LS-DYNA
KEYWORD USER'S MANUAL

VOLUME II
(Material Models, References and Appendices)

March 2001
Version 960

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

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

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

TABLE OF CONTENTS

TABLE OF CONTENTS

TABLE OF CONTENTS

*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.**

	***MAT_ADD_EROSION**
	***MAT_NONLOCAL**
TYPE 1:	***MAT_ELASTIC_{*OPTION*} [0,1H,1B,1I,1T,2,3]**
TYPE 2:	***MAT_*OPTION*TROPIC_ELASTIC [0,2,3]**
TYPE 3:	***MAT_PLASTIC_KINEMATIC [0,1H,1I,1T,2,3]**
TYPE 4:	***MAT_ELASTIC_PLASTIC_THERMAL [0,1H,2,3]**
TYPE 5:	***MAT_SOIL_AND_FOAM [0]**
TYPE 6:	***MAT_VISCOELASTIC [0,1H]**
TYPE 7:	***MAT_BLATZ-KO_RUBBER [0,2]**
TYPE 8:	***MAT_HIGH_EXPLOSIVE_BURN [0]**
TYPE 9:	***MAT_NULL [0,1,2]**
TYPE 10:	***MAT_ELASTIC_PLASTIC_HYDRO_{*OPTION*} [0]**
TYPE 11:	***MAT_STEINBERG [0]**
TYPE 11:	***MAT_STEINBERG_LUND [0]**
TYPE 12:	***MAT_ISOTROPIC_ELASTIC_PLASTIC [0,2,3]**
TYPE 13:	***MAT_ISOTROPIC_ELASTIC_FAILURE [0]**
TYPE 14:	***MAT_SOIL_AND_FOAM_FAILURE [0]**
TYPE 15:	***MAT_JOHNSON_COOK [0,2,3]**
TYPE 16:	***MAT_PSEUDO_TENSOR [0]**
TYPE 17:	***MAT_ORIENTED_CRACK [0]**
TYPE 18:	***MAT_POWER_LAW_PLASTICITY [0,1H,2,3]**
TYPE 19:	***MAT_STRAIN_RATE_DEPENDENT_PLASTICITY [0,2,3]**

TYPE 20: *MAT_RIGID [0,1H,1B,1T,2,3]

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]

TYPE 25: *MAT_GEOLOGIC_CAP_MODEL [0]

TYPE 26: *MAT_HONEYCOMB [0]

TYPE 27: *MAT_MOONEY-RIVLIN_RUBBER [0,2]

TYPE 28: *MAT_RESULTANT_PLASTICITY [1B,2]

TYPE 29: *MAT_FORCE_LIMITED [1B]

TYPE 30: *MAT_SHAPE_MEMORY [0]

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]

TYPE 36: *MAT_3-PARAMETER_BARLAT [2]

TYPE 37: *MAT_TRANSVERSELY_ANISOTROPIC_ELASTIC_PLASTIC [2,3]

TYPE 38: *MAT_BLATZ-KO_FOAM [0,2]

TYPE 39: *MAT_FLD_TRANSVERSELY_ANISOTROPIC [2,3]

TYPE 40: *MAT_NONLINEAR_ORTHOTROPIC [2]

TYPE 41-50: *MAT_USER_DEFINED_MATERIAL_MODELS

TYPE 51: *MAT_BAMMAN [0,2,3]

TYPE 52: *MAT_BAMMAN_DAMAGE [0,2,3]

TYPE 53: *MAT_CLOSED_CELL_FOAM [0]

TYPE 54-55: *MAT_ENHANCED_COMPOSITE_DAMAGE [2]

TYPE 57: *MAT_LOW_DENSITY_FOAM [0]

TYPE 58: *MAT_LAMINATED_COMPOSITE_FABRIC [2]

TYPE 59: *MAT_COMPOSITE_FAILURE_*OPTION*_MODEL [0,2]

TYPE 60: *MAT_ELASTIC_WITH_VISCOSITY [0,2]

TYPE 61: *MAT_KELVIN-MAXWELL_VISCOELASTIC [0]

TYPE 62: *MAT_VISCOUS_FOAM [0]

TYPE 63: *MAT_CRUSHABLE_FOAM [0]

TYPE 64: *MAT_RATE_SENSITIVE_POWERLAW_PLASTICITY [0,2,3]

TYPE 65: *MAT_MODIFIED_ZERILLI_ARMSTRONG [0,2]

TYPE 72: *MAT_CONCRETE_DAMAGE [0]

TYPE 73: *MAT_LOW_DENSITY_VISCOUS_FOAM [0]

TYPE 75: *MAT_BILKHU/DUBOIS_FOAM [0]

TYPE 76: *MAT_GENERAL_VISCOELASTIC [0,2]

TYPE 77: *MAT_HYPERELASTIC_RUBBER [0]

TYPE 77: *MAT_OGDEN_RUBBER [0]

TYPE 78: *MAT_SOIL_CONCRETE [0]

TYPE 79: *MAT_HYSTERETIC_SOIL [0]

TYPE 80: *MAT_RAMBERG-OSGOOD [0]

TYPE 81: *MAT_PLASTICITY_WITH_DAMAGE [2,3]

TYPE 83: *MAT_FU_CHANG_FOAM [0]

TYPE 84: *MAT_WINFRITH_CONCRETE [0]

TYPE 84: *MAT_WINFRITH_CONCRETE_REINFORCEMENT [0]

TYPE 86: *MAT_ORTHOTROPIC_VISCOELASTIC [2]

TYPE 87: *MAT_CELLULAR_RUBBER [0]

TYPE 88: *MAT_MTS [0,2]

TYPE 89: *MAT_PLASTICITY_POLYMER [2]

TYPE 90: *MAT_ACOUSTIC [0]

TYPE 91: *MAT_SOFT_TISSUE_{*OPTION*} [0,2]

TYPE 96: *MAT_BRITTLE_DAMAGE [0]

TYPE 98: *MAT_SIMPLIFIED_JOHNSON_COOK [0,1H,1B,1T,2,3]

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]

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]

TYPE 114: *MAT_LAYERED_LINEAR_PLASTICITY [2,3]

TYPE 115: *MAT_UNIFIED_CREEP [0]

TYPE 116: *MAT_COMPOSITE_LAYUP [2]

TYPE 117: *MAT_COMPOSITE_MATRIX [2]

TYPE 118: *MAT_COMPOSITE_DIRECT [2]

TYPE 120: *MAT_GURSON [2]

TYPE 123: *MAT_MODIFIED_PIECEWISE_LINEAR_PLASTICITY [2,3]

TYPE 124: *MAT_PLASTICITY_COMPRESSION_TENSION [0]

TYPE 126: *MAT_MODIFIED_HONEYCOMB [0]

TYPE 127: *MAT_ARRUDA_BOYCE_RUBBER [0]

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 161: *MAT_COMPOSITE_MSC [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]

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]

For the discrete springs and dampers thirteen material types are available. In the strucutred input separate type numbers are assigned to this element class.

TYPE 1: *MAT_SPRING_ELASTIC

TYPE 2: *MAT_DAMPER_VISCOUS

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 strucutred 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 addition line is read for each section in 80a format which can be used to describe the material. At present LS-DYNA does make use of the title. Inclusion of titles gives greater clarity to input decks.

Material number	Material Title		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					

Material number	Material Title		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
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	Gn

Material number	Material Title		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	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				

Material number	Material Title		Beams	Thin Shells	Thick Shells	Strain-Rate Effects	Failure	Equation-of-State	Thermal Effects	Notes: Gn General Cm Composites Cr Ceramics Fl Fluids Fm Foam Gl Glass Hy Hydro-dyn Mt Metal Pl Plastic Rb Rubber Sl Soil/Conc
71	Cable Discrete Beam (Elastic)		Y							
72	Concrete Damage	Y				Y	Y	Y		Sl
73	Low Density Viscous Foam	Y				Y	Y			Fm
74										
75	Bilkhu/Dubois Foam (Isotropic)	Y				Y				Fm
76	General Viscoelastic (Maxwell Model)	Y				Y				Rb
77	Hyperelastic and Ogden Rubber	Y								Rb
78	Soil Concrete	Y					Y			Sl
79	Hysteretic Soil (Elasto-Perfectly Plastic)	Y					Y			Sl
80										
81	Plasticity with Damage (Elasto-Plastic)			Y	Y	Y	Y			Mt, Pl
82										
83	Fu Chang Foam	Y				Y	Y			Fm
84	Winfrith Concrete	Y								
84	Winfrith Concrete Reinforcement	Y								
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		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
96	Brittle Damage	Y				Y	Y			
98	Simplified Johnson Cook	Y	Y	Y	Y					
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	
110	Johnson Holmquist Ceramics	Y								
111	Johnson Holmquist Concrete	Y								
112	Finite Elastic Strain Plasticity	Y								
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						
123	Modified Piecewise Linear Plasticity			Y	Y					
124	Plasticity Compression Tension	Y								
126	Modified Honeycomb	Y								

Material number	Material Title		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
127	Arruda Boyce Rubber	Y								
128	Heart Tissue	Y								
129	Lung Tissue	Y								
130	Special Orthotropic			Y						
139	Modified Force Limited		Y							
150	CFD									
161	Composite MSC	Y								
191	Seismic Beam		Y							
192	Soil Brick	Y								
193	Drucker Prager	Y								
194	RC Shear Wall			Y						
195	Concrete Beam		Y							
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							

Material number	Material Title	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
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 consitutive 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	1	2	3	4	5	6	7	8
Variable	PFAIL	SIGP1	SIGVM	EPSP1	EPSSH	SIGTH	IMPULSE	
Type	F	F	F	F	F	F	F	
Default	none	none	none	none	none	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, $\sigma_{\max}$.
SIGVM	Equivalent stress at failure, $\bar{\sigma}_{\max}$.
EPSP1	Principal strain at failure, $\varepsilon_{\max}$
EPSSH	Shear strain at failure, $\gamma_{\max}$.
SIGTH	Threshold stress, σ_0.
IMPULSE	Stress impulse for failure, $\mathrm{K_f}$.

The criteria for failure are:

1. $P \le P_{\min}$ where P is the pressure (positive in compression), and $P_{\min}$ is the pressure at failure.
2. $\sigma_1 \ge \sigma_{\max}$, where σ_1 is the maximum principal stress, and $\sigma_{\max}$.is the principal stress at failure.
3. $\sqrt{\frac{3}{2}\sigma'_{ij}\sigma'_{ij}} \ge \bar{\sigma}_{\max}$, where σ'_{ij} are the deviatoric stress components, and $\bar{\sigma}_{\max}$ is the equivalent stress at failure.
4. $\varepsilon_1 \ge \varepsilon_{\max}$, where ε_1 is the maximum principal strain, and $\varepsilon_{\max}$ is the principal strain at failure.
5. $\gamma_1 \ge \gamma_{\max}$, where γ_1 is the shear strain, and $\gamma_{\max}$ is the shear strain at failure.
6. The Tuler-Butcher criterion,

$$\int_0^t [\max(0,\sigma_1 - \sigma_0)]^2 \,\mathrm{dt} \ge \mathrm{K_f},$$

where σ_1 is the maximum principal stress, σ_0 is a specified threshold stress, $\sigma_1 \ge \sigma_0 \ge 0$, and $\mathrm{K_f}$ 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	none	none	none	none	none	none	none

Define one card for each symmetry plane. Up to six symmetry planes can be defined. The next "*" card terminates this input.

Cards 3,...

Variable	XC1	YC1	ZC1	XC2	YC2	ZC2		
Type	F	F	F	F	F	F		
Default	none	none	none	none	none	none		

VARIABLE	DESCRIPTION
IDNL	Nonlocal material input ID.
PID	Part ID for nonlocal material.
P	Exponent of weighting function. A typical value might be 8 depending somewhat on the choice of L. See equations below.
Q	Exponent of weighting function. A typical value might be 2. See equations below.
L	Characteristic length. This length should span a few elements. See 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 $\{e_i\}_{i=1,\ldots,N_r}$ the list of elements included in Ω_r, then

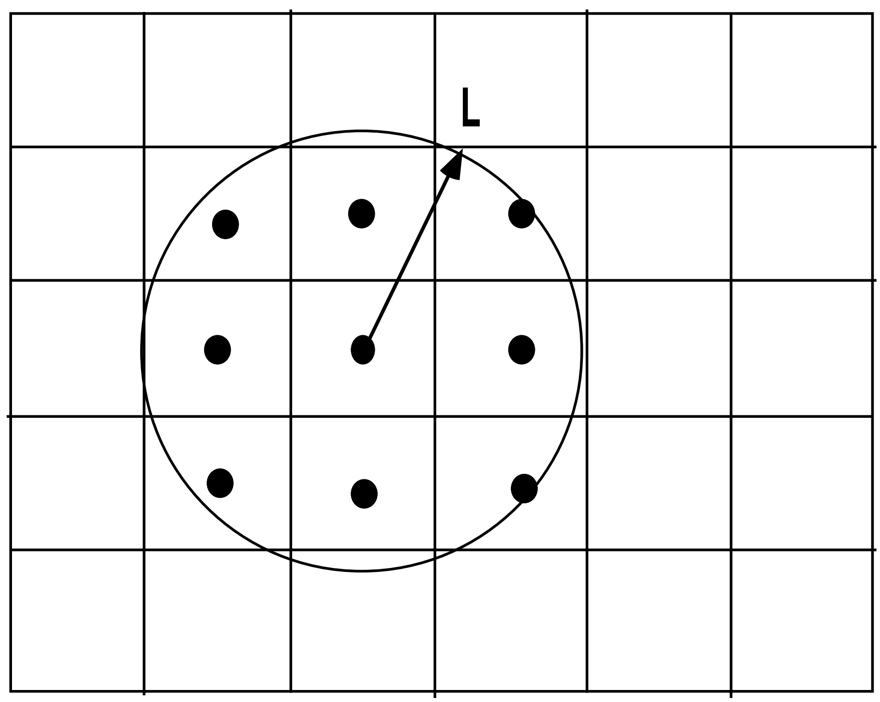
$$\dot{f}_r = \dot{f}(x_r) = \frac{1}{W_r}\int_{\Omega_r} \dot{f}_{local}\, w(x_r - y)\,dy \approx \frac{1}{W_r}\sum_{i=1}^{N_r} \dot{f}_{local}^{i}\, w_{ri} V_i$$

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 $\dot{f}^i_{local}$, V_i and y_i are respectively the local rate of increase of damage, the volume and the center of element e_i.



*MAT_ELASTIC_{*OPTION*}

This is Material Type 1. This is an isotropic elastic material and is available for beam, shell, and solid elements in LS-DYNA. A specialization of this material allows the modeling of fluids.

Options include:

<BLANK>

FLUID

such that the keyword cards appear:

***MAT_ELASTIC**

***MAT_ELASTIC_FLUID**

The fluid option is valid for solid elements only.

Define the following card for all options:

Card Format

	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	DA	DB	K	
Type	I	F	F	F	F	F	F	
Default	none	none	none	none	0.0	0.0	0.0	

Define the following extra card for the FLUID option:

Card Format

	1	2	3	4	5	6	7	8
Variable	VC	CP						
Type	F	F						
Default	none	1.0E+20						

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.
DA	Axial damping factor (used for Belytschko-Schwer beam, type 1, only).
DB	Bending damping factor (used for Belytschko-Schwer beam, type 1, only).
K	Bulk Modulus (define for fluid option only).
VC	Tensor viscosity coefficient, values between .1 and .5 should be okay.
CP	Cavitation pressure (default = 1.0e+20).

Remarks:

The axial and bending damping factors are used to damp down numerical noise. The update of the force resultants, F_i, and moment resultants, M_i, includes the damping factors:

$$F_i^{n+1} = F_i^n + \left(1 + \frac{DA}{\Delta t}\right)\Delta F_i^{n+\frac{1}{2}}$$

$$M_i^{n+1} = M_i^n + \left(1 + \frac{DB}{\Delta t}\right)\Delta M_i^{n+\frac{1}{2}}$$

For the fluid option the bulk modulus (K) has to be defined as Young's modulus, and 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:

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

$$\dot{p} = -K\ \dot{\varepsilon}_{ii}$$

and the shear modulus is set to zero. A tensor viscosity is used which acts only the deviatoric stresses, S_{ij}^{n+1} , given in terms of the damping coefficient as:

$$S_{ij}^{n+1} = VC \cdot \Delta L \cdot a \cdot \rho \dot{\varepsilon}_{ij}^{'}$$

where p, is a characteristic element length, a is the fluid bulk sound speed, ρ is the fluid density, and $\dot{\varepsilon}_{ij}^{'}$ is the deviatoric strain rate.

***MAT_*OPTION* TROPIC_ELASTIC**

This is Material Type 2. This material is valid for modeling the elastic-orthotropic behavior of solids, shells, and thick shells. An anisotropic option is available for solid elements.

Options include:

ORTHO

ANISO

such that the keyword cards appear:

***MAT_ORTHOTROPIC_ELASTIC** (4 cards follow)

***MAT_ANISOTROPIC_ELASTIC** (5 cards follow)

Card Format of Cards 1 and 2 for the ORTHO option.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	EA	EB	EC	PRBA	PRCA	PRCB
Type	I	F	F	F	F	F	F	F

Card 2								
Variable	GAB	GBC	GCA	AOPT				
Type	F	F	F	F				

Card Format of Cards 1, 2, and 3 for the ANISO option.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	C11	C12	C22	C13	C23	C33
Type	I	F	F	F	F	F	F	F

Card 2

Variable	C14	C24	C34	C44	C15	C25	C35	C45
Type	F	F	F	F	F	F	F	F

Card 3

Variable	C55	C16	C26	C36	C46	C56	C66	AOPT
Type	F	F	F	F	F	F	F	F

Card Format of Cards 3/4 and 4/5 for the ORTHO/ANISO options.

Card 3/4

Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card 4/5

Variable	V1	V2	V3	D1	D2	D3	BETA	REF
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
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 *a* material direction
C12	The 1,2 term in the 6×6 anisotropic constitutive matrix. Note that 2 corresponds to the *b* material direction
.	.
.	.
.	.
C66	The 6,6 term in the 6×6 anisotropic constitutive matrix.

