cursor to define points a and b on the diagonal of a window. The element numbers and coordinates of elements lying within the window are typed on the terminal.
DTN a b	Use cursor to define points a and b on the diagonal of a window. The node numbers and coordinates of nodal points lying within the window are typed on the terminal.
DTNC a	Use cursor to define point a. The nodal point number and nodal coordinates of the node lying closest to point a will be printed.

DVS a b r	Use cursor to define points a and b on boundary. Variable space nodes along boundary starting at a, moving 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

The two varieties of rigid body dummies available in LS-DYNA are described in this appendix. These are generated internally by including the appropriate *COMPONENT keyword. A description of the GEBOD dummies begins on this page and the HYBRID III family on page K.7.

GEBOD Dummies

Rigid body dummies can be generated and simulated within LS-DYNA using the keyword *COMPONENT_GEBOD. Physical properties of these dummies draw upon the GEBOD database [Cheng et al, 1994] which represents an extensive measurement program conducted by Wright-Patterson AFB and other agencies. The differential equations governing motion of the dummy are integrated within LS-DYNA separate from the finite element model. Interaction between the dummy and finite element structure is achieved using contact interfaces (see *CONTACT_GEBOD).

The dynamical system representing a dummy is comprised of fifteen rigid bodies (segments) and include: lower torso, middle torso, upper torso, neck, head, upper arms, forearms/hands, upper legs, lower legs, and feet. Ellipsoids are used for visualization and contact purposes. Shown in Figure 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.

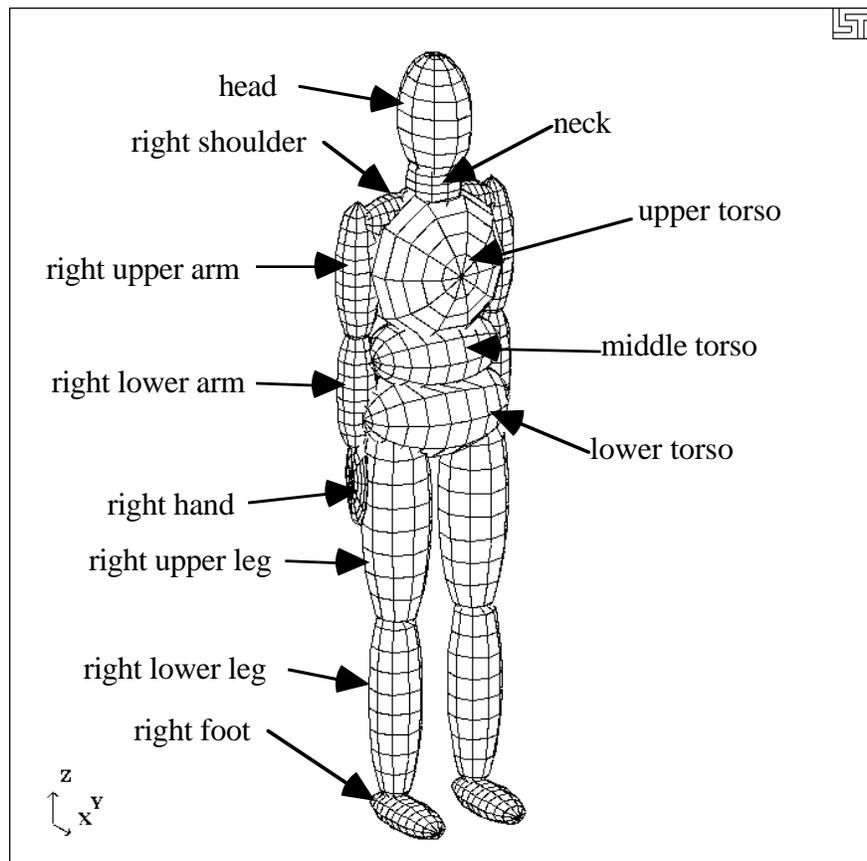


Figure K.1: 50th percentile male dummy in the nominal position.

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

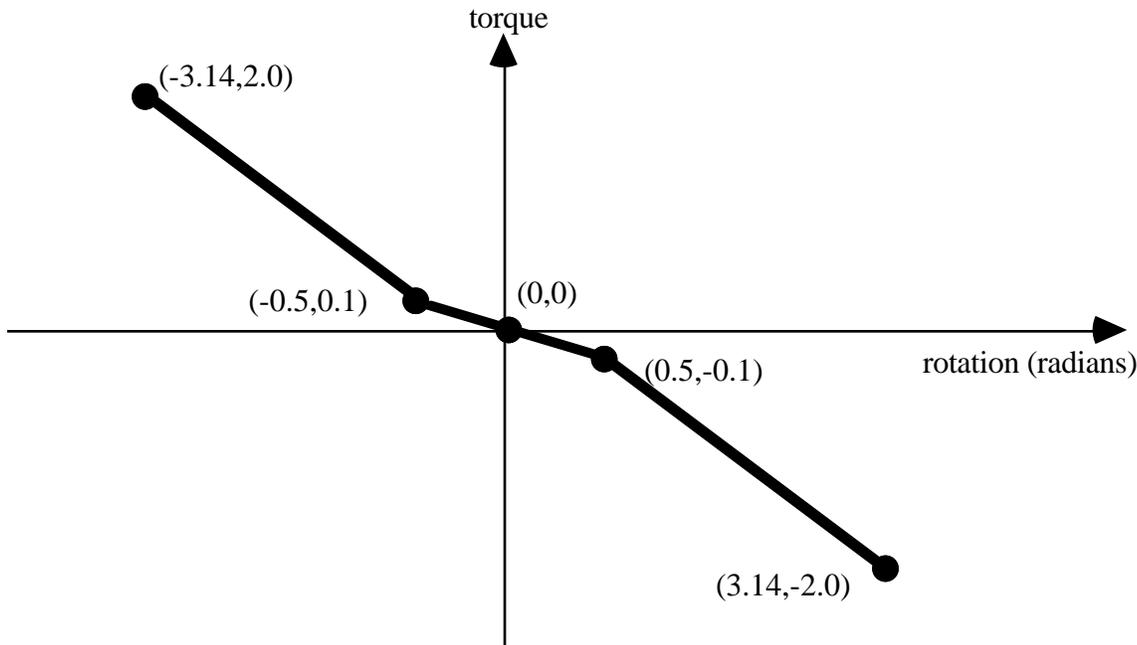


Figure K.2: Characteristic torque curve shape used by all joints.

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

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.

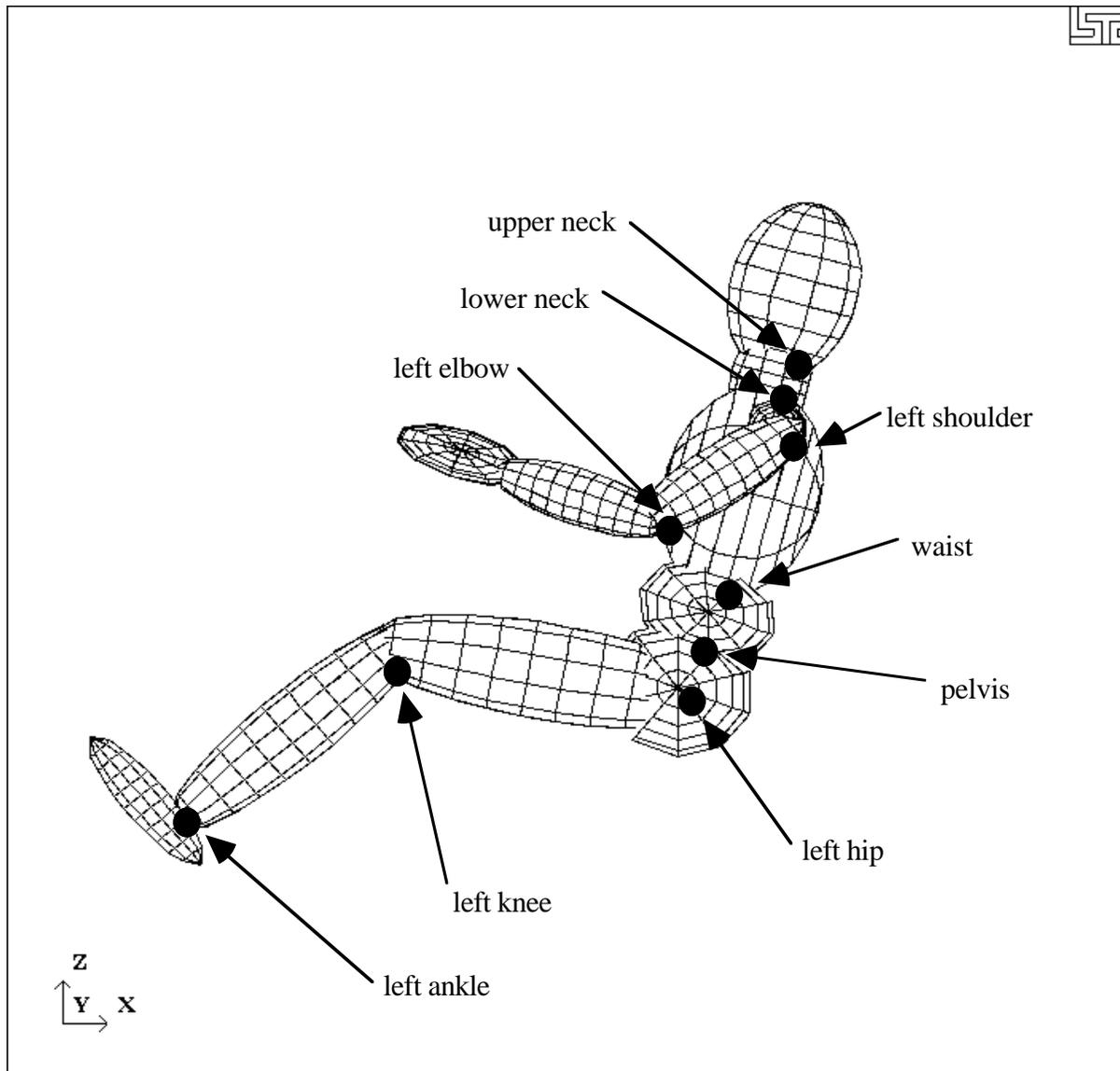


Figure K.3: Dummy seated using the file listed in Table K.3.

HYBRID III Dummies

A listing of applicable joint degrees of freedom of the Hybrid III dummy is given below.

Table K.4: Joints and associated degrees of freedom. Local axes are in parentheses.