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.
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:

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

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

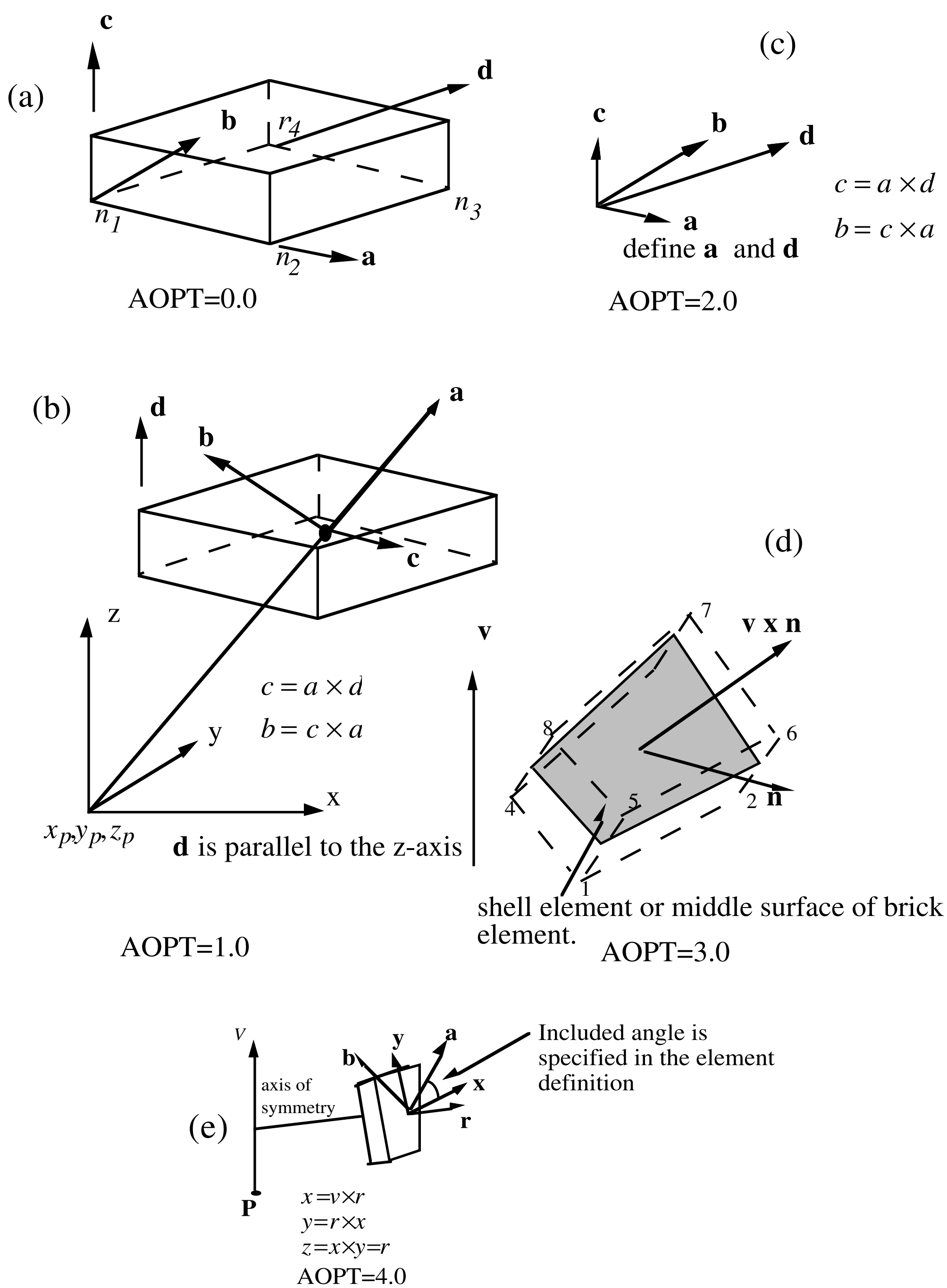


Figure 20.1. Options for determining principal material axes: (a) AOPT = 0.0, (b) AOPT = 1.0, (c) AOPT = 2.0,. Note that $\underset{\sim}{c} = \underset{\sim}{a} \times \underset{\sim}{d}$ and that $\underset{\sim}{b} = \underset{\sim}{c} \times \underset{\sim}{a}$, (d) AOPT = 3.0, and (e) AOPT=4.0 for brick elements.

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

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

*MAT_PLASTIC_KINEMATIC

This is Material Type 3. This model is suited to model isotropic and kinematic hardening plasticity with the option of including rate effects. It is a very cost effective model and is available for beam (Hughes-Liu), shell, and solid elements.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	SIGY	ETAN	BETA	
Type	I	F	F	F	F	F	F	
Default	none	none	none	none	none	0.0	0.0	

Card 2								
Variable	SRC	SRP	FS	VP				
Type	F	F	F	F				
Default	not used	not used	not used	0.0				

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
E	Young’s modulus.
PR	Poisson’s ratio.
SIGY	Yield stress.
ETAN	Tangent modulus, see Figure 20.2.
BETA	Hardening parameter, $0 < \beta' < 1$. See comments below.

VARIABLE	DESCRIPTION
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

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

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

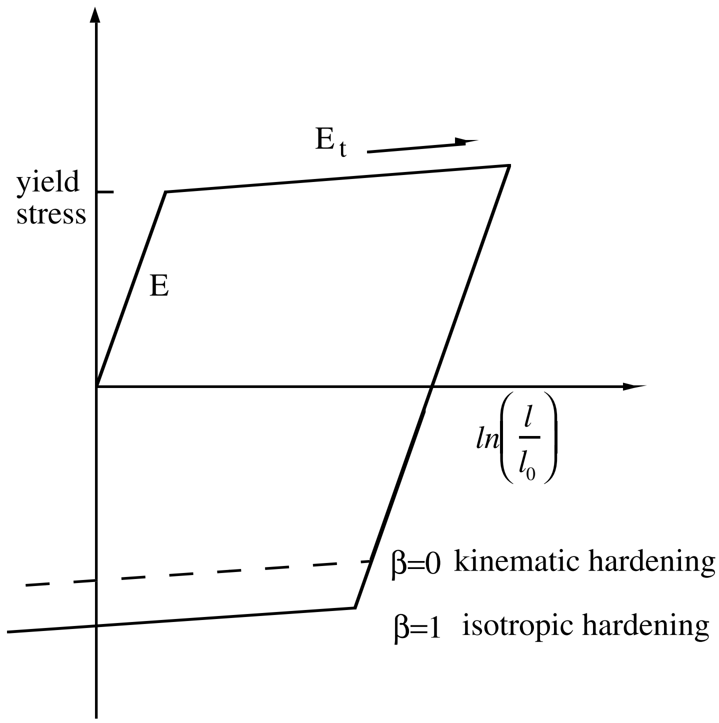


Figure 20.2. Elastic-plastic behavior with kinematic and isotropic hardening where l_0 and l are undeformed and deformed lengths of uniaxial tension specimen. E_t is the slope of the bilinear stress strain curve.

*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 neccessary 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.
ALPHAI	Corresponding coefficients of thermal expansion.
SIGYI	Corresponding yield stresses.
ETANI	Corresponding plastic hardening moduli.

Remarks:

At least two temperatures and their corresponding material properties must be defined. The analysis will be terminated if a material temperature falls outside the range defined in the input. If a thermoelastic material is considered, do not define SIGY and ETAN. The coefficient of thermal expansion is defined as the instantaneous value. Thus, the thermal strain rate becomes:

$$\dot{\varepsilon}_{ij}^{T} = \alpha \dot{T} \delta_{ij}$$

*MAT_SOIL_AND_FOAM

This is Material Type 5. This is a very simple model and works in some ways like a fluid. It should be used only in situations when soils and foams are confined within a structure or when geometric boundaries are present.

Card 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 geometriy is defined by the keyword:*INITIAL_FOAM_REFERENCE_ GEOMETRY. This option doe 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 competely 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].

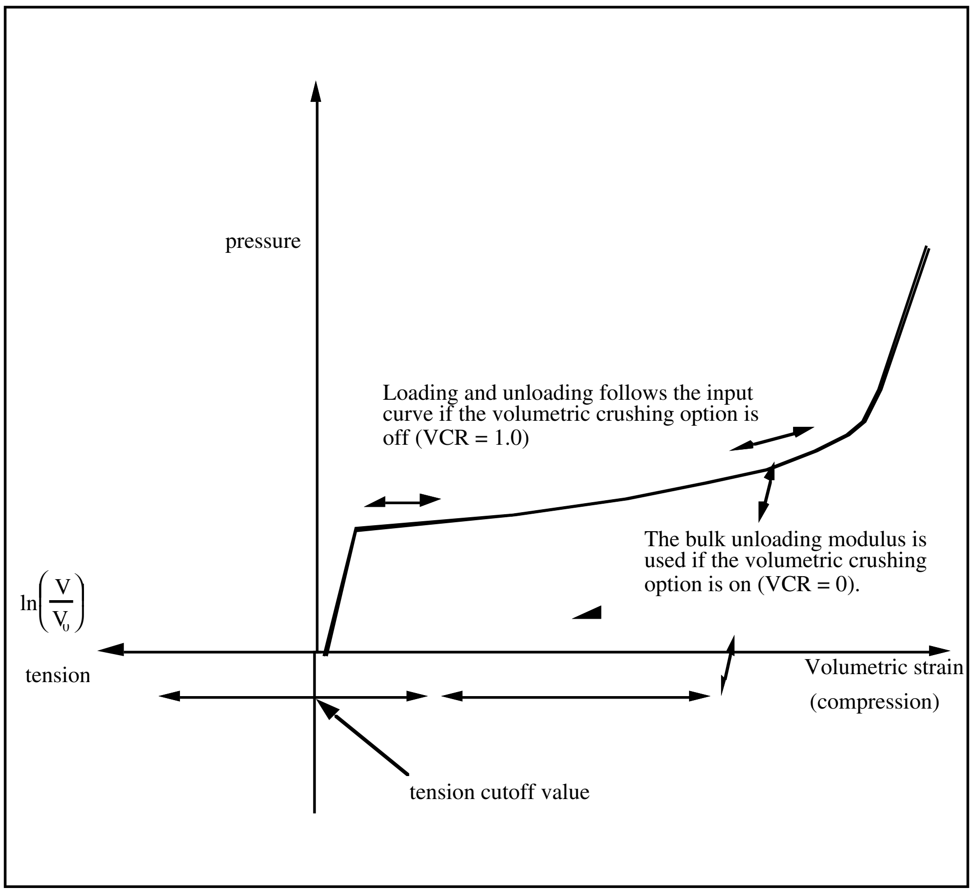


Figure 20.3. Pressure versus volumetric strain curve for soil and crushable foam model. The volumetric st in is given by the natural logarithm of the relative volume, V.

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

$$J_2 = \frac{1}{2} s_{ij} s_{ij} \quad ,$$

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

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

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

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

There is no strain hardening on this surface.

To eliminate the pressure dependence of the yield strength, set:

$$a_1 = a_2 = 0 \qquad a_0 = \frac{1}{3} \sigma_y^2 .$$

This approach is useful when a von Mises type elastic-plastic model is desired for use with the tabulated volumetric data.

*MAT_VISCOELASTIC

This is Material Type 6. This model allows the modeling of viscoelastic behavior for beams (Hughes-Liu), shells, and solids. Also see *MAT_GENERAL_VISCOELASTIC for a more general formulation.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	BULK	G0	GI	BETA		
Type	I	F	F	F	F	F		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density
BULK	Elastic bulk modulus.
G0	Short-time shear modulus, see equations below.
GI	Long-time (infinite) shear modulus, G_∞.
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 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 second Piola-Kirchhoff stress is computed as

$$S_{ij} = G\left[\frac{1}{V} C_{ij} - V^{\left(\frac{1}{1-2\upsilon}\right)} \delta_{ij}\right]$$

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

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
D	Detonation velocity.
PCJ	Chapman-Jouget pressure.
BETA	Beta burn flag, BETA (see comments below): EQ.0.0: beta + programmed burn, EQ.1.0: beta burn only, EQ.2.0: programmed burn only.
K	Bulk modulus (BETA=2.0 only).
G	Shear modulus (BETA=2.0 only).
SIGY	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

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

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

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

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

$$F = F_2$$

and F_1 is not computed.

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

As an option, the high explosive material can behave as an elastic perfectly-plastic solid prior to detonation. In this case we update the stress tensor, to an elastic trial stress, $^*s_{ij}^{n+1}$,

$$^*s_{ij}^{n+1} = s_{ij}^{n} + s_{ip}\Omega_{pj} + s_{jp}\Omega_{pi} + 2G\dot{\varepsilon}'_{ij}\ dt$$

where G is the shear modulus, and $\dot{\varepsilon}'_{ij}$ is the deviatoric strain rate. The von Mises yield condition is given by:

$$\phi = J_2 - \frac{\sigma_y^2}{3}$$

where the second stress invariant, J_2, is defined in terms of the deviatoric stress components as

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

and the yield stress is σ_y. If yielding has occurred, i.e., $\phi > 0$, the deviatoric trial stress is scaled to obtain the final deviatoric stress at time n+1:

$$s_{ij}^{n+1} = \frac{\sigma_y}{\sqrt{3J_2}} * s_{ij}^{n+1}$$

If $\phi \le 0$, then

$$s_{ij}^{n+1} = {}^*s_{ij}^{n+1}$$

Before detonation pressure is given by the expression

$$p^{n+1} = K\left(\frac{1}{V^{n+1}} - 1\right)$$

where K is the bulk modulus. Once the explosive material detonates:

$$s_{ij}^{n+1} = 0$$

and the material behaves like a gas.

*MAT_NULL

This is Material Type 9. This material allows equations of state to be considered without computing deviatoric stresses. Optionally, a viscosity can be defined. Also, erosion in tension and compression is possible.

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

Beams and shells that use this material type are completely bypassed in the element processing; however, the mass of the null shell elements is computed and added to the nodal points which define the connectivity, 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	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:

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

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

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

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.

*MAT_ELASTIC_PLASTIC_HYDRO_{*OPTION*}

Available options are:

<BLANK>
SPALL

This is Material Type 10. This material allows the modeling of an elastic-plastic hydrodynamic material.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	G	SIGY	EH	PC	FS	
Type	I	F	F	F	F	F	F	
Default	none	none	none	0.0	0.0	$-\infty$	0.0	

Define this card if and only if the SPALL option is specified.

Optional								
Variable	A1	A2	SPALL					
Type	F	F	F					

Card 2								
Variable	EPS1	EPS2	EPS3	EPS4	EPS5	EPS6	EPS7	EPS8
Type	F	F	F	F	F	F	F	F

Card 3	1	2	3	4	5	6	7	8
Variable	EPS9	EPS10	EPS11	EPS12	EPS13	EPS14	EPS15	EPS16
Type	F	F	F	F	F	F	F	F

Card 4								
Variable	ES1	ES2	ES3	ES4	ES5	ES6	ES7	ES8
Type	F	F	F	F	F	F	F	F

Card 5								
Variable	ES9	ES10	ES11	ES12	ES13	ES14	ES15	ES16
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
G	Shear modulus.
SIGY	Yield stress, see comment below.
EH	Plastic hardening modulus, see definition below.
PC	Pressure cutoff (≤ 0.0). If zero, a cutoff of -∞ 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: $p \geqq p_{min}$, EQ.2.0: if $\sigma_{max} \geq -p$ min element spalls and tension, $p < 0$, is never allowed, EQ.3.0: $p < -p_{min}$ element spalls and tension, $p < 0$, is never allowed.
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

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

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

$$E_h = \frac{E_t \, E}{E - E_t} \quad .$$

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:

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

where t denotes time and D_{ij}^p is the plastic component of the rate of deformation tensor. In this case the plastic hardening modulus on Card 1 is ignored and the yield stress is given as

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

where the value for $f\left(\bar{\varepsilon}^p\right)$ is found by interpolation from the data curve.

A choice of three spall models is offered to represent material splitting, cracking, and failure under tensile loads. The pressure limit model, SPALL=1, limits the hydrostatic tension to the specified value, p_{cut}. If pressures more tensile than this limit are calculated, the pressure is reset to p_{cut}. This option is not strictly a spall model, since the deviatoric stresses are unaffected by the pressure reaching the tensile cutoff, and the pressure cutoff value, p_{cut}, remains unchanged throughout the analysis. The maximum principal stress spall model, SPALL=2, detects spall if the maximum principal stress, σ_{max}, exceeds the limiting value $-p_{cut}$. Note that the negative sign is required because p_{cut} is 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.

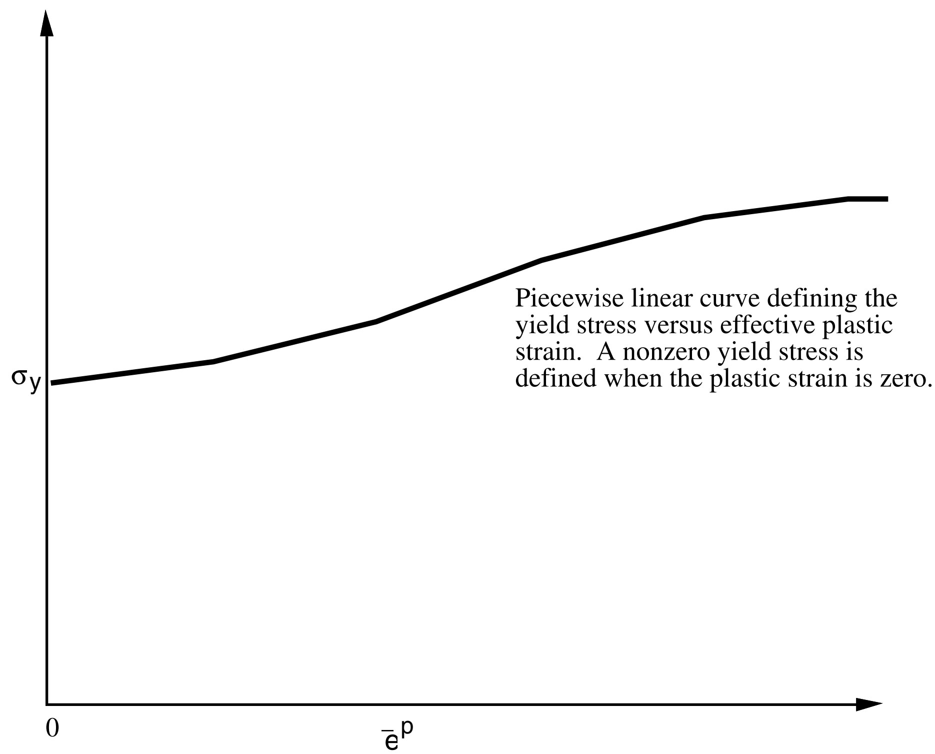


Figure 20.4. Effective stress versus effective plastic strain curve.

*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

VARIABLE	DESCRIPTION
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: $p \geqq p_{cut}$, 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:

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

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

$$E_c(x) = \int_0^x pdx - \frac{900\ R' \exp(ax)}{(1-x)^{2\ (\gamma_0 - a - 1/2)}},$$

$$x = 1 - V,$$

and E_m is the melting energy:

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

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

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

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

In the above equation R′ is defined by

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

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

The yield strength σ_y is given by:

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

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

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

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

If the coefficients EC0,...,EC9 are not defined above, LS-DYNA will fit the cold compression energy to a ten term polynomial expansion either as a function of μ or η depending on the input variable, FLAG, as:

$$E_c\left(\eta^i\right) = \sum_{i=0}^{9} EC_i \eta^i$$

$$E_c\left(\mu^i\right) = \sum_{i=0}^{9} EC_i \mu^i$$

where EC_i is the ith coefficient and:

$$\eta = \frac{\rho}{\rho_o}$$

$$\mu = \frac{\rho}{\rho_o} - 1$$

A linear least squares method is used to perform the fit.

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

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

*MAT_STEINBERG_LUND

This is Material Type 11. This material is a modification of the Steinberg model above to include the rate model of Steinberg and Lund [1989]. An equation of state determines the pressure.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	G0	SIGO	BETA	N	GAMA	SIGM
Type	I	F	F	F	F	F	F	F

Card 2

Variable	B	BP	H	F	A	TMO	GAMO	SA
Type	F	F	F	F	F	F	F	F

Card 3

Variable	PC	SPALL	RP	FLAG	MMN	MMX	ECO	EC1
Type	F	F	F	F	F	F	F	F

Card 4

Variable	EC2	EC3	EC4	EC5	EC6	EC7	EC8	EC9
Type	F	F	F	F	F	F	F	F

Card 5

Variable	UK	C1	C2	YP	YA	YM		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
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)

VARIABLE	DESCRIPTION
SPALL	Spall type: EQ. 0.0: default set to "2.0", EQ. 1.0: $p \geqq p_{min}$, EQ. 2.0: if $\sigma_{max} \geq -p_{min}$ element spalls and tension, $p < 0$, is never allowed, EQ. 3.0: $p < -p_{min}$ element spalls and tension, $p < 0$, is never allowed.
RP	R'. If $R' \neq 0.0$, 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:

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

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

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

and the second is on the thermal part:

$$Y_T \leq Y_P$$

*MAT_ISOTROPIC_ELASTIC_PLASTIC

This is Material Type 12. This is a very low cost isotropic plasticity model for three dimensional solids. In the plane stress implementation for shell elements, a one-step radial return approach is used to scale the Cauchy stress tensor to if the state of stress exceeds the yield surface. This approach to plasticity leads to inaccurate shell thickness updates and stresses after yielding. This is the only model in LS-DYNA for plane stress that does not default to an iterative approach.

Card 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

$$\dot{p} = -K\ \dot{\varepsilon}_{ii}$$

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

*MAT_ISOTROPIC_ELASTIC_FAILURE

This is Material Type 13. This is a non-iterative plasticity with simple plastic strain failure model.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	G	SIGY	ETAN	BULK		
Type	I	F	F	F	F	F		
Default	none	none	none	none	0.0	none		

Card 2

Variable	EPF	PRF	REM	TREM				
Type	F	F	F	F				
Default	none	0.0	0.0	0.0				

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
G	Shear modulus.
SIGY	Yield stress.
ETAN	Plastic hardening modulus.
BULK	Bulk modulus.
EPF	Plastic failure strain.
PRF	Failure pressure (≤ 0.0).

VARIABLE	DESCRIPTION
REM	Element erosion option: EQ.0.0: failed element eroded after failure, NE.0.0: element is kept, no removal except by Δt below.
TREM	Δt for element removal: EQ.0.0: Δt is not considered (default), GT.0.0: element eroded if element time step size falls below Δt.

Remarks:

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

The element erosion option based on Δt must be used cautiously with the contact options. Nodes to surface contact is recommended with all nodes of the eroded brick elements included in the node list. As the elements are eroded the mass remains and continues to interact with the master surface.

*MAT_SOIL_AND_FOAM_FAILURE

This is Material Type 14. The input for this model is the same as for *MATERIAL_SOIL_AND_FOAM (Type 5); however, when the pressure reaches the failure pressure, the element loses its ability to carry tension. It should be used only in situations when soils and foams are confined within a structure or when geometric boundaries are present.

*MAT_JOHNSON_COOK

This is Material Type 15. The Johnson/Cook strain and temperature sensitive plasticity is sometimes used for problems where the strain rates vary over a large range and adiabatic temperature increases due to plastic heating cause material softening. When used with solid elements this model requires an equation-of-state. If thermal effects and damage are unimportant, the much less expensive *MAT_ SIMPLIFIED_JOHNSON_COOK model is recommended. The simplified model can be used with beam elements.

Card 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							

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density
G	Shear modulus
E	Young's Modulus (shell elements only)
PR	Poisson's ratio (shell elements only)
DTF	Minimum time step size for automatic element deletion (shell elements)
VP	Formulation for rate effects: EQ.0.0: Scale yield stress (default), EQ.1.0: Viscoplastic formulation.
A	See equations below.
B	See equations below.
N	See equations below.
C	See equations below.
M	See equations below.
TM	Melt temperature
TR	Room temperature
EPSO	Effective plastic strain rate. This value depends on the time units. Typically, nput 1 for units of seconds, 0.001 for units of milliseconds, 0.000001 for microseconds, etc.
CP	Specific heat

VARIABLE	DESCRIPTION
PC	Failure stress or pressure cutoff ($p_{min} < 0.0$)
SPALL	Spall type: EQ. 0.0: default set to "2.0", EQ. 1.0: $p \geq p_{min}$, EQ. 2.0: if $\sigma_{max} \geq -p_{min}$ element spalls and tension, p < 0, is never allowed, EQ. 3.0: $p < -p_{min}$ element spalls and tension, p < 0, is never allowed.
IT	Plastic strain iteration 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

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

where

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

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

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

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

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

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

The strain at fracture is given by

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

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

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

Fracture occurs when the damage parameter

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

reaches the value of 1.

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 implusive 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								

VARIABLE	DESCRIPTION
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 x 106 psi).

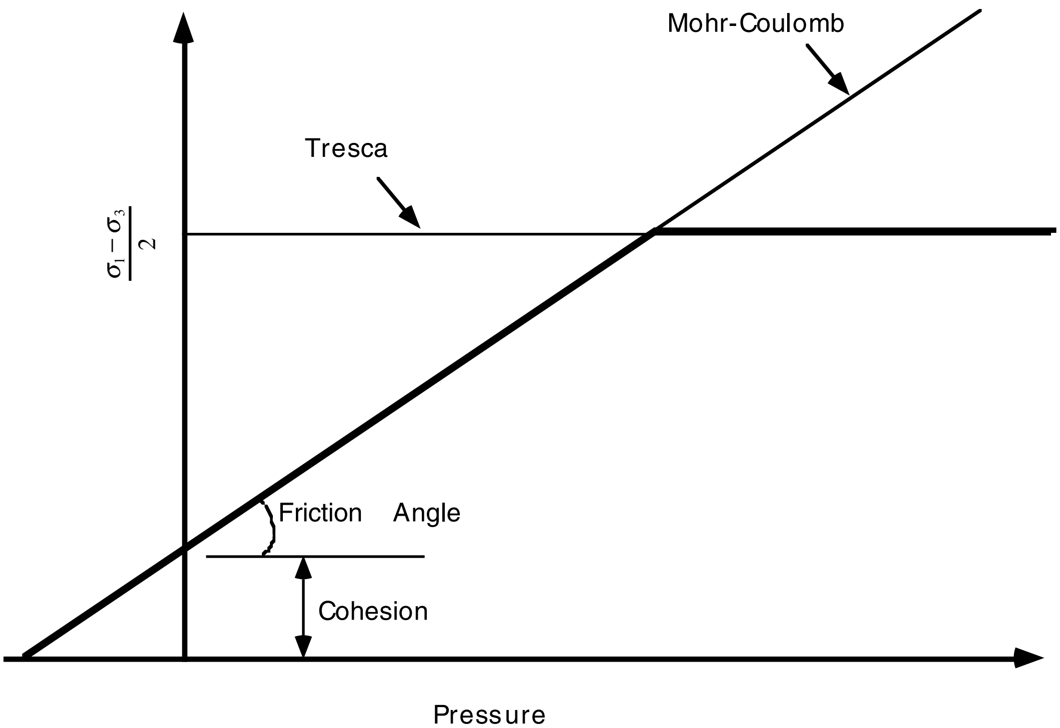


Figure 20.5. Mohr-Coulomb surface with a Tresca limit.

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 5 and 6, and the corresponding values of yield stress should be specified on cards 7 and 8. The parameters relating to reinforcement properties, initial yield stress, and tangent modulus are not used in this response mode, and should be set to zero.

Simple tensile failure

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

Response Mode II. Two Curve Model with Damage and Failure

This approach uses two yield versus pressure curves of the form

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

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

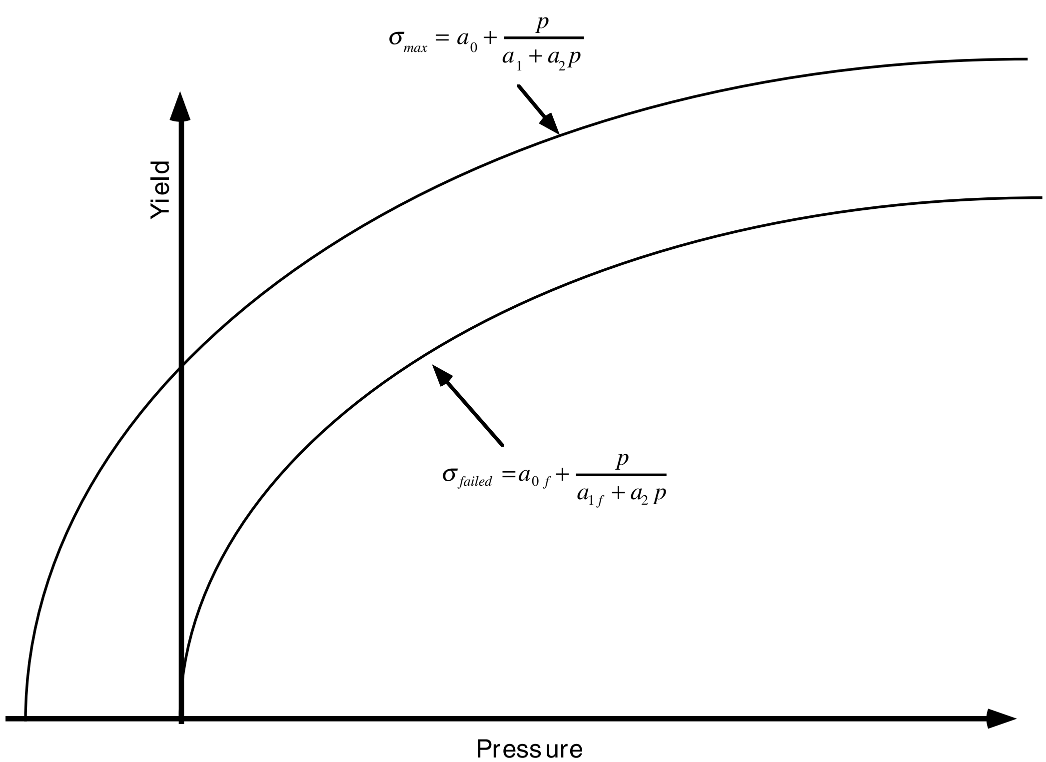


Figure 20.6. Two-curve concrete model with damage and failure.

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

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

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.

Model II.C: Tensile failure plus damage scaling

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

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

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

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

and then apply any tensile failure criteria.

Mode II Concrete Model Options

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

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

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

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

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

$$a_{0f} = 0$$

$$a_{1f} = 0.385$$

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

Mixture model

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

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

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

A Suggestion

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

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

and a damage table of:

Card 4:	0.0	8.62E-06	2.15E-05	3.14E-05	3.95E-04
	5.17E-04	6.38E-04	7.98E-04		
Card 5:	9.67E-04	1.41E-03	1.97E-03	2.59E-03	3.27E-03
	4.00E-03	4.79E-03	0.909		
Card 6:	0.309	0.543	0.840	0.975	1.000
	0.790	0.630	0.469		
Card 7:	0.383	0.247	0.173	0.136	0.114
	0.086	0.056	0.0		

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

*MAT_ORIENTED_CRACK

This is Material Type 17. This material may be used to model brittle materials which fail due to large tensile stresses.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	SIGY	ETAN	FS	PRF
Type	I	F	F	F	F	F	F	F
Default	none	none	none	none	none	0.0	none	0.0

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.
SIGY	Yield stress.
ETAN	Plastic hardening modulus.
FS	Fracture stress.
PRF	Failure or cutoff pressure(≤ 0.0).

Remarks:

This is an isotropic elastic-plastic material which includes a failure model with an oriented crack. The von Mises yield condition is given by:

$$\phi = J_2 - \frac{\sigma_y^2}{3}$$

where the second stress invariant, J_2, is defined in terms of the deviatoric stress components as

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

and the yield stress, σ_y, is a function of the effective plastic strain, ε^p_{eff}, and the plastic hardening modulus, E_p:

$$\sigma_y = \sigma_0 + E_p \varepsilon^p_{eff}$$

The effective plastic strain is defined as:

$$\varepsilon^p_{eff} = \int_0^t d\varepsilon^p_{eff}$$

where $$d\varepsilon^p_{eff} = \sqrt{\frac{2}{3} d\varepsilon^p_{ij} d\varepsilon^p_{ij}}$$

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						

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
E	Young’s modulus.
PR	Poisson’s ratio.
K	Strength coefficient.
N	Hardening exponent.
SRC	Strain rate parameter, C, if zero, rate effects are ignored.
SRP	Strain rate parameter, P, if zero, rate effects are ignored.

VARIABLE	DESCRIPTION
SIGY	Optional input parameter for defining the initial yield stress, σ_y. Generally, this parameter is not necessary and the strain to yield is calculated as described below. LT.0.02: $\varepsilon_{yp} = SIGY$ GE.0.02: See below.
VP	Formulation for rate effects: EQ.0.0: Scale yield stress (default), EQ.1.0: Viscoplastic formulation.

Remarks:

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

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

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

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

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

which gives the elastic strain at yield as:

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

If SIGY yield is nonzero and greater than 0.02 then:

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

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

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

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

*MAT_STRAIN_RATE_DEPENDENT_PLASTICITY

This is Material Type 19. A strain rate dependent material can be defined. For an alternative, see Material Type 24. Required is a curve for the yield stress versus the effective strain rate. Optionally, Young's modulus and the tangent modulus can also be defined versus the effective strain rate. Also, optional failure of the material can be defined either by defining a von Mises stress at failure as a function of the effective strain rate (valid for solids/shells/thick shells) or by defining a minimum time step size (only for shells).

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	VP			
Type	I	F	F	F	F			
Default	none	none	none	none	0.0			

Card 2								
Variable	LC1	ETAN	LC2	LC3	LC4	TDEL	RDEF	
Type	F	F	F	F	F	F	F	
Default	none	0.0	0.0	0.0	0.0	0.0	0.0	

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.

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	Plastic hardening modulus E_t
LC2	Load curve ID defining Young's modulus as a function of the effective strain rate (optional).
LC3	Load curve ID defining tangent modulus as a function of the effective strain rate (optional).
LC4	Load curve ID defining von Mises stress at failure as a function of the effective strain rate (optional).
TDEL	Minimum time step size for automatic element deletion. Use for shells only.
RDEF	Redefinition of failure curve: EQ.1.0: Effective plastic strain, EQ.2.0: Maximum principal stress.

Remarks:

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

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

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

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

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

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

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

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

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

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

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

*MAT_RIGID

This is Material 20. Parts made from this material are considered to belong to a rigid body (for each part ID). Also, the coupling of a rigid body with MADYMO and CAL3D can be defined via this material. Alternatively, a VDA surface can be attached as surface to model the geometry, e.g, for the tooling in metalforming applications. Also, global and local constraints on the mass center can be optionally defined. Optionally, a local consideration for output and user-defined airbag sensors can be chosen.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	N	COUPLE	M	ALIAS
Type	I	F	F	F	F	F	F	C
Default	none	none	none	none	0	0	0	blank

Card 2

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

Optional Card Format for output (Must be included but may be left blank).

Card 3

Variable	LCO or A1	A2	A3	V1	V2	V3		
Type	F	F	F	F	F	F		
Default	0	0	0	0	0	0		

VARIABLE	DESCRIPTION
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 **(not CAL3D)** coupling flag, n: EQ.0: use normal LS-DYNA rigid body updates, GT.0: the rigid body is coupled to MADYMO ellipsoid number n, LT.0: the rigid body is coupled to MADYMO plane number |n|.
COUPLE	Coupling option if applicable: EQ.-1: attach VDA surface in ALIAS (defined in the eighth field) and automatically generate a mesh for viewing the surface in LS-TAURUS. MADYMO3D/CAL3D coupling option: EQ.0: the undeformed geometry input to LS-DYNA corresponds to the local system for MADYMO/CAL3D. The finite element mesh is input, EQ.1: the undeformed geometry input to LS-DYNA corresponds to the global system for MADYMO/CAL3D, EQ.2: generate a mesh for the ellipsoids and planes internally in LS-DYNA3D.
M	MADYMO/CAL3D Coupling option flag: EQ.0: use normal LS-DYNA rigid body updates, EQ.m: this rigid body corresponds to MADYMO/CAL3D system number m. Rigid body updates are performed by MADYMO/CAL3D.
ALIAS	VDA surface alias name, see Appendix I.
CMO	Center of mass constraint option, CMO: EQ.+1.0: constraints applied in global directions,

VARIABLE	DESCRIPTION
	EQ. 0.0: no constraints, EQ. -1.0: constraints applied in local directions (SPC constraint).
CON1	First constraint parameter: If CMO=+1.0, then specify global translational constraint: EQ.0: no constraints, EQ.1: constrained x displacement, EQ.2: constrained y displacement, EQ.3: constrained z displacement, EQ.4: constrained x and y displacements, EQ.5: constrained y and z displacements, EQ.6: constrained z and x displacements, EQ.7: constrained x, y, and z displacements. If CM0=-1.0, then specify local coordinate system ID. See *DEFINE_ COORDINATE_*OPTION*: This coordinate system is fixed in time.
CON2	Second constraint parameter: If CMO=+1.0, then specify global rotational constraint: EQ.0: no constraints, EQ.1: constrained x rotation, EQ.2: constrained y rotation, EQ.3: constrained z rotation, EQ.4: constrained x and y rotations, EQ.5: constrained y and z rotations, EQ.6: constrained z and x rotations, EQ.7: constrained x, y, and z rotations. If CM0=-1.0, then specify local (SPC) constraint: EQ.000000 no constraint, EQ.100000 constrained x translation, EQ.010000 constrained y translation, EQ.001000 constrained z translation, EQ.000100 constrained x rotation, EQ.000010 constrained y rotation, EQ.000001 constrained z rotation. Any combination of local constraints can be achieved by adding the number 1 into the corresponding column.
LCO	Local coordinate system number for output. See *DEFINE_ COORDINATE_*OPTION*. *****Alternative method for specifying local system below.*****
A1-V3	Define two vectors **a** and **v**, fixed in the rigid body which are used for output and the user defined airbag sensor subroutines. The output parameters are in the directions **a**, **b**, and **c** where the latter are given by the cross products **c=a**×**v** and **b=c**×**a.** This input is optional.

Remarks:

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

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

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

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

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

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.

*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 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	Coordinates of point **p** for AOPT = 1.
A1,A2,A3	Components of vector **a** for AOPT = 2.