Joint Name	Degree(s) of Freedom		
	1st	2nd	3rd
lumbar	flexion (y)	torsion (z)	
lower neck	flexion (y)	torsion (z)	
upper neck	flexion (y)	torsion (z)	
shoulders	flexion-extension (y)	abduction-adduction (x)	n/a
elbows	flexion-extension (y)	n/a	n/a
wrists	flexion-extension (x)	n/a	n/a
hips	abduction-adduction (x)	medial-lateral rotation (z)	flexion-extension (y)
knees	flexion-extension (y)	n/a	n/a
ankles	inversion-eversion (x)	medial-lateral rotation (z)	dorsi-plantar flexion (y)
ribcage	translation (x)	rotation (y)	rotation (z)

Joint springs of the *COMPONENT_HYBRIDIII dummies are formulated in the following manner.

$$T = a_{lo}(q - q_{lo}) + b_{lo}(q - q_{lo})^3 \quad q \leq q_{lo}$$

$$T = a_{hi}(q - q_{hi}) + b_{hi}(q - q_{hi})^3 \quad q \geq q_{hi}$$

$$T = 0 \quad q_{lo} < q < q_{hi}$$

where

T is the joint torque

q is the joint generalized coordinate

a_{lo} and b_{lo} are the linear and cubic coefficients, respectively, for the low regime

a_{hi} and b_{hi} are the linear and cubic coefficients, respectively, for the high regime

q_{lo} and q_{hi} are the activation values for the low and high regimes, respectively

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

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's Manual. If you have questions about this guide, find errors or omissions in it, please email manual@lstc.com.

Supported Features

The only input formats currently supported are 920 and later, including keyword. Models in any of the older formats will need to be converted to one of these input format before then can be run with the current version of LS-DYNA for massively parallel processors, mpp.

The large majority of LS-DYNA options are available on MPP computers. Those that are not supported are being systematically added. Unless otherwise noted here, all the options of LS-DYNA version 93x are supported by MPP/LS-DYNA

Here is the list of **unsupported** features:

- *ALE
- *BOUNDARY_CONVECTION
- *BOUNDARY_FLUX
- *BOUNDARY_RADIATION
- *BOUNDARY_USA_SURFACE
- *CONTACT_TIEBREAK_NODES_TO_SURFACE
- *CONTACT_TIEBREAK_SURFACE_TO_SURFACE
- *CONTACT_1D
- *DATABASE_AVIS
- *DATABASE_MOVIE
- *DATABASE_MPGS
- *DATABASE_TRACER
- *DATABASE_BINARY_XTFIELD
- *INTERFACE_JOY
- *LOAD_SUPERELASTIC_OPTION
- *USER
- *PART_REPOSITION
- *TERMINATION_NODE

MPP/LS-DYNA can restart; however, the restart options are somewhat 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

MPP/LS-DYNA uses a completely redesigned, highly parallel contact algorithm. The contact options currently **unsupported** include:

- *CONTACT_TIEBREAK_SURFACE_TO_SURFACE
- *CONTACT_FORCE_TRANSDUCER_CONSTRAINT

Because these options are all supported via the new, parallel contact algorithms, slight differences in results may be observed as compared to the serial and SMP versions of LS-DYNA. Work has been done to minimize these differences, but they may still be evident in some models.

For each of the supported CONTACT_ control cards, there is an optional string _MPP which can be appended to the end. Adding these characters triggers the reading of a new control card immediately following (but after the TITLE card, if any). This card contains 5 integer parameters in I10 format.

The parameters are:

t r a c k p e n

If 1, any initial penetrations for slave nodes are compensated for in the contact algorithm.

No nodes are moved to eliminate penetrations, and no initial penetration checking is performed. The algorithm detects these penetrations and allows for them in computing forces, so excessively large forces are avoided. As the slave node moves in such a way as to reduce or eliminate the penetration, the full contact distance/material thickness is imposed. Use of this option is encouraged as it can greatly help stability, particularly in models with many initial penetrations. By default this option is disabled.

`bucket`

Bucket sorting frequency for this contact interface

`lcbucket`

Load curve giving bucket sort frequency as a function of simulation time. Currently this option is not supported by any of the contact algorithms.

`nseg2track`

Number of contact segments to track for each slave node

`inititer`

Number of iterations for initial penetration checking

The defaults for each are taken from the corresponding options in the pfile (described below). For example, if you had the control card:

```
*CONTACT_SINGLE_SURFACE_TITLE
```

```
This is my title card
```

you could change this to

```
*CONTACT_SINGLE_SURFACE_TITLE_MPP
```

```
This is my title card
```

```
1
```

to turn on the initial penetration tracking option. The serial/SMP code will ignore these options.

Output Files and Post-Processing

For performance reasons, many of the ASCII output files normally created by LS-DYNA have been combined into a new binary format used by MPP/LS-DYNA. There is a post-processing program `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
- *DATABASE_NODOUT
- *DATABASE_ELOUT
- *DATABASE_GLSTAT
- *DATABASE_DEFORC
- *DATABASE_MATSUM
- *DATABASE_NCFORC
- *DATABASE_RCFORC
- *DATABASE_DEFGEO
- *DATABASE_ABSTAT
- *DATABASE_NODOFR
- *DATABASE_RBDOUT
- *DATABASE_SLEOUT
- *DATAGASE_JNTFORC
- *DATABASE_SBTOUT

Some of the normal LS-DYNA files will have corresponding collections of files produced by MPP/LS-DYNA, with one per processor. These include the d3dump files (new names = d3dump.nnnn), the messag files (now 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=80m* and *memory2=20m*. To run the same problem on 16 processors, you still need *memory=80m*, but can set *memory2=6m*. The value for *memory2* drops nearly linearly with the number of processors used to run the program, which works well for shared-memory systems.

The full deck restart capability is supported by the MPP version of LS-DYNA, but in a manner slightly different than the SMP code. Each time a restart dump file is written, a separate restart file is also written with the base name D3FULL. For example, when the third restart file *d3dump03* is written (one for each processor, *d3dump03.0000*, *d3dump03.0001*, etc), there is also a single file written named *d3full03*. This file is required in order to do a full deck restart and the *d3dump* files are not used in this case by the MPP code. In order to perform a full deck restart with the MPP code, you first must prepare a full deck restart input file as for the serial/SMP version. Then, instead of giving the command line option *r=d3dump03* you would use the special option *n=d3full03*. The presence of this command line option tells the MPP code that this is a restart, not a new problem, and that the file *d3full03* contains the geometry and stress data carried over from the previous run.