VARIABLE	DESCRIPTION
V1,V2,V3	Components of vector **v** for AOPT = 3.
D1,D2,D3	Components of vector **d** for AOPT = 2.
BETA	Material angle in degrees for AOPT = 3, may be overridden on the element card, see *ELEMENT_SHELL_BETA or *ELEMENT_SOLID_ORTHO.
REF	Use reference geometry to initialize the stress tensor. The reference 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:

In the implementation for three dimensional continua a total Lagrangian formulation is used. In this approach the material law that relates second Piola-Kirchhoff stress S to the Green-St. Venant strain E is

$$S = C \cdot E = T^t C_l T \cdot E$$

where T is the transformation matrix [Cook 1974].

$$T = \begin{bmatrix} l_1^2 & m_1^2 & n_1^2 & l_1 m_1 & m_1 n_1 & n_1 l_1 \\ l_2^2 & m_2^2 & n_2^2 & l_2 m_2 & m_2 n_2 & n_2 l_2 \\ l_3^2 & m_3^2 & n_3^2 & l_3 m_3 & m_3 n_3 & n_3 l_3 \\ 2l_1 l_2 & 2m_1 m_2 & 2n_1 n_2 & \left(l_1 m_2 + l_1 m_1\right) & \left(m_1 n_2 + m_2 n_1\right) & \left(n_1 l_2 + n_2 l_1\right) \\ 2l_2 l_3 & 2m_2 m_3 & 2n_2 n_3 & \left(l_2 m_3 + l_3 m_2\right) & \left(m_2 n_3 + m_3 n_2\right) & \left(n_2 l_3 + n_3 l_2\right) \\ 2l_3 l_1 & 2m_3 m_1 & 2n_3 n_1 & \left(l_3 m_1 + l_1 m_3\right) & \left(m_3 n_1 + m_1 n_3\right) & \left(n_3 l_1 + n_1 l_3\right) \end{bmatrix}$$

l_i, m_i, n_i are the direction cosines

$$x_i^{'} = l_i x_1 + m_i x_2 + n_i x_3 \qquad for\ i = 1,2,3$$

and $x_i^{'}$ denotes the material axes. The constitutive matrix C_l is defined in terms of the material axes as

$$C_l^{-1} = \begin{bmatrix} \frac{1}{E_{11}} & -\frac{\upsilon_{21}}{E_{22}} & -\frac{\upsilon_{31}}{E_{33}} & 0 & 0 & 0 \\ -\frac{\upsilon_{12}}{E_{11}} & \frac{1}{E_{22}} & -\frac{\upsilon_{32}}{E_{33}} & 0 & 0 & 0 \\ -\frac{\upsilon_{13}}{E_{11}} & -\frac{\upsilon_{23}}{E_{22}} & \frac{1}{E_{33}} & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1}{G_{12}} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1}{G_{23}} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1}{G_{31}} \end{bmatrix}$$

where the subscripts denote the material axes, i.e.,

$$\upsilon_{ij} = \upsilon_{x'_i x'_j} \quad \text{and} \quad E_{ii} = E_{x'_i}$$

Since C_l is symmetric

$$\frac{\upsilon_{12}}{E_{11}} = \frac{\upsilon_{21}}{E_{22}}, \text{etc.}$$

The vector of Green-St. Venant strain components is

$$E^t = \lfloor E_{11}, E_{22}, E_{33}, E_{12}, E_{23}, E_{31}, \rfloor$$

which include the local thermal strains which are integrated in time:

$$\varepsilon_{aa}^{n+1} = \varepsilon_{aa}^{n} + \alpha_a \left(T^{n+1} - T^n\right)$$
$$\varepsilon_{bb}^{n+1} = \varepsilon_{bb}^{n} + \alpha_b \left(T^{n+1} - T^n\right)$$
$$\varepsilon_{cc}^{n+1} = \varepsilon_{cc}^{n} + \alpha_c \left(T^{n+1} - T^n\right)$$

After computing S_{ij} we 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	none	none	none	none	none	none	none

Card 2

Variable	GAB	GBC	GCA	KFAIL	AOPT	MACF		
Type	F	F	F	F	F	F		
Default	none	none	none	0.0	0.0	1.0		

Card 3

Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		
Default	0.0	0.0	0.0	0.0	0.0	0.0		

Card 4

Variable	V1	V2	V3	D1	D2	D3	BETA	
Type	F	F	F	F	F	F	F	
Default	0.0	0.0	0.0	0.0	0.0	0.0	0.0	

Card 5

Variable	SC	XT	YT	YC	ALPH	SN	SYZ	SZX
Type	F	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	none	none

VARIABLE	DESCRIPTION
MID	Material identification. A unique number 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	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.
MACF	Material axes change flag for brick elements: EQ.1.0: default, EQ.2.0: switch material axes a and b, EQ.3.0: switch material axes a and c.
XP,YP,ZP	Coordinates of point **p** for AOPT = 1.
A1,A2,A3	Components of vector **a** for AOPT = 2.
V1,V2,V3	Components of vector **v** for AOPT = 3.
D1,D2,D3	Components of vector **d** for AOPT = 2.
BETA	Material angle in degrees for AOPT = 3, may be overridden on the element card, see *ELEMENT_SHELL_BETA or *ELEMENT_SOLID_ORTHO.
SC	Shear strength, ab plane, see 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 *(solid elements only)*
SYZ	Transverse shear strength *(solid elements only)*
SZX	Transverse shear strength *(solid elements only)*

Remarks:

The number of additional integration point variables for shells written to the LS-DYNA database is input by the optional *DATABASE_BINARY as variable NEIPS. These additional variables are tabulated below (*ip* = shell integration point):

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

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

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

Examples:

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

LS-TAURUS commands: *intg 3 frin 81*

b) Sum of failure indicator of tensile matrix mode:

LS-TAURUS commands: *intg 2 frin 7*

*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 defined. The next "*" card terminates the input.

Cards 1 for Temperature Ti

Variable	EAi	EBi	ECi	PRABi	PRCAi	PRCBi		
Type	F	F	F	F	F	F		

Cards 2 for Temperature Ti

Variable	AAi	ABi	ACi	GABi	GCAi	GCBi	Ti	
Type	F	F	F	F	F	F	F	

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
AOPT	Material axes option (see MAT_*OPTION* TROPIC_ELASTIC for a more complete description): EQ. 0.0: locally orthotropic with material axes determined by element nodes 1, 2, and 4, as with *DEFINE_ COORDINATE_NODES. EQ. 1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center; this is the a-direction. This option is for solid elements only. EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR. EQ. 3.0: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle,BETA, from a line in the plane of the element defined by the cross product of the vector **v** with the element normal. EQ. 4.0: locally orthotropic in cylindrical coordinate system with the material axes determined by a vector **v**, and an originating point, P, which define the centerline axis. This option is for solid elements only.
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.
XP,YP,ZP	Coordinates of point **p** for AOPT = 1.
A1,A2,A3	Components of vector **a** for AOPT = 2.
V1,V2,V3	Components of vector **v** for AOPT = 3.
D1,D2,D3	Components of vector **d** for AOPT = 2.
BETA	Material angle in degrees for AOPT = 3, may be overridden on the element card, see *ELEMENT_SHELL_BETA or *ELEMENT_SOLID_ORTHO.

VARIABLE	DESCRIPTION
EAi	E_a, Young's modulus in a-direction at temperature Ti.
EBi	E_b, Young's modulus in b-direction at temperature Ti.
ECi	E_c, Young's modulus in c-direction at temperature Ti.
PRBAi	ν_{ba}, Poisson's ratio ba at temperature Ti.
PRCAi	ν_{ca}, Poisson's ratio ca at temperature Ti.
PRCBi	ν_{cb}, Poisson's ratio cb at temperature Ti.
AAi	α_a, coefficient of thermal expansion in a-direction at temperature Ti.
ABi	α_b, coefficient of thermal expansion in b-direction at temperature Ti.
ACi	α_c, coefficient of thermal expansion in c-direction at temperature Ti.
GABi	G_{ab}, Shear modulus ab at temperature Ti.
GBCi	G_{bc}, Shear modulus bc at temperature Ti.
GCAi	G_{ca}, Shear modulus ca at temperature Ti.
Ti	ith temperature

Remarks:

In the implementation for three dimensional continua a total Lagrangian formulation is used. In this approach the material law that relates second Piola-Kirchhoff stress S to the Green-St. Venant strain E is

$$S = C \cdot E = T^t C_l T \cdot E$$

where T is the transformation matrix [Cook 1974].

$$T = \begin{bmatrix} l_1^2 & m_1^2 & n_1^2 & l_1 m_1 & m_1 n_1 & n_1 l_1 \\ l_2^2 & m_2^2 & n_2^2 & l_2 m_2 & m_2 n_2 & n_2 l_2 \\ l_3^2 & m_3^2 & n_3^2 & l_3 m_3 & m_3 n_3 & n_3 l_3 \\ 2l_1 l_2 & 2m_1 m_2 & 2n_1 n_2 & (l_1 m_2 + l_1 m_1) & (m_1 n_2 + m_2 n_1) & (n_1 l_2 + n_2 l_1) \\ 2l_2 l_3 & 2m_2 m_3 & 2n_2 n_3 & (l_2 m_3 + l_3 m_2) & (m_2 n_3 + m_3 n_2) & (n_2 l_3 + n_3 l_2) \\ 2l_3 l_1 & 2m_3 m_1 & 2n_3 n_1 & (l_3 m_1 + l_1 m_3) & (m_3 n_1 + m_1 n_3) & (n_3 l_1 + n_1 l_3) \end{bmatrix}$$

l_i, m_i, n_i are the direction cosines

$$x_i' = l_i x_1 + m_i x_2 + n_i x_3 \qquad for\ i = 1,2,3$$

and x_i' denotes the material axes. The temperature dependent constitutive matrix C_l is defined in terms of the material axes as

$$C_l^{-1} = \begin{bmatrix} \frac{1}{E_{11}(T)} & -\frac{\upsilon_{21}(T)}{E_{22}(T)} & -\frac{\upsilon_{31}(T)}{E_{33}(T)} & 0 & 0 & 0 \\ -\frac{\upsilon_{12}(T)}{E_{11}(T)} & \frac{1}{E_{22}(T)} & -\frac{\upsilon_{32}(T)}{E_{33}(T)} & 0 & 0 & 0 \\ -\frac{\upsilon_{13}(T)}{E_{11}(T)} & -\frac{\upsilon_{23}(T)}{E_{22}(T)} & \frac{1}{E_{33}(T)} & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1}{G_{12}(T)} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1}{G_{23}(T)} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1}{G_{31}(T)} \end{bmatrix}$$

where the subscripts denote the material axes, i.e.,

$$\upsilon_{ij} = \upsilon_{x_i' x_j'} \quad \text{and} \quad E_{ii} = E_{x_i'}$$

Since C_l is symmetric

$$\frac{\upsilon_{12}}{E_{11}} = \frac{\upsilon_{21}}{E_{22}}, \text{etc.}$$

The vector of Green-St. Venant strain components is

$$E^t = \lfloor E_{11}, E_{22}, E_{33}, E_{12}, E_{23}, E_{31}, \rfloor$$

which include the local thermal strains which are integrated in time:

$$\varepsilon_{aa}^{n+1} = \varepsilon_{aa}^{n} + \alpha_a (T^{n+\frac{1}{2}}) \left[T^{n+1} - T^n\right]$$

$$\varepsilon_{bb}^{n+1} = \varepsilon_{bb}^{n} + \alpha_b (T^{n+\frac{1}{2}}) \left[T^{n+1} - T^n\right]$$

$$\varepsilon_{cc}^{n+1} = \varepsilon_{cc}^{n} + \alpha_c (T^{n+\frac{1}{2}}) \left[T^{n+1} - T^n\right]$$

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

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
E	Young’s modulus.
PR	Poisson’s ratio.
SIGY	Yield stress.
ETAN	Tangent modulus, ignored if (LCSS.GT.0) is defined.
FAIL	Failure flag. LT.0.0: User defined failure subroutine is called to determine failure EQ.0.0: Failure is not considered. This option is recommended if failure is not of interest since many caluculations will be saved. GT.0.0: Plastic strain to failure. When the plastic strain reaches this value, the element is deleted from the calculation.
TDEL	Minimum time step size for automatic element deletion.
C	Strain rate parameter, C, see formula below.
P	Strain rate parameter, P, see formula below.
LCSS	Load curve ID or Table ID. Load curve ID defining effective stress versus effective plastic strain. If defined EPS1-EPS8 and ES1-ES8 are ignored. The table ID defines for each strain rate value a load curve ID giving the stress versus effectiveplastic strain for that rate, See Figure 20.7. The stress versus effective plastic strain curve for the lowest value of strain rate is used if the strain rate falls below the minimum value. Likewise, the stress versus effective plastic strain curve for the highest value of strain rate is used if the strain rate exceeds the maximum value. The strain rate parameters: C and P;

VARIABLE	DESCRIPTION
	the curve ID, LCSR; EPS1-EPS8 and ES1-ES8 are ignored if a Table ID is defined.
LCSR	Load curve ID defining strain rate scaling effect on yield stress.
VP	Formulation for rate effects: EQ.-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

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

where $\dot{\varepsilon}$ is the strain rate. $\dot{\varepsilon}=\sqrt{\dot{\varepsilon}_{ij}\dot{\varepsilon}_{ij}}$. If VP=-1. the deviatoric strain rates are used instead.

If the viscoplastic option is active, VP=1.0, and if SIGY is > 0 then the dynamic yield stress is computed from the sum of the static stress, $\sigma_y^s\left(\varepsilon_{eff}^p\right)$, which is typically given by a load curve ID, and the initial yield stress, SIGY, multiplied by the Cowper-Symonds rate term as follows:

$$\sigma_y\left(\varepsilon_{eff}^p,\dot{\varepsilon}_{eff}^p\right)=\sigma_y^s\left(\varepsilon_{eff}^p\right)+SIGY\cdot\left(\frac{\dot{\varepsilon}_{eff}^p}{C}\right)^{1/p}$$

where the plastic strain rate is used. With this latter approach similar results can be obtained between this model and material model: *MAT_ANISOTROPIC_VISCOPLASTIC. If SIGY=0, the following equation is used instead where the static stress, $\sigma_y^s\left(\varepsilon_{eff}^p\right)$, must be defined by a load curve:

$$\sigma_y\left(\varepsilon_{eff}^p,\dot{\varepsilon}_{eff}^p\right)=\sigma_y^s\left(\varepsilon_{eff}^p\right)\left[1+\left(\frac{\dot{\varepsilon}_{eff}^p}{C}\right)^{1/p}\right]$$

This latter equation is always used if the viscoplastic option is off.

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

III. If different stress versus strain curves can be provided for various strain rates, the option using the reference to a table (LCSS) can be used. Then the table input in *DEFINE_TABLE has to be used, see Figure 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 is results can be dramatic.

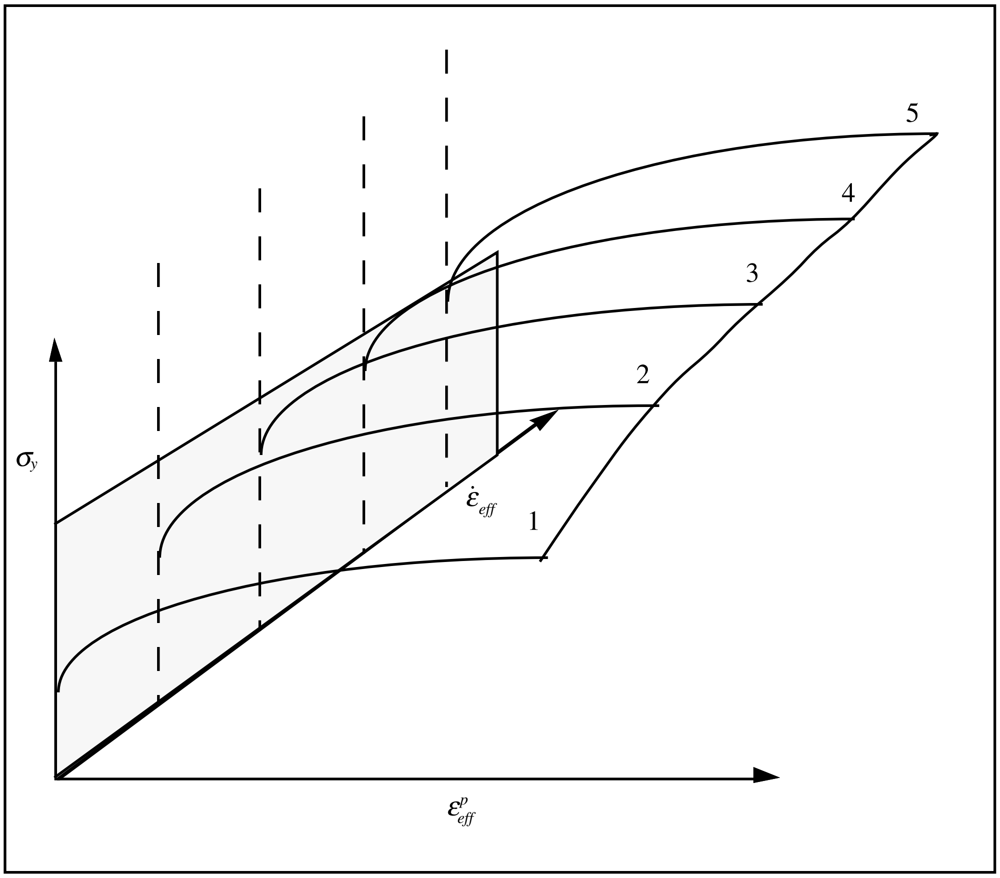


Figure 20.7. Rate effects may be accounted for by defining a table of curves. If a table ID is specified a curve ID is given for each strain rate, see *DEFINE_TABLE. Intermediate values are found by interpolating between curves. Effective plastic strain versus yield stress is expected. If the strain 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 invarient, J_1, EQ.5: second stress invarient, $\sqrt{J_2}$, 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.

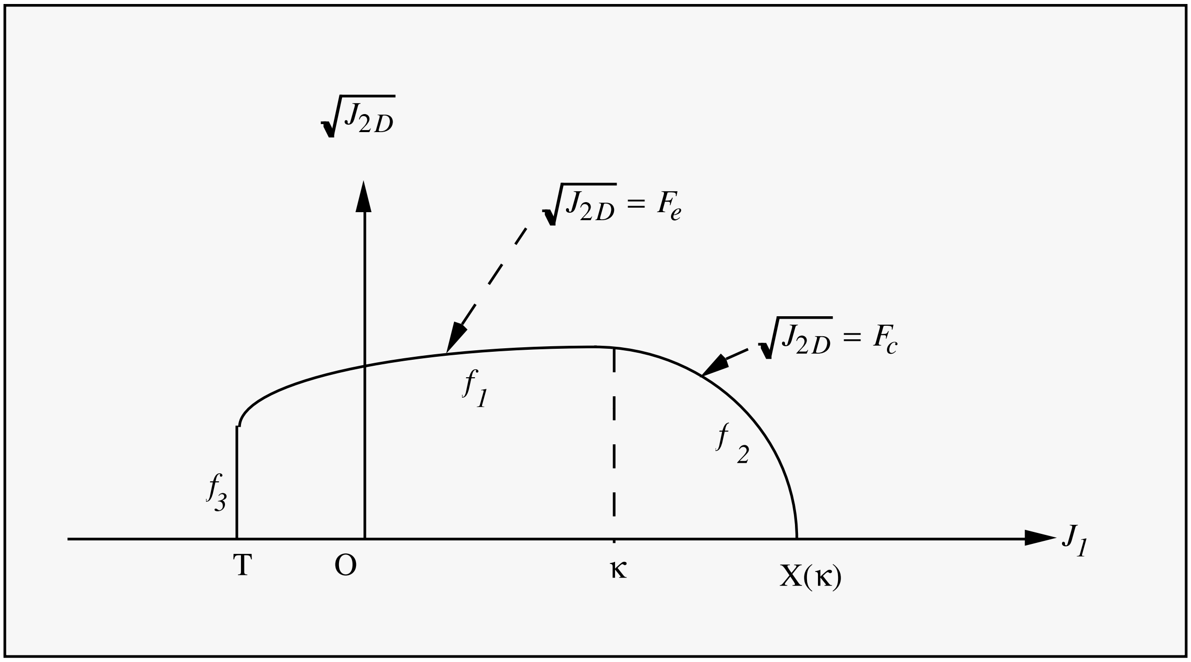


Figure 20.8. The yield surface of the two-invariant cap model in pressure $\sqrt{J_{2D}} - J_1$ space. Surface f_1 is the failure envelope, f_2 is the cap surface, and f_3 is the tension cutoff.

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

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

where F_e is given by

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

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

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

where F_c is defined by

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

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

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

and $L(\kappa)$ is defined by

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

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

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

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

$$f_3 \equiv T - J_1,$$

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

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

$$\boldsymbol{\varepsilon} = \boldsymbol{\varepsilon}^e + \boldsymbol{\varepsilon}^p,$$

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

$$\boldsymbol{\sigma} = \mathbf{C}(\boldsymbol{\varepsilon} - \boldsymbol{\varepsilon}^p),$$

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

The yield condition may be written

$$f_1(\mathrm{s}) \le 0$$
$$f_2(\mathrm{s},\kappa) \le 0$$
$$f_3(\mathrm{s}) \le 0$$

and the plastic consistency condition requires that

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

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

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

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

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 and N are specified. In this case, the failure envelope surface is replaced by a family of yield surfaces bounded by an initial yield surface and a limiting failure envelope surface. Thus, the shape of the yield surfaces described above remains unchanged, but they may translate in a plane orthogonal to the J axis,

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

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

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

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

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

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

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

VARIABLE	DESCRIPTION
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 μ=0 This will give results identical to previous versions of LS-DYNA.
LCA	Load curve ID, see *DEFINE_CURVE, for sigma-aa versus either relative volume or volumetric strain. See notes below.
LCB	Load curve ID, see *DEFINE_CURVE, for sigma-bb versus either relative volume or volumetric strain. Default LCB=LCA. See notes below.
LCC	Load curve ID, see *DEFINE_CURVE, for sigma-cc versus either relative volume or volumetric strain. Default LCC=LCA. See notes below.
LCS	Load curve ID, see *DEFINE_CURVE, for shear stress versus either relative volume or volumetric strain. Default LCS=LCA. Each component of shear stress may have its own load curve. See notes below.
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 abcissa. If this recommendation is followed the cost of the table lookup is insignificant. Conversely, the cost increases significantly if the abcissa strain values are not consistent between load curves.

The behavior before compaction is orthotropic where the components of the stress tensor are uncoupled, i.e., an a component of strain will generate resistance in the local *a*-direction with no coupling to the local *b* and *c* directions. The elastic 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

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

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

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

The relative volume, V, is defined as the ratio of the current volume 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:

$$\varepsilon_V = 1 - V$$

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

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

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

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

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

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

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

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

Each component of the updated stresses is then independently checked to ensure that they do not exceed the permissible values determined from the load curves; e.g., if

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

then

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

On Card 2 $\sigma_{ij}(V)$ is defined by LCA for the aa stress component, LCB for the bb component, LCC for the cc component, and LCS for the ab, bc, cb shear stress components. The parameter λ is either unity or a value taken from the load curve number, LCSR, that defines λ as a function of strain-rate. Strain-rate is defined here as the Euclidean norm of the deviatoric strain-rate tensor.

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

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

where the deviatoric strain increment is defined as

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

Now a check is made to see if the yield stress for the fully compacted material is exceeded by comparing

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

the effective trial stress to the defined yield stress, SIGY. If the effective trial stress exceeds the yield stress the stress components are simply scaled back to the yield surface

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

Now the pressure is updated using the elastic bulk modulus, K

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

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

to obtain the final value for the Cauchy stress

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

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

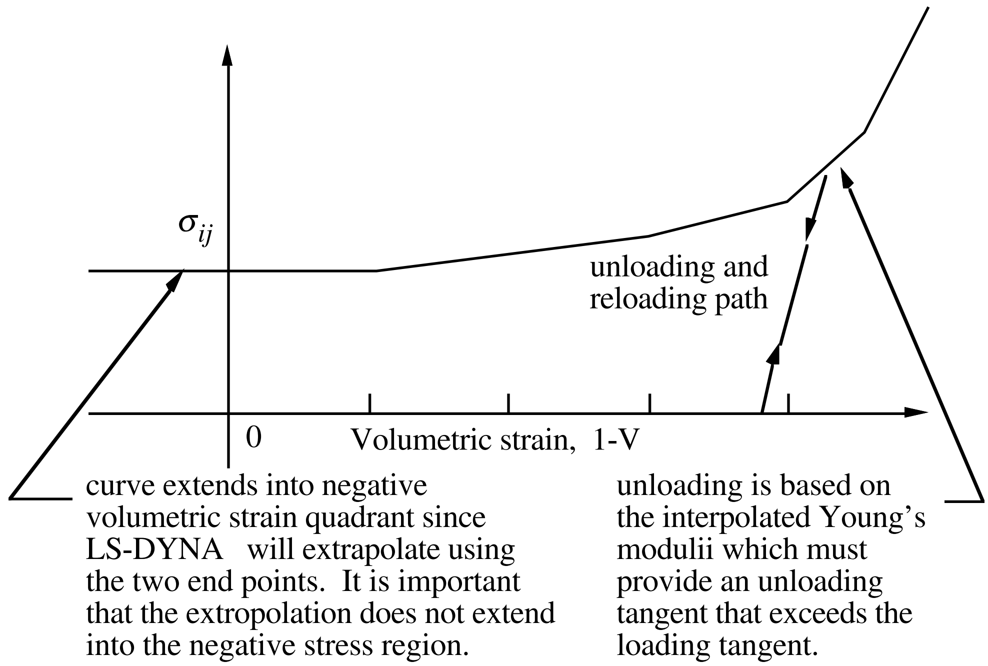


Figure 20.9. Stress quantity versus volumetric strain. Note that the "yield stress" at a volumetric strain of zero is non-zero. In the load curve definition, see *DEFINE_CURVE, the "time" value is the volumetric strain and the "function" value is the yield stress.

*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				

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
PR	Poisson’s ratio (> .49 is recommended, smaller values may not work).
A	Constant, see literature and equations defined below.
B	Constant, see literature and equations defined below.
REF	Use reference geometry to initialize the stress tensor. The reference 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.

VARIABLE	DESCRIPTION
SW	Specimen width, see Figure 20.10.
ST	Specimen thickness, see Figure 20.10.
LCID	Load curve ID, see *DEFINE_CURVE, giving the force versus actual change ΔL in the gauge length. See also Figure 20.11 for an alternative definition.

Remarks:

The strain energy density function is defined as:

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

where

$$C = 0.5A + B$$

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

υ = Poisson's ratio

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

I, II, III = invariants of right Cauchy-Green Tensor C.

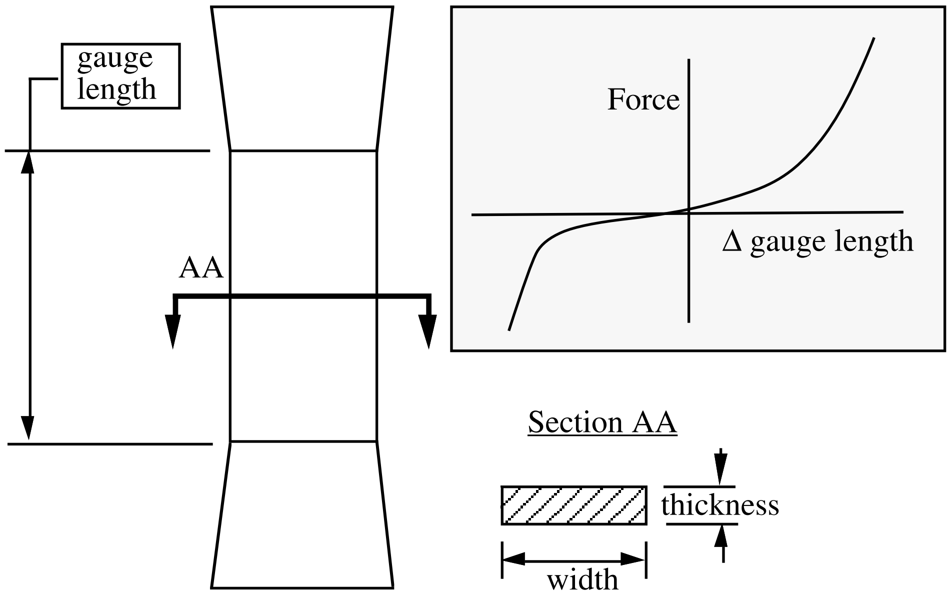
The load curve definition that provides the uniaxial data should give the change in gauge length, ΔL, versus the corresponding force. In compression both the force and the change in gauge length must be specified as negative values. In tension the force and change in gauge length should be input as positive values. The principal stretch ratio in the uniaxial direction, λ_1, is then given by

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

with L_0 being the initial length and L being the actual length.

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

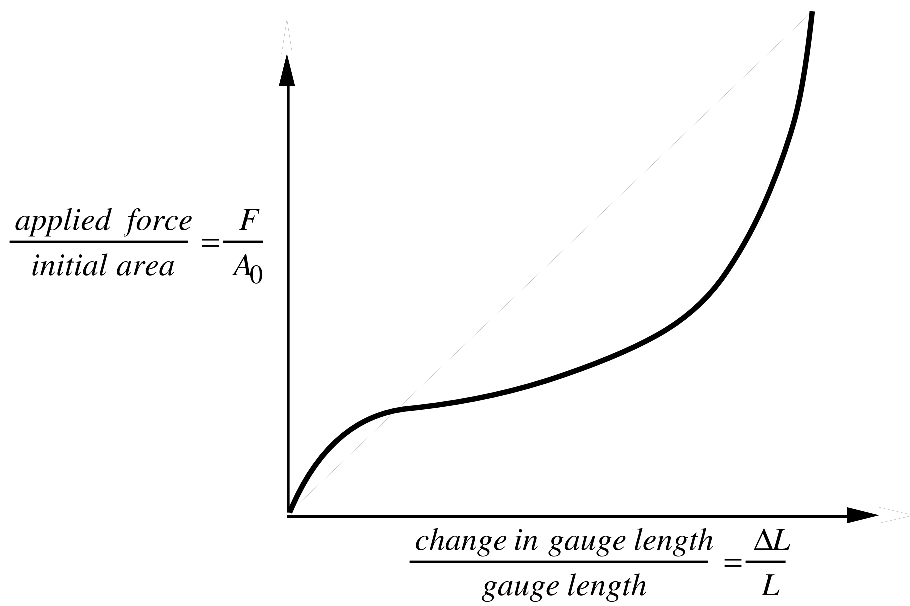


Figure 20.11 The stress versus strain curve can used instead of the force versus the change in the gauge length by setting the gauge length, thickness, and width to unity (1.0) and defining the engineering strain in place of the change in gauge length and the nominal (engineering) stress in place of 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		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density
E	Young’s modulus
PR	Poisson’s ratio
SIGY	Yield stress
ETAN	Plastic hardening modulus (for shells only)

*MAT_FORCE_LIMITED

This is Material Type 29. With this material model, for the Belytschko-Schwer beam only, plastic hinge forming at the ends of a beam can be modeled using curve definitions. Optionally, collapse can also be modelled.

Description: FORCE LIMITED Resultant Formulation

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	DF	AOPT	YTFLAG	ASOFT
Type	I	F	F	F	F	F	F	F
Default	none	none	none	none	0.0	0.0	0.0	0.0

Card 2								
Variable	M1	M2	M3	M4	M5	M6	M7	M8
Type	F	F	F	F	F	F	F	F
Default	none	0	0	0	0	0	0	0

Card 3								
Variable	LC1	LC2	LC3	LC4	LC5	LC6	LC7	LC8
Type	F	F	F	F	F	F	F	F
Default	none	0	0	0	0	0	0	0

Card 4

Variable	LPS1	SFS1	LPS2	SFS2	YMS1	YMS2		
Type	F	F	F	F	F	F		
Default	0	1.0	LPS1	1.0	1.0E+20	YMS1		

Card 5

Variable	LPT1	SFT1	LPT2	SFT2	YMT1	YMT2		
Type	F	F	F	F	F	F		
Default	0	1.0	LPT1	1.0	1.0E+20	YMT1		

Card 6

Variable	LPR	SFR	YMR					
Type	F	F	F					
Default	0	1.0	1.0E+20					

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density
E	Young's modulus
PR	Poisson's ratio
DF	Damping factor, see definition in notes below. A proper control for the timestep has to be maintained by the user!

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.

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

$$M_{r_{upper}} = MAX\left(M_r, \frac{M_{r_{yield}}}{2}\right)$$

and similar for M_s and M_t.

Thereafter the plastic moments will be given by

M_{rp}, = min (M_{rupper}, M_{rcurve}) and similar for s and t

where

M_{rp} = current plastic moment

M_{rcurve} = moment taken from load curve at the current rotation scaled according to the scale factor.

The effect of this is to provide an upper limit to the moment that can be generated; it represents the softening effect of local buckling at a hinge site. Thus if a member is bent about is local s-axis it will then be weaker in torsion and about its local t-axis. For moments-softening curves, the effect is to trim off the initial peak (although if the curves subsequently harden, the final hardening will also be trimmed off).

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

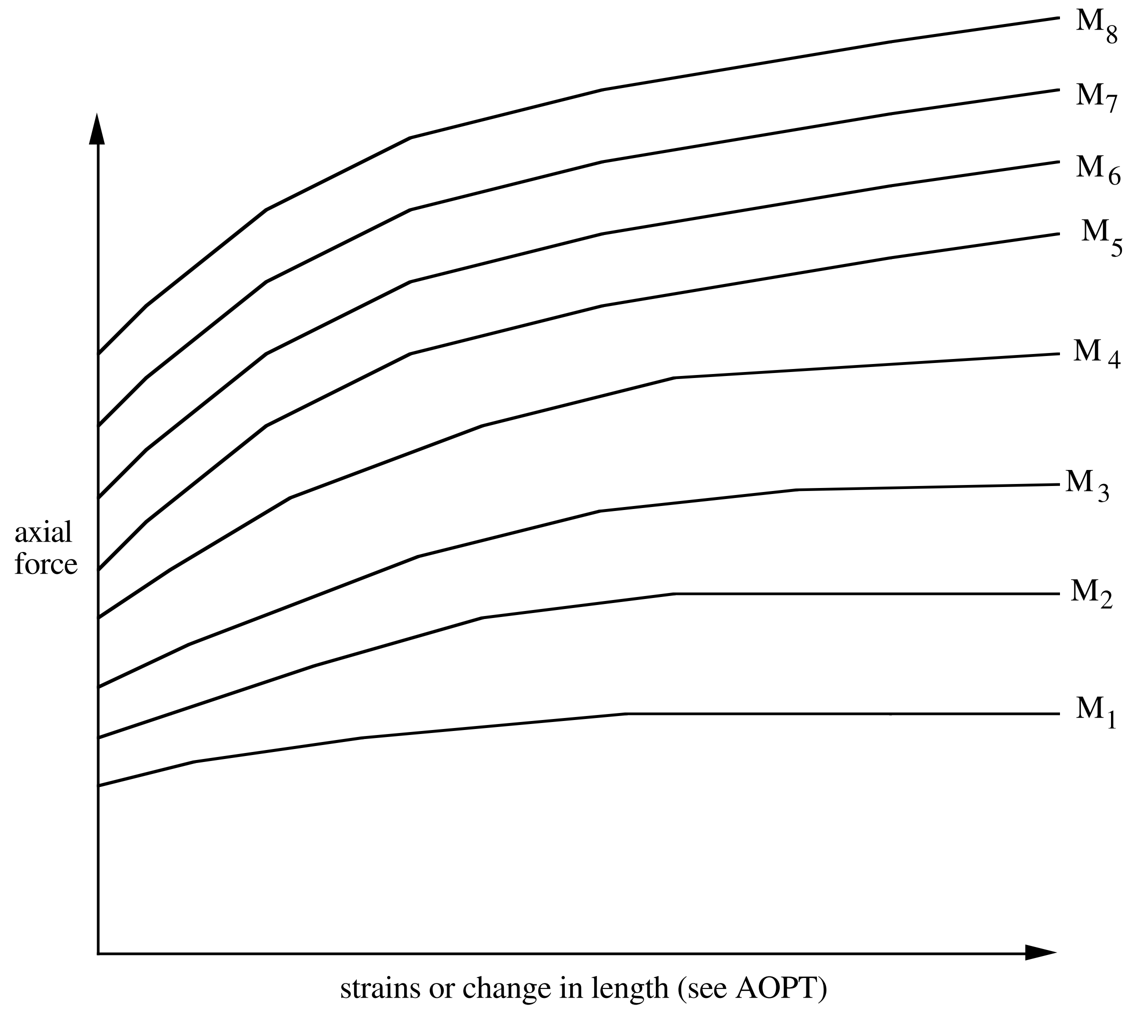


Figure 20.12. The force magnitude is limited by the applied end moment. For an intermediate value of the end moment LS-DYNA interpolates between the curves to determine the allowable force value.

*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:

$$\alpha = \frac{\sigma_s^{AS,-} - \sigma_s^{AS,+}}{\sigma_s^{AS,-} + \sigma_s^{AS,+}}$$

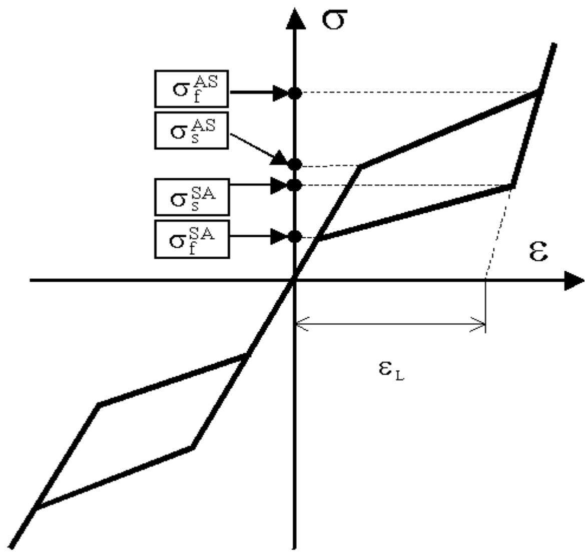


Figure 20.13: Pictorial representation of superelastic behavior for a shape-memory material.

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.

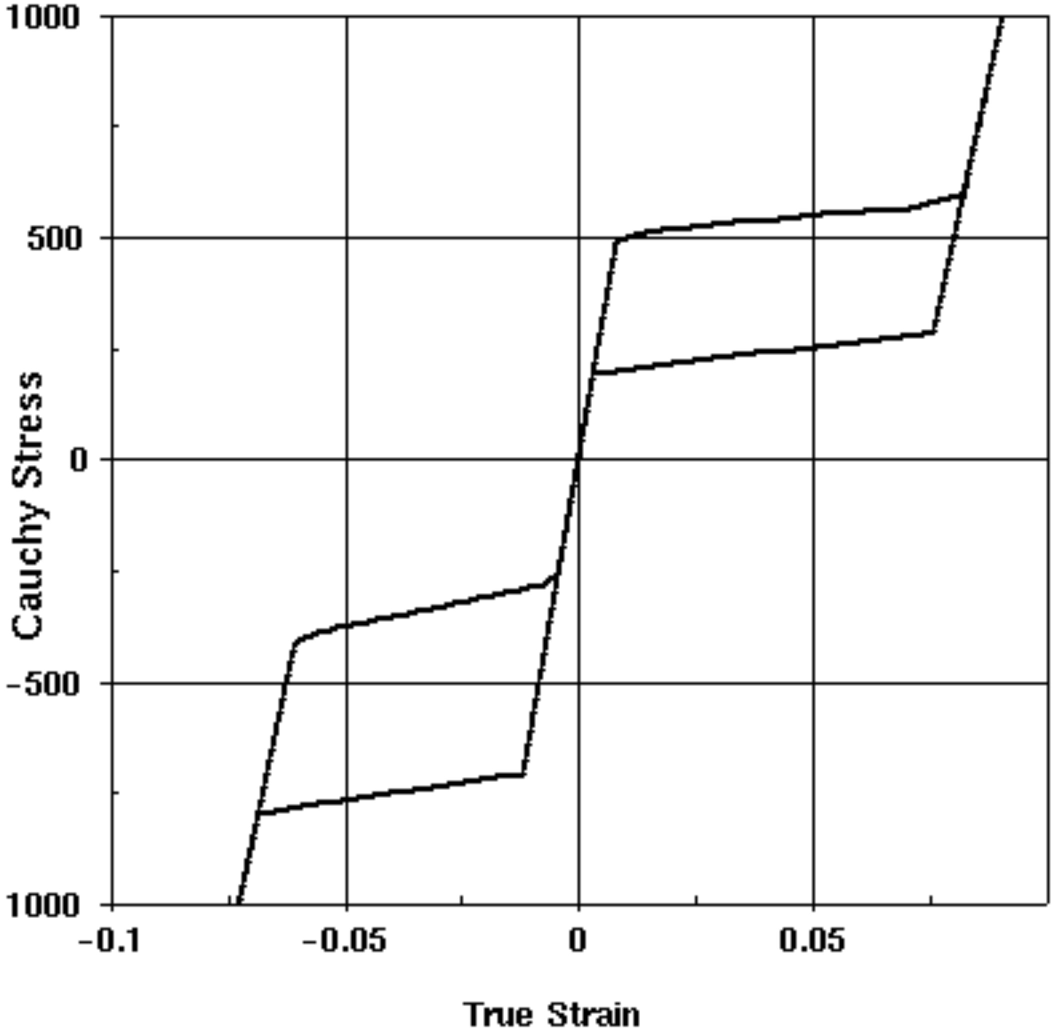


Figure 20.14. Complete loading-unloading test in tension and compression

*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	C_{100} (EQ.1.0 if term is in the least squares fit.).
C200	C_{200} (EQ.1.0 if term is in the least squares fit.).