PFILE

There is a new command line option: *p=pfile*. *pfile* contains MPP specific parameters that affect the execution of the program. The file is split into sections, with several options in each section. Currently, these sections: **directory**, **decomposition**, **contact**, and **general** are available. First, here is a sample *pfile*:

```
directory {
  global rundir
  local /tmp/rundir
}
contact {
  inititer 3
}
```

The file is case insensitive and free format input, 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.** Holds directory specific options

global *path*

Relative path to a directory accessible to all processors. This directory will be created if necessary.

Default = current working directory

local *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.

Default = global path

- **decomposition** Holds decomposition specific options

file *filename*

The name of the file that holds the decomposition information. This file will be created in the current working directory if it does not exist. If the filename does not end with the extension *.pre* then this extension is added. If this option is not specified, MPP/LS-DYNA will perform the decomposition.

Default = None

numproc *n*

The problem will be decomposed for *n* processors. If $n > 1$ and you are running on 1 processor, or if the number of processors you are running on does not evenly divide *n*, then execution terminates immediately after decomposition. Otherwise, the decomposition file is written and execution continues. For a decomposition only run, both numproc and file should be specified.

Default = the number of processors you are running on.

costinc *n*

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.

Default = 0

method *name*

Currently, there are two decomposition methods supported, namely *rcb* and *greedy*. Method *rcb* is Recursive Coordinate Bisection. Method *greedy* is a simple neighborhood expansion algorithm. The impact on overall runtime is problem dependent, but *rcb* generally gives the best performance.

Default = rcb

expdir *n*

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.

Default = 1

This is a deprecated feature. See the section below on Special Decompositions.

expsf *t*

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.

Default = 1

This is a deprecated feature. See the section below on Special Decompositions.

rx ry rz sx sy sz c2r s2r 3vec mat

See the section below on Special Decompositions for details about these decomposition options.

show

If this keyword appears in the decomposition section, the d3plot file is doctored so that the decomposition can be viewed with the post processor. Displaying material 1 will show that portion of the problem assigned to processor 0, and so on. The problem will not actually be run, but the code will terminate once the initial d3plot state has been written.

rcblog *filename*

This option is ignored unless the decomposition method is RCB. If the indicated file does not exist, then a record is stored of the steps taken during decomposition. If the file exists, then this record is read and applied to the current model during decomposition. This results in a decomposition as similar as possible between the two runs. For example, suppose a simulation is run twice, but the second time with a slightly different mesh. Because of the different meshes the problems will be distributed differently between the processors, resulting in slightly different answers due to roundoff errors. If an rcblog is used, then the resulting decompositions would be as similar as possible.

slist *n1,n2,n3,...*

This option changes the behavior of the decomposition in the following way. *n1,n2,n3* must be a list of sliding interfaces occurring in the model (numbered according to the order in which they appear, starting with 1) delimited by commas and containing no spaces (eg "1,2,3" but not "1, 2, 3"). Then all elements belonging to the first interface listed will be distributed across all the processors. Next, elements belonging to the second listed interface will be distributed among all processors, and so on, until the remaining elements in the problem are distributed among the processors. Up to 5 interfaces can be listed. It is generally recommended that at most 1 or 2 interfaces be listed, and then only if they contribute substantially to the total computational cost. Use of this option can increase speed due to improved load balance.

sidist *n1,n2,n3,...*

This is the opposite of the silist option: the indicated sliding interfaces are each forced to lie wholly on a single processor (perhaps a different one for each interface). This can improve speed for very small interfaces by reducing synchronization between the processors.

vspeed

If this option is specified a brief measurement is taken of the performance of each processor by timing a short floating point calculation. The resulting information is used during the decomposition to distribute the problem according to the relative speed of the processors.

• contact

Holds contact specific options. Because the MPP contact algorithms are in some ways significantly different than the serial/SMP algorithms, many of the parameters available on the contact control cards do not make sense for the MPP code, and so are ignored. There is a way to specify MPP specific contact options on a per contact interface basis. These options in the pfile provide default values for many of these options.

bucket *n*

Specifies the frequency for bucket sort contact searches.

Default = 200 (5 for type a13)

ntrack *n*

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.

Default = 3

inititer *n*

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.

Default = 2

- **general** Holds general options

nodump

If this keyword appears, all restart dump file writing will be suppressed

nofull

If this keyword appears, writing of d3full (full deck restart) files will be suppressed.

nofail

If this keyword appears, the check for failed elements in the contact routines will be skipped. This can improve efficiency if you do not have element failure in the model.

swapbytes

If this keyword appears, the d3plot and interface component analysis files are written in swapped byte ordering.

Special Decompositions

These options appear in the "decomposition" section of the pfile and are only valid if the decomposition method is **rcb**. The rcb decomposition method works by recursively dividing the model in half, each time slicing the current piece of the model perpendicularly to one of the three coordinate axes. It chooses the axis along which the current piece of the model is longest. The result is that it tends to generate cube shaped domains aligned along the coordinate axes. This is inherent in the algorithm, but is often not the behavior desired.