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} :

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

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

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

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

The load curve definition that provides the uniaxial data should give the change in gauge length, ΔL, and the corresponding force . In compression both the force and the change in gauge length must be specified as negative values. In tension the force and change in gauge length should be input as positive values. The principal stretch ratio in the uniaxial direction, λ_1, is then given by

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

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

VARIABLE	DESCRIPTION
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

VARIABLE	DESCRIPTION
SYG	Yield stress for glass
ETG	Plastic hardening modulus for glass
EFG	Plastic strain at failure for glass
EP	Young's modulus for polymer
PRP	Poisson's ratio for polymer
SYP	Yield stress for polymer
ETP	Plastic hardening modulus for polymer
F1,..FN	Integration point material: f_n = 0.0: glass, f_n = 1.0: polymer. A user-defined integration rule must be specified, see *INTEGRATION_SHELL.

Remarks:

Isotropic hardening for both materials is assumed. The material to which the glass is bonded is assumed to stretch plastically without failure. A user defined integration rule specifies the thickness of the layers making up the glass. F_i defines whether the integration point is glass (0.0) or polymer (1.0). The material definition, F_i, has to be given for the same number of integration points (NIPTS) as specified in the rule. A maximum of 32 layers is allowed.

*MAT_BARLAT_ANISOTROPIC_PLASTICITY

This is Material Type 33. This model was developed by Barlat, Lege, and Brem [1991] for modelling anisotropic material behavior in forming processes. The finite element implementation of this model is described in detail by Chung and Shah [1992] and is used here. It is based on a six parameter model, which is ideally suited for 3D continuum problems, see notes below. For sheet forming problems, material 36 based on a 3-parameter model is recommended.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	K	E0	N	M
Type	I	F	F	F	F	F	F	F

Card 2								
Variable	A	B	C	F	G	H		
Type	F	F	F	F	F	F		

Card 3								
Variable	AOPT	OFFANG						
Type	F	F						

Card 4								
Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card 5

Variable	V1	V2	V3	D1	D2	D3		
Type	F	F	F	F	F	F		

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.
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.
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.

VARIABLE	DESCRIPTION
	EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR. EQ. 3.0: locally orthotropic material axes determined by offsetting the material axes by an angle, OFFANG, from a line determined by taking the cross product of the vector **v** with the normal to the plane of 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:

$$\Phi = |S_1 - S_2|^m + |S_2 - S_3|^m + |S_3 - S_1|^m = 2\overline{\sigma}^m$$

where $\overline{\sigma}$ is the effective stress and $S_{i=1,2,3}$ are the principal values of the symmetric matrix $S_{\alpha\beta}$,

$$\begin{aligned}
S_{xx} &= [c(\sigma_{xx} - \sigma_{yy}) - b(\sigma_{zz} - \sigma_{xx})]/3 \\
S_{yy} &= [a(\sigma_{yy} - \sigma_{zz}) - c(\sigma_{xx} - \sigma_{yy})]/3 \\
S_{zz} &= [b(\sigma_{zz} - \sigma_{xx}) - a(\sigma_{yy} - \sigma_{zz})]/3 \\
S_{yz} &= f\sigma_{yz} \\
S_{zx} &= g\sigma_{zx} \\
S_{xy} &= h\sigma_{xy}
\end{aligned}$$

The material constants a, b, c, f, g and h represent anisotropic properties. When $a = b = c = f = g = h = 1$, the material is isotropic and the yield surface reduces to the Tresca yield surface for $m = 1$ and von Mises yield surface for $m = 2 or 4$.

For face centered cubic (FCC) materials m=8 is recommended and for body centered cubic (BCC) materials $m = 6$ is used. The yield strength of the material is

$$\sigma_y = k\left(\varepsilon^p + \varepsilon_0\right)^n$$

where ε_0 is the strain corresponding to the initial yield stress and ε^p is the plastic strain.

*MAT_BARLAT_YLD96

This is Material Type 33. This model was developed by Barlat, Maeda, Chung, Yanagawa, Brem, Hayashida, Lege, Matsui, Murtha, Hattori, Becker, and Makosey [1997] for modeling anisotropic material behavior in forming processes in particular for aluminum alloys. This model is available for shell elements only.

Card 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		

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.
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 **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:

$$\sigma_y = k\left(\varepsilon_0 + \varepsilon^p\right)^n \left(\frac{\dot{\varepsilon}}{\varepsilon_{SR0}}\right)^m$$

The second, the Voce equation, is defined as:

$$\sigma_y = a - be^{-c\varepsilon^p}$$

and the third option is to give a load curve ID that defines the yield stress as a function of effective plastic strain. The yield function Φ is defined as:

$$\Phi = \alpha_1 \mid s_1 - s_2 \mid^a + \alpha_2 \mid s_2 - s_3 \mid^a + \alpha_3 \mid s_3 - s_1 \mid^a = 2\sigma_y{}^a$$

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 $\underset{\sim}{s}$ andthe principal anisotropy axes.

*MAT_FABRIC

This is Material Type 34. This material is especially developed for airbag materials. The fabric model is a variation on the layered orthotropic composite model of material 22 and is valid for 3 and 4 node membrane elements only. In addition to being a constitutive model, this model also invokes a special membrane element formulation which is more suited to the deformation experienced by fabrics under large deformation. For thin fabrics, buckling can result in an inability to support compressive stresses; thus a flag is included for this option. A linearly elastic liner is also included which can be used to reduce the tendency for these elements to be crushed when the no-compression option is invoked. In LS-DYNA versions after 931 the isotropic elastic option is available.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	EA	EB	EC	PRBA	PRCA	PRCB
Type	I	F	F	F	F	F	F	F

Card 2								
Variable	GAB	GBC	GCA	CSE	EL	PRL	LRATIO	DAMP
Type	F	F	F	F	F	F	F	F
Remarks				1	2	2	2	

Card 3								
Variable	AOPT	FLC	FAC	ELA	LNRC	FORM		
Type	F	F	F	F	F	F		
Remarks		3	3		4	0		

Card 4

Variable				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		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
EA	Young's modulus - longitudinal direction. For an isotopic elastic fabric material only EA and PRBA are defined and are used as the isotropic Young's modulus and Poisson's ratio, respectively. The input for the fiber directions and liner should be input as zero for the isotropic elastic fabric.
EB	Young's modulus - transverse direction, set to zero for isotropic elastic material.
EC	Young's modulus - normal direction, set to zero for isotropic elastic material.
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 **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:invarient 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.
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.
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	Load curve ID for stress versus strain along the b-axis fiber; available for FORM=4 only. If zero, LCB is used.
LCUAB	Load 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 discritization 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 there 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.

*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	1	2	3	4	5	6	7	8
Variable	SIGY	ETAN	SRC	SRP	BETA			
Type	F	F	F	F	F			

VARIABLE	DESCRIPTION
MID	Material identification
RO	Density
E	Young’s modulus
PR	Poisson’s ratio
SIGY	Yield stress
ETAN	Plastic hardening modulus
SRC	Strain rate parameter, C
SRP	Strain rate parameter, P
BETA	Hardening parameter, $0 < \beta' < 1$

*MAT_3-PARAMETER_BARLAT

This is Material Type 36. This model was developed by Barlat and Lian [1989] for modelling sheets with anisotropic materials under plane stress conditions. This material allows the use of the Lankford parameters for the definition of the anisotropy. This particular development is due to Barlat and Lian [1989].

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	HR	P1	P2	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 parmeter determined from experiments
R45	R_{45}, Lankford parmeter determined from experiments
R90	R_{90}, Lankford parmeter 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
SPI	*spi*, if ε_0 is zero above. (Default=0.0)0 EQ.0.0: $\varepsilon_0 = (E/k)**[1/(n-1)]$ LE..02: $\varepsilon_0 = spi$ GT..02: $\varepsilon_0 = (spi/k)**[1/n]$
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.

Remarks:

The anisotopic 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 ϕ=45 and define a function g as

$$g(p) = \frac{2m\sigma_Y^m}{\left(\frac{\partial\Phi}{\partial\sigma_x} + \frac{\partial\Phi}{\partial\sigma_y}\right)\sigma_{\phi}} - 1 - R_{45}$$

An iterative search is used to find the value of p.

For 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 (logrithmic). If SIGY is set to zero, the strain to yield if found by solving for the intersection of the linearly elastic loading equation with the strain hardening equation:

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

which gives the elastic strain at yield as:

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

If SIGY yield is nonzero and greater than 0.02 then:

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

*MAT_TRANSVERSELY_ANISOTROPIC_ELASTIC_PLASTIC

This is Material Type 37. This model is for simulating sheet forming processes with anisotropic material. Only transverse anisotropy can be considered. Optionally an arbitrary dependency of stress and effective plastic strain can be defined via a load curve. This plasticity model is fully iterative and is available only for shell elements. Also see the notes below.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	SIGY	ETAN	R	HLCID
Type	I	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.
SIGY	Yield stress.
ETAN	Plastic hardening modulus.
R	Anisotropic hardening parameter.
HLCID	Load curve ID defining effective yield stress versus effective plastic strain.

Remarks:

Consider Cartesian reference axes which are parallel to the three symmetry planes of anisotropic behavior. Then, the yield function suggested by [Hill 1948] can be written

$$F(\sigma_{22}-\sigma_{33})^2+G(\sigma_{33}-\sigma_{11})^2+H(\sigma_{11}-\sigma_{22})^2+2L\sigma_{23}^2+2M\sigma_{31}^2+2N\sigma_{12}^2-1=0$$

where 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

$$F(\sigma) \equiv \Big[\sigma_{11}^2 + \sigma_{22}^2 + K^2\sigma_{33}^2 - K^2\sigma_{33}(\sigma_{11}+\sigma_{22}) - (2-K^2)\sigma_{11}\sigma_{22} + 2L\sigma_y^2(\sigma_{23}^2+\sigma_{31}^2) + 2\left(2-\frac{1}{2}K^2\right)\sigma_{12}^2\Big]^{1/2}$$

The rate of plastic strain is assumed to be normal to the yield surface so $\dot{\varepsilon}_{ij}^p$ is found from

$$\dot{\varepsilon}_{ij}^p = \lambda \frac{\partial F}{\partial \sigma_{ij}}.$$

Now consider the case of plane stress, where $\sigma_{33} = 0$. Also, define the anisotropy input parameter, R, as the ratio of the in-plane plastic strain rate to the out-of-plane plastic strain rate,

$$R = \frac{\dot{\varepsilon}_{22}^p}{\dot{\varepsilon}_{33}^p}.$$

It then follows that

$$R = \frac{2}{K^2} - 1.$$

Using the plane stress assumption and the definition of R, the yield function may now be written

$$F(\sigma) = \left[\sigma_{11}^2 + \sigma_{22}^2 - \frac{2R}{R+1}\sigma_{11}\sigma_{22} + 2\frac{2R+1}{R+1}\sigma_{12}^2\right]^{1/2}.$$

Note that there are several differences between this model and other plasticity models for shell elements such as the model, MAT_PIECEWISE_LINEAR_PLASTICITY. First, the yield function for plane stress does not include the transverse shear stress components which are updated elastically, and, secondly, this model is always fully iterative. Consequently, in comparing results for the isotropic case where R=1.0 with other isotropic model, differences in the results are expected, even though they are usually insignificant.

*MAT_BLATZ-KO_FOAM

This is Material Type 38. This model is for the definition of rubber like foams of polyurethane. It is a simpe 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 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 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							

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.
SIGY	Yield stress.
ETAN	Plastic hardening modulus, see notes for model 37.
R	Anisotropic hardening parameter, see notes for model 37.

VARIABLE	DESCRIPTION
HLCID	Load curve ID defining effective stress versus effective plastic strain. The yield stress and hardening modulus are ignored with this option.
LCIDFLD	Load curve ID defining the Forming Limit Diagram. Minor strains in percent are defined as abcissa 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:

$$\frac{\varepsilon_{major_{workpiece}}}{\varepsilon_{major_{fld}}}$$

corresponding to $\varepsilon_{minor_{workpiece}}$.

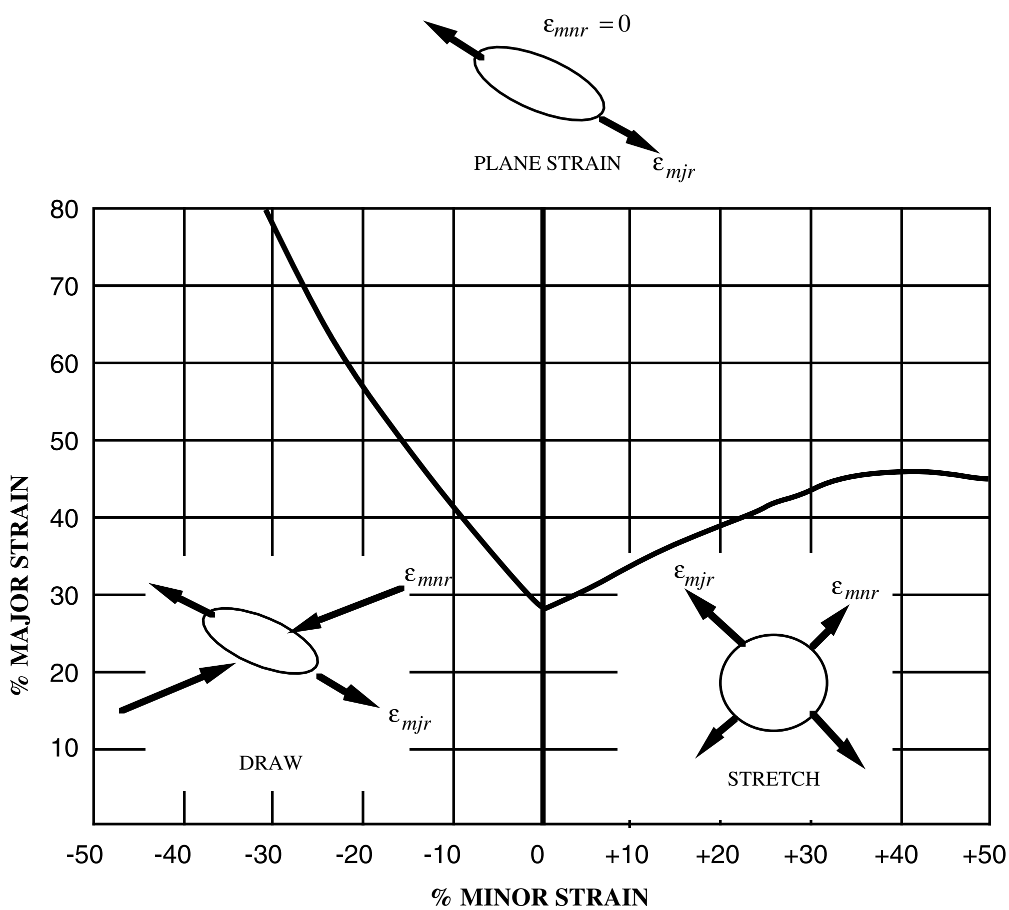


Figure 20.15. Forming Limit Diagram.

*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 elements. We do not recommend using this model at this time since it can be unstable especially if the stress-strain curves increase in stiffness with increasing strain.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	EA	EB	EC	PRBA	PRCA	PRCB
Type	I	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	none	none

Card 2	1	2	3	4	5	6	7	8
Variable	GAB	GBC	GCA	DT	TRAMP	ALPHA		
Type	F	F	F	F	F	F		
Default	none	none	none	0	0	0		

Card 3	1	2	3	4	5	6	7	8
Variable	LCIDA	LCIDB	EFAIL	DTFAIL	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 λ_a-1 where λ_a is the stretch ratio along the a axis.
LCIDB	Optional load curve ID defining the nominal stress versus strain along b-axis. Strain is defined as λ_b-1 where λ_b is the stretch ratio along the b axis.
EFAIL	Failure strain, λ-1.
DTFAIL	Time step for automatic element erosion
CDAMP	Damping coefficient.
AOPT	Material axes option (see **MAT_*OPTION* TROPIC_ELASTIC** for 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.

LCIDC	Load curve ID defining the nominal stress versus strain along c-axis. Strain is defined as λ_c-1 where λ_c is the stretch ratio along the c axis.

VARIABLE	DESCRIPTION
LCIDAB	Load curve ID defining the nominal ab shear stress versus ab-strain in the ab-plane. Strain is defined as the sin(γ_{ab}) where γ_{ab} is the shear angle.
LCIDBC	Load curve ID defining the nominal ab shear stress versus ab-strain in the bc-plane. Strain is defined as the sin(γ_{bc}) where γ_{bc} is the shear angle.
LCIDCA	Load curve ID defining the nominal ab shear stress versus ab-strain in the ca-plane. Strain is defined as the sin(γ_{ca}) where γ_{ca} is the shear angle.

*MAT_USER_DEFINED_MATERIAL_MODELS

These are Material Types 41-50. The user can supply his own subroutines. See also Appendix A. The keyword input has to be used for the user interface with data. Isotopic and anisotropic material models with failure can be handled.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	MT	LMC	NHV	IORTHO	IBULK	IG
Type	I	F	I	I	I	I	I	I

Card 2

Variable	IVECT	IFAIL	ITERMAL					
Type	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

VARIABLE	DESCRIPTION
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 the elements due to a material failure criterion.
ITHERMAL	Failure 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.

EQ. 4.0: locally orthotropic in cylindrical coordinate system with the material axes determined by a vector **v**, and an originating point, P, which define the centerline axis. This option is for solid elements only.

VARIABLE	DESCRIPTION
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.
.	.
.	.
.	.
PLCM	LCMth material parameter.

*MAT_BAMMAN

This is Material Type 51. It allows the modeling of temperature and rate dependent plasticity with a fairly complex model that has many input parameters [Bamman, 1989].

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	T	HC		
Type	I	F	F	F	F	F		

Card 2

Variable	C1	C2	C3	C4	C5	C6	C7	C8
Type	F	F	F	F	F	F	F	F

Card 3

Variable	C9	C10	C11	C12	C13	C14	C15	C16
Type	F	F	F	F	F	F	F	F

Card 4

Variable	C17	C18	A1	A2	A3	A4	A5	A6
Type	F	F	F	F	F	F	F	F

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 (^oR)
HC	Heat generation coefficient (^oR/psi)
C1	Psi
C2	^oR
C3	Psi
C4	^oR
C5	$^1/_s$
C6	^oR
C7	$^1/_{psi}$
C8	^oR
C9	Psi
C10	^oR
C11	$^1/_{psi-s}$
C12	^oR
C13	$^1/_{psi}$
C14	^oR
C15	psi
C16	^oR
C17	$^1/_{psi-s}$
C18	^oR

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
C_1	*1/145	*1/145
C_2	—	*5/9
C_3	*1/145	*1/145
C_4	—	*5/9
C_5	—	—
C_6	—	*5/9
C_7	*145	*145
C_8	—	*5/9
C_9	*1/145	*1/145
C_{10}	—	*5/9
C_{11}	*145	*145
C_{12}	—	*5/9
C_{13}	*145	*145
C_{14}	—	*5/9
C_{15}	*1/145	*1/145
C_{16}	—	*5/9
C_{17}	*145	*145
C_{18}	—	*5/9
C0=HC	*145	*145*5/9
E	*1/145	*1/145
υ	—	—
T	—	*5/9

Remarks:

The kinematics associated with the model are discussed in references [Hill 1948, Bammann and Aifantis 1987, Bammann 1989]. The description below is taken nearly verbatim from Bammann [1989].