This situation is addressed by providing a set of coordinate transformation functions which are applied to the model before it is decomposed. The resulting deformed geometry is then passed to the decomposition algorithm, and the resulting domains are mapped back to the undeformed model. As a simple example, suppose you wanted rectangular domains aligned along a line in the xy plane, 30 degrees from the x axis, and twice as long along this line as in the other two dimensions. If you applied these transformations:

```
sx 0.5  
rz -30
```

then you would achieve the desired effect. Any number of transformations may be specified. They are applied to the model in the order given. The transformations available are:

sx t

scale the current x coordinates by t .

sy t

scale the current y coordinates by t .

sz t

scale the current z coordinates by t .

rx t

rotate around the current x axis by t degrees.

ry t

rotate around the current y axis by t degrees.

rz t

rotate around the current z axis by t degrees.

mat m11 m12 m13 m21 m22 m23 m31 m32 m33

transform the coordinates by matrix multiplication:

transformed		original
x	m11 m12 m13	x
y =	m21 m22 m23	y
z	m31 m32 m33	z

3vec v11 v12 v13 v21 v22 v23 v31 v32 v33

Transform the coordinates by the inverse of the transpose matrix:

original		transformed
x	v11 v21 v31	x
y =	v12 v22 v32	y
z	v13 v23 v33	z

This appears complicated, but in practice is very intuitive: instead of decomposing into cubes aligned along the coordinate axes, `rcb` will decompose into parallelipipeds whose edges are aligned with the three vectors $(v11, v12, v13)$, $(v21, v22, v23)$, and $(v31, v32, v33)$. Furthermore, the relative lengths of the edges of the decomposition domains will correspond to the relative lengths of these vectors.

C2R x0 y0 z0 vx1 vy1 vz1 vx2 vy2 vz2

The part is converted into a cylindrical coordinate system with origin at $(x0, y0, z0)$, cylinder axis $(vx1, vy1, vz1)$ and $\theta=0$ along the vector $(vx2, vy2, vz2)$. You can think of this as tearing the model along the $(vx2, vy2, vz2)$ vector and unwrapping it around the $(vx1, vy1, vz1)$ axis. The effect is to create decomposition domains that are "cubes" in cylindrical coordinates: they are portions of cylindrical shells. The actual transformation is:

$$\text{new } (x,y,z) = \text{cylindrical coordinates } (r,\theta,z)$$

Knowing the order of the coordinates is important if combining transformations, as in the example below.

S2R x0 y0 z0 vx1 vy1 vz1 vx2 vy2 vz2

Just like the above, but for spherical coordinates. The (vx1,vy1,vz1) vector is the phi=0 axis.

New (x,y,z) = spherical coordinates (rho, theta, phi)

Examples:

The old "expdir 2 expsf 5.0" is exactly equivalent to "sy 5.0"

```
rz 45
```

will generate domains rotated -45 degrees around the z axis.

```
C2R 0 0 0 0 0 1 1 0 0
```

will generate cylindrical shells of domains. They will have their axis along the vector (0,0,1), and will start at the vector (1,0,0) Note that the part will be cut at (1,0,0), so no domains will cross this boundary. If there is a natural boundary or opening in your part, the "theta=0" vector should point through this opening. Note also that if the part is, say, a cylinder 100 units tall and 50 units in radius, after the C2R transformation the part will fit inside the box $x=[0,50]$, $y=[0, 2\text{PI}]$, $z=[0,100]$. In particular, the new y coordinates (theta) will be very small compared to the other coordinate directions. It is therefore likely that every decomposition domain will extend through the complete transformed y direction. This means that each domain will be a shell completely around the original cylinder. If you want to split the domains along radial lines, try this pair of transformations:

```
C2R 0 0 0 0 0 1 1 0 0
```

```
SY 5000
```

This will do the above C2R, but then scale y by 5000. This will result in the part appearing to be about 30,000 long in the y direction -- long enough that every decomposition domain will divide the part in this (transformed) y direction. The result will be decomposition domains that are radial "wedges" in the original part.

General combinations of transformations can be specified, and they are applied in order:

```
SX 5 SY .2 RZ 30
```

will scale x, then y, then rotate.

Execution of MPP/LS-DYNA

MPP/LS-DYNA runs under a parallel environment which provided by the hardware vendor. The execution of the program therefore varies from machine to machine. On some platforms, command line parameters can be passed directly on the command line. For others, the use of the names file is required. The names file is supported on all systems.

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

```
rsh <machine name> kill -INT <PID>
```

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

Here is a simple table to show how to run the program on various platforms. Of course, scripts are often written to mask these differences.

Platform	Execution Command
DEC Alpha	<code>dmpirun --np <i>n</i> mpp-dyna</code>
Fujitsu	<code>jobexec -vp <i>n</i> -mem <i>m</i> mpp-dyna</code>
Hitachi	<code>mpirun --np <i>n</i> mpp-dyna</code>
HP	<code>mpp-dyna -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 mpp-dyna</code>
NEC	<code>mpirun --np <i>n</i> mpp-dyna</code>
SGI	<code>mpirun --np <i>n</i> mpp-dyna</code>
Sun	<code>tmrn -n-p <i>n</i> mpp-dyna</code>

Where *n* is number of processors, *mpp-dyna* is the name of the MPP/LS-DYNA executable and *m* is the MB of real memory.

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.

The implicit analysis capability was first released in Version 950. Initially targeted at metal forming springback simulation, this new capability allowed static stress analysis. Version 970 adds many additional implicit features, including new element formulations for linear and modal analysis.

For best implicit performance, it is important to provide enough memory to allow the stiffness matrix factorization to run in-core. In most cases, the default memory size must be increased. See the Linear Equation Solver section below.