With the assumption of linear elasticity we can write,

$$\overset{o}{\sigma} = \lambda \, \mathrm{tr}\left(D^e\right)1 + 2\mu D^e$$

where the Cauchy stress σ is convected with the elastic spin W^e as,

$$\underline{\overset{o}{\sigma} = \dot{\sigma} - W^e\sigma + \sigma W^e}$$

This is equivalent to writing the constitutive model with respect to a set of directors whose direction is defined by the plastic deformation [Bammann and Aifantis 1987, Bammann and Johnson 1987]. Decomposing both the skew symmetric and symmetric parts of the velocity gradient into elastic and plastic parts we write for the elastic stretching D^e and the elastic spin W^e,

$$D^e = D - D^p - D^{th}, \quad W^e = W = W^p .$$

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,

$$D^{th} = \dot{A}\dot{T}1 .$$

The evolution of the internal variables α and κ are prescribed in a hardening minus recovery format as,

$$\overset{o}{\alpha} = h(T)D^p - \left[r_d(T)\left|D^p\right| + r_s(T)\right]|\alpha|\alpha,$$

$$\dot{\kappa} = H(T)D^p - \left[R_d(T)\left|D^p\right| - R_s(T)\right]\kappa^2$$

where h and H are the hardening moduli, r_s (T) and R_s (T) are scalar functions describing the diffusion controlled 'static' or 'thermal' recovery, and r_d (T) and R_d (T) are the functions describing dynamic recovery.

If we assume that $W^p = 0$, we recover the Jaumann stress rate which results in the prediction of an oscillatory shear stress response in simple shear when coupled with a Prager kinematic hardening assumption [Johnson and Bammann 1984]. Alternatively we can choose,

$$W^p = R^T \dot{U} U^{-1} R,$$

which recovers the Green-Naghdi rate of Cauchy stress and has been shown to be equivalent to Mandel's isoclinic state [Bammann and Aifantis 1987]. The model employing this rate allows a reasonable prediction of directional softening for some materials, but in general under-predicts the softening and does not accurately predict the axial stresses which occur in the torsion of the thin walled tube.

The final equation necessary to complete our description of high strain rate deformation is one which allows us to compute the temperature change during the deformation. In the absence of a coupled thermo-mechanical finite element code we assume adiabatic temperature change and follow the empirical assumption that 90 -95% of the plastic work is dissipated as heat. Hence,

$$\dot{T} = \frac{.9}{\rho C_v}\left(\sigma \cdot D^p\right),$$

where ρ is the density of the material and C_v the specific heat.

In terms of the input parameters the functions defined above become:

V(T)	=	C1 exp(-C2/T)
Y(T)	=	C3 exp(C4/T)
f(T)	=	C5 exp(-C6/T)
rd(T)	=	C7 exp(-C8/T)
h(T)	=	C9 exp(C10/T)
rs(T)	=	C11exp(-C12/T)
RD(T)	=	C13exp(-C14/T)
H(T)	=	C15exp(C16/T)
RS(T)	=	C17exp(-C18/T)

and the heat generation coefficient is

$$HC = \frac{.9}{\rho C_V}.$$

*MAT_BAMMAN_DAMAGE

This is Material Type 52. This is an extension of model 51 which includes the modeling of damage. See [Bamman, et.al., 1990].

Card 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					

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 (^oR)
HC	Heat generation coefficient (^oR/psi)
C1	Psi
C2	^oR
C3	Psi
C4	^oR
C5	1/s
C6	^oR
C7	1/psi
C8	^oR
C9	Psi
C10	^oR
C11	1/psi-s
C12	^oR

VARIABLE	DESCRIPTION
C13	^{1}/psi
C14	^{o}R
C15	psi
C16	^{o}R
C17	^{1}/psi-s
C18	^{o}R
A1	α_1, initial value of internal state variable 1
A2	α_2, initial value of internal state variable 2
A3	α_3, initial value of internal state variable 3
A4	α_4, initial value of internal state variable 4
A5	α_5, initial value of internal state variable 5
A6	α_6, initial value of internal state variable 6
N	Exponent in damage evolution
D0	Initial damage (porosity)
FS	Failure strain for erosion.

Remarks:

The evolution of the damage parameter, ϕ, is defined by [Bammann, et al. 1990]

$$\dot{\phi} = \beta\left[\frac{1}{(1-\phi)^N} - (1-\phi)\right]^{\left|D^p\right|}$$

in which

$$\beta = \sinh\left[\frac{2(2N-1)p}{(2N-1)\overline{\sigma}}\right]$$

where p is the pressure and $\overline{\sigma}$ is the effective stress.

*MAT_CLOSED_CELL_FOAM

This is Material Type 53. This allows the modeling of low density, closed cell polyurethane foam. It is for simulating impact limitors in automotive applications. The effect of the confined air pressure is included with the air being treated as an ideal gas. The general behavior is isotropic with uncoupled components of the stress tensor.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E	A	B	C	P0	PHI
Type	I	F	F	F	F	F	F	F

Card 2								
Variable	GAMA0	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 abcissa.

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}

$$\sigma_{ij}^{skt} = \sigma_{ij}^{sk} + E\,\dot{\varepsilon}_{ij}\,\Delta t$$

where E is Young's modulus. Since Poisson's ratio is zero, the update of each stress component is uncoupled and 2G=E where G is the shear modulus. The yield condition is applied to the principal skeletal stresses such that, if the magnitude of a principal trial stress component, σ_i^{skt}, exceeds the yield stress, σ_y, then

$$\sigma_i^{sk} = \min(\sigma_y, |\sigma_i^{skt}|)\frac{\sigma_i^{skt}}{|\sigma_i^{skt}|}$$

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 orthothropic materials, e.g., unidirectional layers in composite shell structures can be defined. Optionally, various types of failure can be specified following either the suggestions of [Chang and Chang, 1984] or [Tsai and Wu, 1981]. In addition special measures are taken for failure under compression. See [Matzenmiller and Schweizerhof, 1990]. This model is only valid for thin shell elements. 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	

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density
EA	E_a, Young's modulus - longitudinal direction
EB	E_b, Young's modulus - transverse direction
(EC)	E_c, Young's modulus - normal direction (not used)
PRBA	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

VARIABLE	DESCRIPTION
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 failuire 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 elrment 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 ≤ BETA ≤ 1.0)

Remarks:

The Chang/Chang criteria is given as follows:

for the tensile fiber mode,

$$\sigma_{aa} > 0 \quad then \quad e_f^2 = \left(\frac{\sigma_{aa}}{X_t}\right)^2 + \beta\left(\frac{\sigma_{ab}}{S_c}\right) - 1 \begin{cases} \geq 0 \; failed \\ < 0 \; elastic \end{cases},$$

$$E_a = E_b = G_{ab} = \nu_{ba} = \nu_{ab} = 0,$$

for the compressive fiber mode,

$$\sigma_{aa} < 0 \quad then \quad e_c^2 = \left(\frac{\sigma_{aa}}{X_c}\right)^2 - 1 \begin{cases} \geq 0 \; failed \\ < 0 \; elastic \end{cases},$$

$$E_a = \nu_{ba} = \nu_{ab} = 0.$$

for the tensile matrix mode,

$$\sigma_{bb} > 0 \quad then \quad e_m^2 = \left(\frac{\sigma_{bb}}{Y_t}\right)^2 + \left(\frac{\sigma_{ab}}{S_c}\right)^2 - 1 \begin{cases} \geq 0 \; failed \\ < 0 \; elastic \end{cases},$$

$$E_b = \nu_{ba} = 0. \quad \to G_{ab} = 0,$$

and for the compressive matrix mode,

$$\sigma_{bb} < 0 \quad then \quad e_d^2 = \left(\frac{\sigma_{bb}}{2S_c}\right)^2 + \left[\left(\frac{Y_c}{2S_c}\right)^2 - 1\right]\frac{\sigma_{bb}}{Y_c} + \left(\frac{\sigma_{ab}}{S_c}\right)^2 - 1 \begin{cases} \geq 0 \; failed \\ < 0 \; elastic \end{cases},$$

$$_b = \nu_{ba} = \nu_{ab} = 0. \quad \to G_{ab} = 0$$
$$X_c = 2Y_c \quad for\ 50\%\ fiber\ volume.$$

In the Tsai-Wu criteria the tensile and compressive fiber modes are treated as in the Chang-Chang criteria. The failure criterion for the tensile and compressive matrix mode is given as:

$$e_{md}^2 = \frac{\sigma_{bb}^2}{Y_c Y_t} + \left(\frac{\sigma_{ab}}{S_c}\right)^2 + \frac{(Y_c - Y_t)\sigma_{bb}}{Y_c Y_t} - 1 \begin{cases} \geq 0 \; failed \\ < 0 \; elastic \end{cases}$$

For β =1 we get the original criterion of Hashin [1980] in the tensile fiber mode. For β =0 we get the maximum stress criterion which is found to compare better to experiments.

Failure can occur in any of four different ways:

1. If DFAILT is zero, failure occurs if the Chang-Chang failure criterion is satisfied in the tensile fiber mode.

2. If DFAILT is greater than zero, failure occurs if the tensile fiber strain is greater than DFAILT or less than DFAILC.

3. If EFS is greater than zero, failure occurs if the effecive strain is greater than EFS.

4. If TFAIL is greater than zero, failure occurs according to the element timestep as described in the definition of TFAIL above.

When failure has occurred in all the composite layers (through-thickness integration points), the element is deleted. Elements which share nodes with the deleted element become "crashfront" elements and can have their strengths reduced by using the SOFT parameter with TFAIL greater than zero.

Information about the status in each layer (integration point) and element can be plotted using additional integration point variables. The number of additional integration point variables for shells written to the LS-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):

<table>
<tr><th>History Variable</th><th>Description</th><th>Value</th><th>LS-TAURUS Component</th></tr>
<tr><td>1. $ef(i)$</td><td>tensile fiber mode</td><td rowspan="5">1 - elastic
0 - failed</td><td>81</td></tr>
<tr><td>2. $ec(i)$</td><td>compressive fiber mode</td><td>82</td></tr>
<tr><td>3. $em(i)$</td><td>tensile matrix mode</td><td>83</td></tr>
<tr><td>4. $ed(i)$</td><td>compressive matrix mode</td><td>84</td></tr>
<tr><td>5. efail</td><td>max[ef(ip)]</td><td>85</td></tr>
<tr><td>6. dam</td><td>damage parameter</td><td>-1 - element intact
10^{-8} - element in crashfront
+1 - element failed</td><td>86</td></tr>
</table>

These variables can be plotted in LS-TAURUS as element components 81, 82, ..., 80+ NEIPS. The following components, defined by the sum of failure indicators over all through-thickness integration points, are stored as element component 7 instead of the effective plastic strain.:

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

Examples:

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

LS-TAURUS commands in Phase I: *intg 3 frin 81*

LS-TAURUS commands in Phase II: *etime 81 n e1 e2 ... en*

b) Sum of failure indicator of compressive fiber mode:

LS-TAURUS commands in Phase I: *intg 2 frin 7*

LS-TAURUS commands in Phase II: *etime 7 3 e e e* (with *e* ... element number)

c) Visualization of crashfront via *dam* parameter

LS-TAURUS commands: *frin 86*

*MAT_LOW_DENSITY_FOAM

This is Material Type 57. It is mainly for modeling highly compressible low density foams. Its main applications are for seat cushions and padding on the Side Impact Dummies (SID). Optionally, a tension cut-off failure can be defined. Also, see the notes below.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E	LCID	TC	HU	BETA	DAMP
Type	I	F	F	F	F	F	F	F
Default	---	---	---	---	1.E+20	1.		
Remarks	---	---	---	---	---	3	1	---

Card 2								
Variable	SHAPE	FAIL	BVFLAG	ED	BETA1	KCON	REF	
Type	F	F	F	F	F	F	F	
Default	1.0	0.0	0.0	0.0	0.0	0.0	0.0	
Remarks	3	---	2					

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density
E	Young’s modulus
LCID	Load curve ID, see *DEFINE_CURVE, for nominal stress versus strain.
TC	Tension cut-off stress

VARIABLE	DESCRIPTION
HU	Hysteretic unloading factor between 0 and 1 (default=1, i.e., no energy dissipation), see also Figure 20.16.
BETA	β, decay constant to model creep in unloading
DAMP	Viscous coefficient (.05< recommended value <.50) to model damping effects.
SHAPE	Shape factor for unloading. Active for nonzero values of the hysteretic unloading factor. Values less than one reduces the energy dissipation and greater than one increases dissipation, see also Figure 20.16.
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. 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:

$$\varepsilon_i = \lambda_i - 1$$

The stretch ratios are found by solving for the eigenvalues of the left stretch tensor, V_{ij}, which is obtained via a polar decomposition of the deformation gradient matrix, F_{ij}. Recall that,

$$F_{ij} = R_{ik}U_{kj} = V_{ik}R_{kj}$$

The update of V_{ij} follows the numerically stable approach of [Taylor and Flanagan 1989]. After solving for the principal stretches, we compute the elongations and, if the elongations are compressive, the corresponding values of the nominal stresses, τ_i, are interpolated. If the elongations are tensile, the nominal stresses are given by

$$\tau_i = E\varepsilon_i$$

and the Cauchy stresses in the principal system become

$$\sigma_i = \frac{\tau_i}{\lambda_j \lambda_k}$$

The stresses can now be transformed back into the global system for the nodal force calculations.

Additional Remarks:

1. When hysteretic unloading is used the reloading will follow the unloading curve if the decay constant, β, is set to zero. If β is nonzero the decay to the original loading curve is governed by the expression:

$$1.-e^{-\beta t}$$

2. The bulk viscosity, which generates a rate dependent pressure, may cause an unexpected volumetric response and, consequently, it is optional with this model.

3. The hysteretic unloading factor results in the unloading curve to lie beneath the loading curve as shown in Figure 20.16. This unloading provide energy dissipation which is reasonable in certains kinds of foam.

Rate effects are accounted for through linear viscoelasticity by a convolution integral of the form

$$\sigma_{ij}^r = \int_0^t g_{ijkl}(t-\tau) \frac{\partial \varepsilon_{kl}}{\partial \tau} d\tau$$

where $g_{ijkl}(t-\tau)$ is the relaxation function. The stress tensor, σ_{ij}^r, augments the stresses determined from the foam, σ_{ij}^f; consequently, the final stress, σ_{ij}, is taken as the summation of the two contributions:

$$\sigma_{ij} = \sigma_{ij}^f + \sigma_{ij}^r.$$

Since we wish to include only simple rate effects, the relaxation function is represented by one term from the Prony series:

$$g(t) = \alpha_0 + \sum_{m=1}^{N} \alpha_m e^{-\beta t}$$

given by,

$$g(t) = E_d e^{-\beta_1 t}$$

This model is effectively a Maxwell fluid which consists of a damper and spring in series. We characterize this in the input by a Young's modulus, E_d, and decay constant, β_1. The formulation is performed in the local system of principal stretches where only the principal values of stress are computed and triaxial coupling is avoided. Consequently, the one-dimensional nature of this foam material is unaffected by this addition of rate effects. The addition of rate effects necessitates twelve additional history variables per integration point. The cost and memory overhead of this model comes primarily from the need to "remember" the local system of principal stretches.

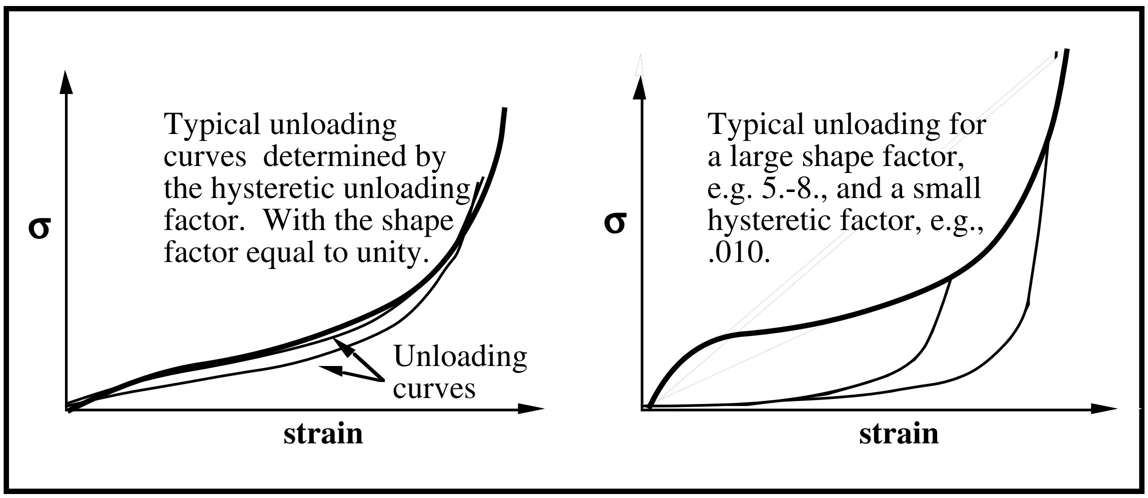


Figure 20.16. Behavior of the low density urethane foam model.

*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			

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density
EA	E_a, Young's modulus - longitudinal direction
EB	E_b, Young's modulus - transverse direction
(EC)	E_c, Young's modulus - normal direction (not used)
PRBA	ν_{ba}, Poisson's ratio ba
TAU1	$\tau 1$, stress limit of the first slightly nonlinear part of the of the shear stress versus shear strain curve. The values $\tau 1$ and $\gamma 1$ are used to define a curve of shear stress versus shear strain. These values are input if FS, defined below, is set to a value of -1.
GAMMA1	$\gamma 1$, strain limit of the first slightly nonlinear part of the of the shear stress versus shear strain curve.

VARIABLE	DESCRIPTION
GAB	G_{ab}, shear modulus ab
GBC	G_{bc}, shear modulus bc
GCA	G_{ca}, shear modulus ca
SLIMT1	Factor to determine the minimum stress limit after stress maximum (fiber tension).
SLIMC1	Factor to determine the minimum stress limit after stress maximum (fiber compression).
SLIMT2	Factor to determine the minimum stress limit after stress maximum (matrix tension).
SLIMC2	Factor to determine the minimum stress limit after stress maximum (matrix compression).
SLIMS	Factor to determin the minimum stress limit after stress maximum. (shear).
AOPT	Material axes option (see MAT_*OPTION* TROPIC_ELASTIC for a more complete description): EQ. 0.0: locally orthotropic with material axes determined by element nodes 1, 2, and 4, as with *DEFINE_ COORDINATE_NODES. EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR. EQ. 3.0: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle (BETA) from a line in the plane of the element defined by the cross product of the vector **v** with the element normal.
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 appropiate 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,

$$\sigma_{\min} = SLIMxx \cdot strength,$$

thus, the damage value is slightly modified such that elastoplastic like behavior is achieved with the threshold stress. As a factor for *SLIMxx* a number between 0.0 and 1.0 is possible. With a factor of 1.0, the stress remains at a maximum value identical to the strength, which is similar to ideal elastoplastic behavior. For tensile failure a small value for *SLIMTx* is often reasonable; however, for compression *SLIMCx = 1.0* is preferred. This is also valid for the corresponding shear value. If *SLIMxx* is smaller than 1.0 then localization can be observed depending on the total behavior of the lay-up. If the user is intentionally using *SLIMxx < 1.0,* it is generally recommended to avoid a drop to zero and set the value to something in between 0.05 and 0.10. Then elastoplastic behaviour is achieved in the limit which often leads to less numerical problems. Defaults for SLIMXX = 1.0E-8.

The crashfront-algorithm is started if and only if a value for TSIZE (time step size, with element elimination after the actual time step becomes smaller than TSIZE) is input .

The damage parameters can be written to the postprocessing database for each integration point as the first three additional element variables and can be visualized.

Material models with FS=1 or FS=-1 are favorable for complete laminates and fabrics, as all directions are treated in a similar fashion.