SETTING UP AN IMPLICIT SIMULATION

The keyword `*CONTROL_IMPLICIT_GENERAL` is used to activate the implicit method. LS-DYNA can conduct either a linear or a nonlinear implicit analysis. The keyword `*CONTROL_IMPLICIT_SOLUTION` is used to select between these implicit analysis types. In addition, an implicit eigenvalue analysis can be performed to extract frequencies and mode shapes.

To perform a linear implicit analysis, use the `*CONTROL_IMPLICIT_GENERAL` keyword to activate the implicit method and to specify the time step size. Enter the termination time using the `*CONTROL_TERMINATION` keyword. For a single step analysis, select the step size to be equal to the termination time. Use the `*CONTROL_IMPLICIT_SOLUTION` keyword to request a linear analysis. Select linear element formulations using the `*SECTION_SOLID` and/or `*SECTION_SHELL` keywords. For best accuracy, a double precision version of LS-DYNA should be used for linear analysis.

To perform an eigenvalue analysis, use the `*CONTROL_IMPLICIT_GENERAL` keyword to activate the implicit method and to specify a time step size. Enter the termination time using the `*CONTROL_TERMINATION` keyword (the time step size and termination time must be nonzero, but will otherwise be ignored as LS-DYNA will presently just compute the eigenvalues and stop.) Use the `*CONTROL_IMPLICIT_EIGENVALUE` keyword to indicate the desired number of eigenvalues and frequency ranges of interest. For best accuracy, a double precision version of LS-DYNA should be used for eigenvalue analysis.

A nonlinear implicit simulation is typically divided into several steps. In a dynamic simulation, these are *time steps*. In a static simulation, these are *load steps*. Multiple steps may be used to divide the nonlinear behavior into manageable pieces, to obtain results at intermediate stages during the simulation, or perhaps to resolve a particular frequency of motion in dynamic simulations. In each step, an equilibrium geometry is sought which balances internal and external forces in the model. The *nonlinear equation solver* performs an iterative search using one of several Newton based methods. *Convergence* of this iterative process is obtained when norms of displacement and/or energy fall below user-prescribed tolerances.

Control parameters for the nonlinear equation solver are input using the keyword `*CONTROL_IMPLICIT_SOLUTION`. By default, the progress of the equilibrium search is not shown to the screen. This output can be activated either using the `NLPRINT` input parameter, or interactively toggled on and off by entering “<ctrl-c> nlprint”. The box below shows a typical iteration sequence, where the norms of displacement (du/u) and energy (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_SOLVER`. Several different linear equation solvers are available, including direct (Gaussian elimination) and iterative (conjugate gradient, Lanczos) methods. A sparse storage scheme is used to minimize memory requirements, which are still often substantial. Two options are available for matrix reordering, allowing nodes and elements to be numbered arbitrarily by the user.

It is very important to allow enough memory for the stiffness matrix factorization to run incore. Although the direct solvers can run out-of-core, using disk files for scratch space, this can slow performance by 100x or more. To view memory requirements for a particular model, select

LPRINT=1 on *CONTROL_IMPLICIT_SOLVER, or interactively type “<ctrl-c> lprint”. Summary information will be printed to the screen and messag files. Use the command line option “memory=...” to increase memory until this summary reports that the TOTAL AVAILABLE memory is large enough that the solver runs “INCORE”. The memory size may also be specified on *KEYWORD.

NONLINEAR EQUATION SOLVER

Several different nonlinear equation solvers are available for finding equilibrium within each step. All are iterative in nature. In the *full Newton method*, a new stiffness matrix is formed and inverted each equilibrium iteration. This is the most costly method, but can require fewer iterations to reach equilibrium. In the *modified Newton method*, several iterations are performed using the same stiffness matrix. After each iteration, the geometry is updated using $\Delta \mathbf{u}$ and a new \mathbf{R} is computed. This approach reduces cost by avoiding some forming and factoring of the stiffness matrix \mathbf{K} , but usually requires more iterations to reach equilibrium.

The default nonlinear equation solver is the BFGS solver, which uses a *quasi-Newton method*. In this method, the inverted stiffness matrix \mathbf{K} is used for several iterations, but is improved after each iteration using an inexpensive rank two update. If convergence is not reached after 10 iterations, or if *divergence* (increasing \mathbf{R}) is detected, then a new stiffness matrix is automatically formed and inverted. This hybrid method combines the efficiency of the modified Newton method with the reliability of the full Newton method. The number of iterations between stiffness matrix reformations is a user input, defaulting to 10. If a value of one is chosen, then the full Newton method is recovered.

```

      BEGIN time step      1
      =====
            time = 1.00000E+00
      current step size = 1.00000E+00

Iteration:  1      *|du|/|u| = 2.5517753E+00      *Ei/E0 = 1.0000000E+00

DIVERGENCE (increasing residual norm) detected:
      |{Fe}-{Fi}| ( 7.5426269E+03) exceeds |{Fe}| ( 5.0000000E+00)
      automatically REFORMING stiffness matrix...

Iteration:  2      *|du|/|u| = 6.0812935E-01      *Ei/E0 = 4.0526413E-01
Iteration:  4      *|du|/|u| = 1.0974191E-02      *Ei/E0 = 2.3907781E-04
Iteration:  5      *|du|/|u| = 1.0978787E-02      *Ei/E0 = 1.7910795E-04
Iteration:  6      *|du|/|u| = 4.2201181E-03      *Ei/E0 = 4.2557768E-05
Iteration:  7      *|du|/|u| = 4.1142219E-03      *Ei/E0 = 3.0658711E-05
Iteration:  8      *|du|/|u| = 1.9794757E-03      *Ei/E0 = 9.1215551E-06
Iteration:  9      *|du|/|u| = 1.7957653E-03      *Ei/E0 = 6.1669480E-06
Iteration: 10      *|du|/|u| = 1.2022830E-03      *Ei/E0 = 2.9031284E-06

ITERATION LIMIT reached, automatically REFORMING stiffness matrix...

Iteration: 11      *|du|/|u| = 5.4011414E-04      *Ei/E0 = 1.0553019E-06
    
```

The print-out above shows typical behavior of the default BFGS nonlinear equation solver. Two automatic stiffness reformations are performed, initially due to divergence, and later when the default limit of 10 iterations is exceeded. By default, the progress of the equilibrium search is not shown to the screen. This output can be activated either using the NLPRINT input parameter, or interactively toggled on and off by entering “<ctrl-c> nlprint”.

$$\mathbf{K}_{n+1}^{-1} = (\mathbf{I} + \mathbf{w}\mathbf{v}^T)\mathbf{K}_n^{-1}(\mathbf{I} + \mathbf{v}\mathbf{w}^T)$$

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.

ELEMENT FORMULATIONS FOR IMPLICIT ANALYSIS

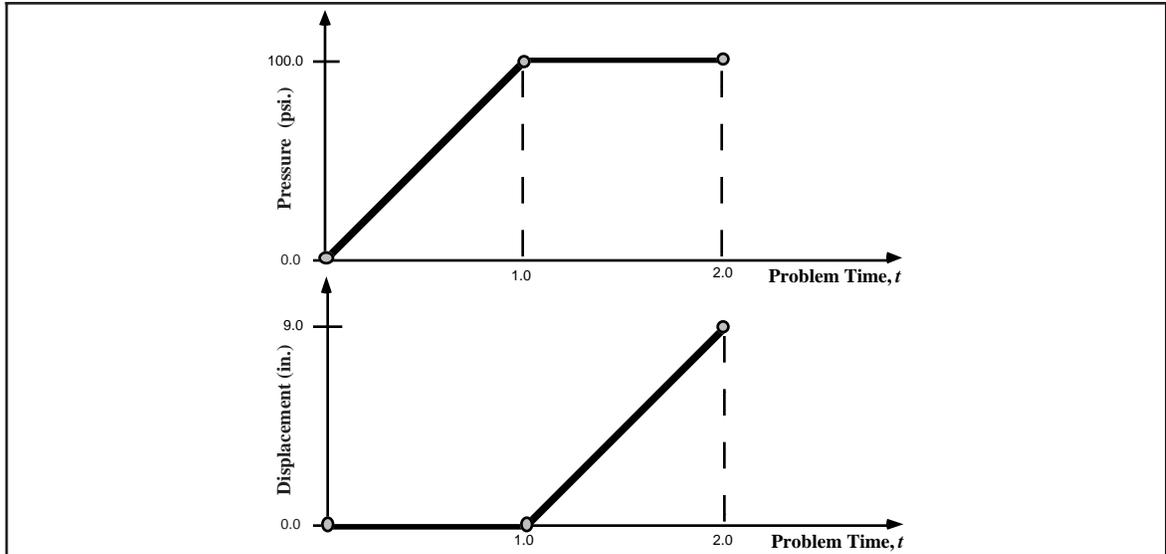
The default element formulations in LS-DYNA are highly efficient, using single point integration. For implicit analysis it is generally more effective to use more expensive element formulations which are less susceptible to hourglass instability. The Hughes-Liu brick element #2 and shell element #6, and the fast shell #16 are good choices for implicit analysis. Stiffness forms of hourglass control are recommended, with hourglass type #6 required for use with implicit solid elements.

APPLYING LOADS DURING IMPLICIT ANALYSIS

Loading is applied using the same keywords as in explicit analysis. Load curves are used to control the magnitude of each load as the simulation proceeds. Typically, the magnitude of each load begins at zero, and is increased to its full value at the end of the last step in the simulation. In this case, the load curve may be defined using only two points.

For example, consider a static analysis where a pressure of 100 psi. is to be applied in 4 steps. Since the analysis is static, the step size can be chosen arbitrarily. For convenience, choose a step size of 0.25, giving a termination time of 1.0. For this problem, the load curve has only two points: (0.0 , 0.0) and (1.0, 100.0). LS-DYNA will automatically use linear interpolation to determine the load magnitude at each of the intermediate steps.

In a more complex example, consider a static problem with two types of loading. First, a static pressure of 100 psi. is to be applied, followed by a prescribed displacement of 9 inches. Two load curves are used for this problem, one to control the pressure, and one for the displacement, as shown below. Notice that the displacement is prescribed to be zero while the pressure is applied, then the pressure is held constant while the displacement is applied.



Load curves for a static simulation with two loading events. For convenience in this static simulation, the pressure loading is specified to begin at time 0.0 and end at time 1.0, and the displacement begins at time 1.0 and ends at time 2.0.

AUTOMATIC TIME STEP SIZE CONTROL

In the most simple multi-step nonlinear implicit analysis, the user specifies the *termination time* using the `*CONTROL_TERMINATION` keyword, and the *time step size* using the `*CONTROL_IMPLICIT_GENERAL` keyword, and each step is the same size. But for many simulations, the degree of nonlinearity varies through the course of the analysis. In this case the step size should ideally be varied such that solving for equilibrium in each step is equally difficult. This is accomplished by invoking automatic time step control, using the `*CONTROL_IMPLICIT_AUTO` keyword.

There are two advantages to using automatic time step control. First, the time step size is automatically increased and/or decreased in response to the nonlinearity of the analysis. Nonlinearity is measured simply by the number of iterations required to reach equilibrium. An additional advantage is that if the equilibrium search fails during a time step, LS-DYNA does not terminate. Instead, the step is automatically repeated using a different step size. This process of backing up and retrying difficult steps lends much persistence to the analysis, and is often the only procedure for solving highly nonlinear problems short of adjusting the step size manually.

The input parameters for automatic time step control allow specification of the *optimum number of equilibrium iterations per step*. This indicates how hard LS-DYNA should work in each time step. If equilibrium is reached in fewer than optimum iterations, the size of the next step is increased, and likewise if the equilibrium search requires more than the optimum number of iterations, then the next step size is decreased. Minimum and maximum limits for step size are also input.

IMPLICIT STRESS INITIALIZATION

A common application of the implicit method is to perform static stress initialization for an explicit dynamic calculation. This can be done using two individual calculations, or by switching methods during a calculation. In the first approach, the keyword `*INTERFACE_SPRINGBACK_LSDYNA`

is used to generate a "dynain" output file at the end of the simulation. This file is written in keyword format at the end of the simulation, and contains *NODE, *ELEMENT, and *INITIAL_STRESS data. The dynain file can be included into a second input deck to initialize the explicit dynamic analysis.

LS-DYNA can switch "on-the-fly" between the implicit and explicit methods. To use this feature, define a curve which indicates which formulation to use as a function of simulation time. Formulation switching incurs no overhead, and may be performed several times during a simulation. See the IMFLAG parameter on the *CONTROL_IMPLICIT_GENERAL keyword for more information.

TROUBLESHOOTING CONVERGENCE PROBLEMS

Convergence of the nonlinear equilibrium iteration process presents one of the greatest challenges to using the implicit mode of LS-DYNA. Below are some useful troubleshooting approaches:

Eigenvalue Analysis

Many convergence problems in static implicit analysis are caused by unconstrained rigid body modes. These are created when an insufficient number of constraints are applied to the model, or when individual model parts are left disconnected. Eigenvalue analysis is an excellent diagnostic tool to check for these problems.

To perform an eigenvalue analysis, simply add the *CONTROL_IMPLICIT_EIGENVALUE keyword to an implicit input deck. Use the first parameter NEIGV=20 to compute the lowest 20 modes. Then view the frequencies in the output text file "eigout" and animate the mode shapes in the binary output file d3eigv using LS-PREPOST. Look for frequencies which are nearly zero. Add constraints as necessary to eliminate unconstrained motion.

D3ITER Plot Database

To diagnose convergence trouble which develops in the middle of a simulation, get a picture of the deformed mesh. Adjust the d3plot output interval to produce an output state after every step leading up to the problematic time.

An additional binary plot database named "d3iter" is available which shows the deformed mesh during each equilibrium iteration. This output is activated and de-activated interactively by entering "<ctrl-c> iteration". View this database using LS-PREPOST. Note that stress data is not included. Frequently the problem will become obvious, especially as deformation is magnified.

Prescribed Motion with Death Time

A common static analysis problem occurs when small contact gaps exist between parts at time=0. An example is a load-driven punch which deforms a panel, with a small initial contact gap. This creates instantaneous unconstrained rigid body modes until contact between parts is established. (These modes will be obvious in an eigenvalue analysis, as described above.) To overcome this problem, apply a prescribed motion boundary condition to move the parts into contact. Once contact is established, use the optional death time to "kill" the prescribed motion, and allow the applied force or pressure to provide further loading. Monitor reaction forces from the prescribed motion, and adjust the applied loads to match reasonably well at the death time.

APPENDIX N: User-Written Weld Failure
--

The addition of a user weld failure subroutine into LS-DYNA is relatively simple. The UWELDFAIL subroutine is called every time step when OPT=2 is specified in MAT_SPOTWELD. As data, the identification number for the spotweld material, six constants specified in the input by the locations NRR through MTT, the radius of the cross section of the spotwelds, the current time, and the current values of the resultants for the spotwelds, which are stored in array STRR, are passed to the subroutine. The subroutine loops over the welds from LFT through LLT, and sets the values of the failure flag array FLAG.

```

      SUBROUTINE UWELDFAIL (IDWELD, STRR, FAIL, FIBL, CM, TT, LFT, LLT)
C*****
C|  LIVERMORE SOFTWARE TECHNOLOGY CORPORATION  (LSTC)
C|  -----
C|  COPYRIGHT 2002 JOHN O. HALLQUIST, LSTC
C|  ALL RIGHTS RESERVED
C*****
C
C***  SPOTWELD FAILURE ROUTINE
C
C***  LOCAL COORDINATES: X IS TANGENT TO BEAM, Y & Z ARE NORMAL
C
C***  VARIABLES
C      IDWELD ---- WELD ID NUMBER
C      STRR ----- STRESS RESULTANTS
C                  (1) AXIAL (X DIRECTION) FORCE
C                  (2) Y SHEAR FORCE
C                  (3) Z SHEAR FORCE
C                  (4) MOMENT ABOUT Z
C                  (5) MOMENT ABOUT Y
C                  (6) TORSIONAL RESULTANT
C      FAIL ----- FAILURE FLAG
C                  =0 NOT FAILED
C                  =1 FAIL ELEMENT
C      FIBL ----- LOCATION (1,*) GIVES THE SPOTWELD DIAMETER
C      CM ----- 6 CONSTANTS SUPPLIED BY USER
C      TT ----- CURRENT SIMULATION TIME
C      LFT,LLT --- DO-LOOP RANGE FOR STRR
C
C      DIMENSION IDWELD (*), STRR (6, *), FAIL (*), CM (*), FIBL (5, *)
C
C      RETURN
C      END

```