For material model FS=1 an interaction between normal stresses and the shear stresses is assumed for the evolution of damage in the a and b-directions. For the shear damage is always the maximum value of the damage from the criterion in a or b-direction is taken.

For material model FS=-1 it is assumed that the damage evolution is independent of any of the other stresses. A coupling is only present via the elastic material parameters and the complete structure.

In tensile and compression directions and in a as well as in b- direction different failure surfaces can be assumed. The damage values, however, increase only also when the loading direction changes.

Special control of shear behavior of fabrics

For fabric materials a nonlinear stress strain curve for the shear part for failure surface FS=-1 can be assumed as given below. This is not possible for other values of FS.

The curve, shown in Figure 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.

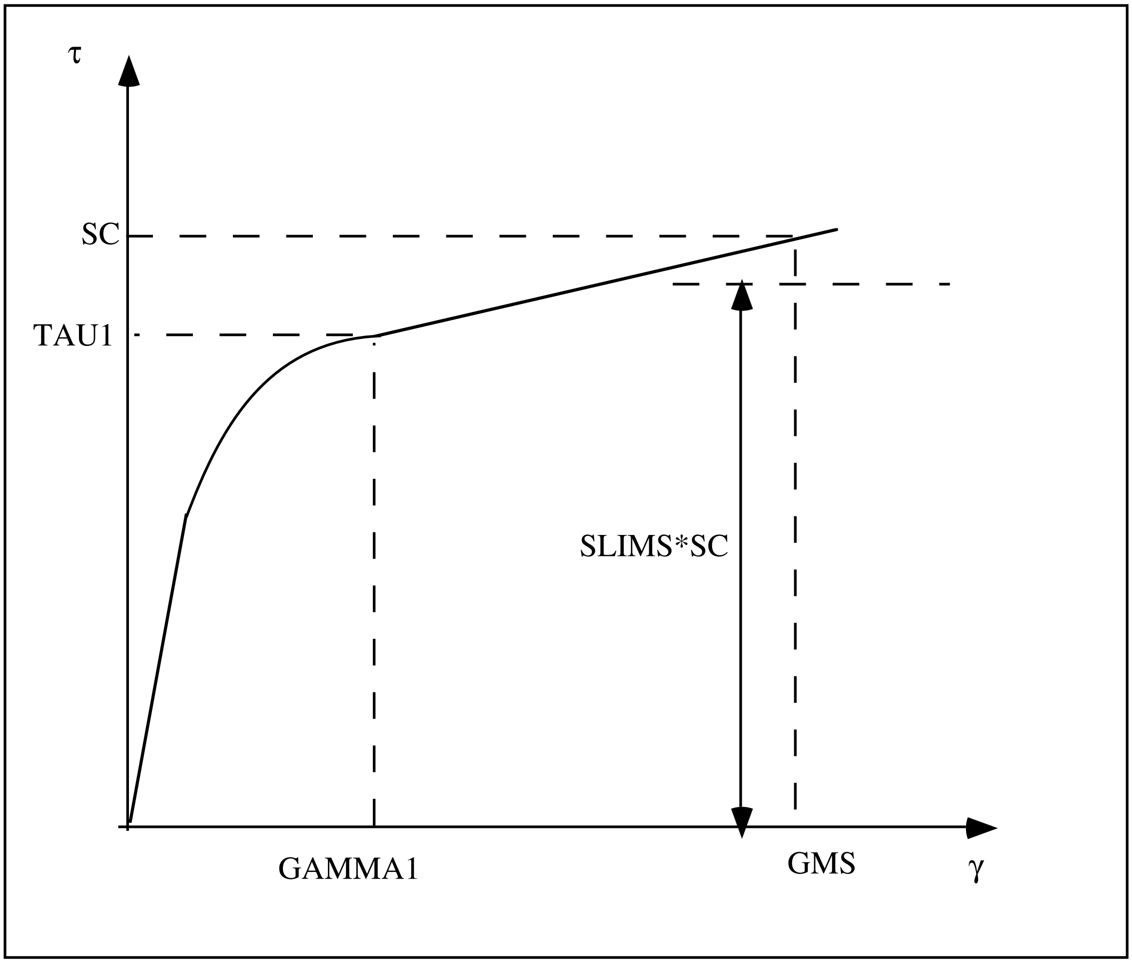


Figure 20.17. Stress-strain diagram for shear.

*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 **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	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 time. (Optional).
T1, T2,...TN	Temperatures, define up to 8 values
PR1, PR2,...PRN	Poisson's ratios for the temperatures T_i
V1, V2,...VN	Corresponding viscosity coefficients (define only one if not varying with temperature).
E1, E2,...EN	Corresponding Young's moduli coefficients (define only one if not varying with temperature).
ALPHA....	Corresponding thermal expansion coefficients

Remarks:

Volumetric behavior is treated as linear elastic. The deviatoric strain rate is considered to be the sum of elastic and viscous strain rates:

$$\dot{\underset{\sim}{\varepsilon}}'_{total} = \dot{\underset{\sim}{\varepsilon}}'_{elastic} + \dot{\underset{\sim}{\varepsilon}}'_{viscous} = \frac{\underset{\sim}{\sigma}'}{2G} + \frac{\underset{\sim}{\sigma}'}{2v}$$

where G is the elastic shear modulus, v is the viscosity coefficient, and ~ indicates a tensor. The stress increment over one timestep dt is

$$d\underset{\sim}{\sigma}' = 2G\,\dot{\underset{\sim}{\varepsilon}}'_{total}\,dt - \frac{G}{\upsilon}dt\,\underset{\sim}{\sigma}'$$

The stress before the update is used for $\underset{\sim}{\sigma}'$. For shell elements the through-thickness strain rate is calculated as follows.

$$d\sigma_{33} = 0 = K\left(\dot{\varepsilon}_{11} + \dot{\varepsilon}_{22} + \dot{\varepsilon}_{33}\right)dt + 2G\,\dot{\varepsilon}'_{33}\,dt - \frac{G}{\upsilon}dt\sigma'_{33}$$

where the subscript ij = 33 denotes the through-thickness direction and K is the elastic bulk modulus. This leads to:

$$\dot{\varepsilon}_{33} = -a\left(\dot{\varepsilon}_{11} + \dot{\varepsilon}_{22}\right) + bp$$

$$a = \frac{\left(K - \frac{2}{3}G\right)}{\left(K + \frac{4}{3}G\right)}$$

$$b = \frac{Gdt}{\upsilon\left(K + \frac{4}{3}G\right)}$$

in which p is the pressure defined as the negative of the hydrostatic stress.

The variation of viscosity with temperature can be defined in any one of the 3 ways.

(i) Constant, $V = V_0$ Do not define constants, A, B, and C or the piecewise curve.(leave card 4 blank)

(ii) $V = V_0 \times 10^{**}$ (A/(T-B) + C)

(iii) Piecewise curve: define the variation of viscosity with temperature.

Note: Viscosity is inactive during dynamic relaxation.

*MAT_KELVIN-MAXWELL_VISCOELASTIC

This is Material Type 61. It is a classical Kelvin-Maxwell model for modelling viscoelastic bodies, e.g., foams. Only valid for solid elements. See also notes below.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	BULK	G0	GI	DC	FO	SO
Type	I	F	F	F	F	F	F	F
Default	none	none	none	none	none	0.0	0.0	0.0

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density
BULK	Bulk modulus (elastic)
G0	Short-time shear modulus, G_0
GI	Long-time (infinite) shear modulus, G_∞
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:

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

A Jaumann rate formulation is used

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

where the prime denotes the deviatoric part of the stress rate, $\overset{\nabla}{\sigma}_{ij}$, and the strain rate D_{ij}. For the Kelvin model the stress evolution equation is defined as:

$$\dot{s}_{ij} + \frac{1}{\tau} s_{ij} = \left(1+\delta_{ij}\right) G_0\, \dot{e}_{ij} + \left(1+\delta_{ij}\right) \frac{G_\infty}{\tau} \dot{e}_{ij}$$

The strain data as written to the LS-DYNA database may be used to predict damage, see [Bandak 1991].

*MAT_VISCOUS_FOAM

This is Material Type 62. It was written to represent the Confor Foam on the ribs of EuroSID side impact dummy. It is only valid for solid elements, mainly under compressive loading.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E1	N1	V2	E2	N2	PR
Type	I	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density
E1	Initial Young's modulus (E_1)
N1	Exponent in power law for Young's modulus (n_1)
V2	Viscous coefficient (V_2)
E2	Elastic modulus for viscosity (E_2), see notes below.
N2	Exponent in power law for viscosity (n_2)
PR	Poisson's ratio, ν

Remarks:

The model consists of a nonlinear elastic stiffness in parallel with a viscous damper. The elastic stiffness is intended to limit total crush while the viscosity absorbs energy. The stiffness E_2 exists to prevent timestep problems. It is used for time step calculations a long as E_1^t is smaller than E_2. It has to be carefully chosen to take into account the stiffening effects of the viscosity. Both E_1 and V_2 are nonlinear with crush as follows:

$$E_1^t = E_1\left(V^{-n_1}\right)$$

$$V_2^t = V_2\left(abs(1-V)\right)^{n_2}$$

where viscosity generates a shear stress given by

$$\tau = V_2 \dot{\gamma}$$

$\dot{\gamma}$ is the engineering shear strain rate, and V is the relative volume defined by the ratio of the current to initial volume. Typical values are (units of N, mm, s)

E_1=0.0036	n_1=4.0	V_2 =0.0015
E_2=100.0	n_2=0.2	ν =0.05

*MAT_CRUSHABLE_FOAM

This is Material Type 63 which is dedicated to modeling crushable foam with optional damping and tension cutoff. Unloading is fully elastic. Tension is treated as completely elastic-plastic.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	LCID	TSC	DAMP	
Type	I	F	F	F	F	F	F	

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density
E	Young's modulus
PR	Poisson's ratio
LCID	Load curve ID defining yield stress versus volumetric strain, γ, see Figure 20.18.
TSC	Tensile stress cutoff
DAMP	Rate senitivity via damping coefficient (.05<recommended value<.50).

Remarks:

The volumetric strain is defined in terms of the relative volume, V, as:

$$\gamma = 1.-V$$

The relative volume is defined as the ratio of the current to the initial volume.

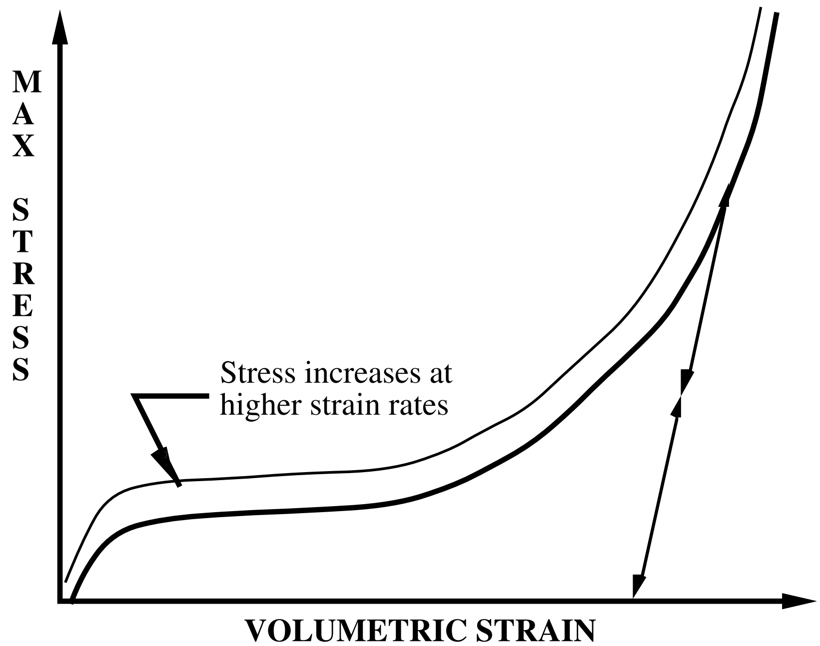


Figure 20.18. Behavior of strain rate sensitive crushable foam. Unloading is elastic to the tension cutoff. Subsequent reloading follows the unloading curve.

*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.

VARIABLE	DESCRIPTION
N	Strain rate sensitivity coefficient, n. If n<0 the absolute value of n is taken as the load curve number that defines n as a function of effective plastic strain.
E0	Initial strain rate (default = 0.0002)
VP	Formulation for rate effects: EQ.0.0: Scale yield stress (default) EQ.1.0: Viscoplastic formulation
EPS0	Factor to normalize strain rate EQ.1.0: Time units of seconds (default) EQ.1.E-3: Time units of milliseconds EQ.1.E-6: Time units of microseconds

Remarks:

This material model follows a constitutive relationship of the form:

$$\sigma = k\varepsilon^m \dot{\varepsilon}^n$$

where σ is the yield stress, ε is the effective plastic strain, $\dot{\varepsilon}$ is the normalized effective plastic strain rate, and the constants *k*, *m*, and *n* can be expressed as functions of effective plastic strain or can be constant with respect to the plastic strain. The case of no strain hardening can be obtained by setting the exponent of the plastic strain equal to a very small positive value, i.e. 0.0001.

This model can be combined with the superplastic forming input to control the magnitude of the pressure in the pressure boundary conditions in order to limit the effective plastic strain rate so that it does not exceed a maximum value at any integration point within the model.

A fully viscoplastic formulation is optional. An additional cost is incurred but the improvement is results can be dramatic.

*MAT_MODIFIED_ZERILLI_ARMSTRONG

This is Material Type 65 which is a rate and temperature sensitive plasticity model which is sometimes preferred in ordnance design calculations.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	G	E0	N	TROOM	PC	SPALL
Type	I	F	F	F	F	F	F	F

Card 2	1	2	3	4	5	6	7	8
Variable	C1	C2	C3	C4	C5	C6	EFAIL	VP
Type	F	F	F	F	F	F	F	F

Card 3	1	2	3	4	5	6	7	8
Variable	B1	B2	B3	G1	G2	G3	G4	BULK
Type	F	F	F	F	F	F	F	

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density
G	Shear modulus
E0	$\dot{\varepsilon}_0$, factor to normalize strain rate
N	n, exponent for bcc metal
TROOM	T_r, room temperature

VARIABLE	DESCRIPTION
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),

$$\sigma = C_1 + \left\{ C_2 \left(\varepsilon^p \right)^{1/2} \left[e^{\left(-C_3 + C_4 \ln\left(\dot{\varepsilon}^*\right)\right)T} \right] + C_5 \right\} \left(\frac{\mu(T)}{\mu(293)} \right)$$

ε^p = effective plastic strain

$\dot{\varepsilon}^* = \dfrac{\dot{\varepsilon}}{\dot{\varepsilon}_0}$ effective plastic strain rate where $\dot{\varepsilon}_0$ = 1, 1e-3, 1e-6 for time units of seconds, milliseconds, and microseconds, respectively.

For bcc metals (n>0),

$$\sigma = C_1 + C_2 e^{\left(-C_3 + C_4 \ln\left(\dot{\varepsilon}^*\right)\right)T} + \left[C_5 \left(\varepsilon^p \right)^n + C_6 \right] \left(\frac{\mu(T)}{\mu(293)} \right)$$

where

$$\left(\frac{\mu(T)}{\mu(293)} \right) = B_1 + B_2 T + B_3 T^2 \quad .$$

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 is results can be dramatic.

*MAT_CONCRETE_DAMAGE

This is Material Type 72. This model has been used to analyze buried steel reinforced concrete structures subjected to implusive 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	λ1	λ2	λ3	λ4	λ5	λ6	λ7	λ8
Type	F	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	none	none

Card 6

Variable	λ9	λ10	λ11	λ12	λ13			
Type	F	F	F	F	F			
Default	none	none	none	none	none			

Card 7	1	2	3	4	5	6	7	8
Variable	η1	η2	η3	η4	η5	η6	η7	η8
Type	F	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	none	none

Card 8

Variable	η9	η10	η11	η12	η13			
Type	F	F	F	F	F			
Default	none	none	none	none	none			

VARIABLE	DESCRIPTION
MID	Material identification. A unique number 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.

VARIABLE	DESCRIPTION
SIGY	Initial yield stress.
ETAN	Tangent modulus/plastic hardening modulus.
LCP	Load curve ID giving rate sensitivity for principal material, see *DEFINE_CURVE.
LCR	Load curve ID giving rate sensitivity for reinforcement, see *DEFINE_CURVE.
$\lambda 1 - \lambda 13$	Tabulated damage function
$\eta 1 - \eta 13$	Tabulated scale factor.

Remarks:

Cohesion for failed material $a_{0f} = 0.0$

b3 must be positive or zero.

$\lambda_n < \lambda_{n+1}$. The first point must be zero.

*MAT_LOW_DENSITY_VISCOUS_FOAM

This is Material Type 73. It is mainly for Modeling Low Density Urethane Foam with high compressibility and with rate sensitivity which can be characterized by a relaxation curve. Its main applications are for seat cushions, padding on the Side Impact Dummies (SID), bumpers, and interior foams. Optionally, a tension cut-off failure can be defined. Also, see the notes below and the 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					

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
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.
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.

VARIABLE	DESCRIPTION
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 curveIf constants $\beta\iota$ 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 $\beta\kappa_2$, 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 are 3 are recommended, since each term used adds significantly to the cost. Caution should be exercised when taking the results from the fit. Preferably, all generated coefficients should be positive. Negative values may lead to unstable results. Once a satisfactory fit has been achieved it is recommended that the coefficients which are written into the output file be input in future runs.
GI	Optional shear relaxation modulus for the ith term
BETAI	Optional decay constant if ith term
REF	Use reference geometry to initialize the stress tensor. The reference 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 viscoelastic foam model is available to model highly compressible viscous foams. The hyperelastic formulation of this models follows that of material 57.

Rate effects are accounted for through linear viscoelasticity by a convolution integral of the form

$$\sigma_{ij}^r = \int_0^t g_{ijkl}(t-\tau)\frac{\partial \varepsilon_{kl}}{\partial \tau}d\tau$$

where $g_{ijkl}(t-\tau)$ is the relaxation function. The stress tensor, σ_{ij}^r, augments the stresses determined from the foam, σ_{ij}^{f}; consequently, the final stress, σ_{ij}, is taken as the summation of the two contributions:

$$\sigma_{ij} = \sigma_{ij}^{f} + \sigma_{ij}^{r}.$$

Since we wish to include only simple rate effects, the relaxation function is represented by up to six terms of the Prony series:

$$g(t) = \alpha_0 + \sum_{m=1}^{N} \alpha_m e^{-\beta t}$$

This model is effectively a Maxwell fluid which consists of a dampers and springs in series. The formulation is performed in the local system of principal stretches where only the principal values of stress are computed and triaxial coupling is avoided. Consequently, the one-dimensional nature of this foam material is unaffected by this addition of rate effects. The addition of rate effects necessitates 42 additional history variables per integration point. The cost and memory overhead of this model comes primarily from the need to “remember” the local system of principal stretches and the evaluation of the viscous stress components.

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 certains kinds of foam.

*MAT_BILKHU/DUBOIS_FOAM

This is Material Type 75. This model is for the simulation of isotropic crushable forms. Uniaxial and triaxial test data have to be used. For the elastic response, the Poisson ratio is set to zero.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	YM	LCPY	LCUYS	VC		
Type	I	F	F	F	F	F		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density
YM	Young's modulus (E)
LCPY	Load curve ID giving pressure for plastic yielding versus volumetric strain, see Figure 20.24.
LCUYS	Load curve ID giving unixial yield stress versus volumetric strain, see Figure 20.24.
VC	Viscous damping coefficient (.05<recommended value<.50).

Remarks:

The logarithmic volumetric strain is defined in terms of the relative volume, V, as:

$$\gamma = -\ln(V)$$

In defining the curves the stress and strain pairs should be positive values starting with a volumetric strain value of zero.

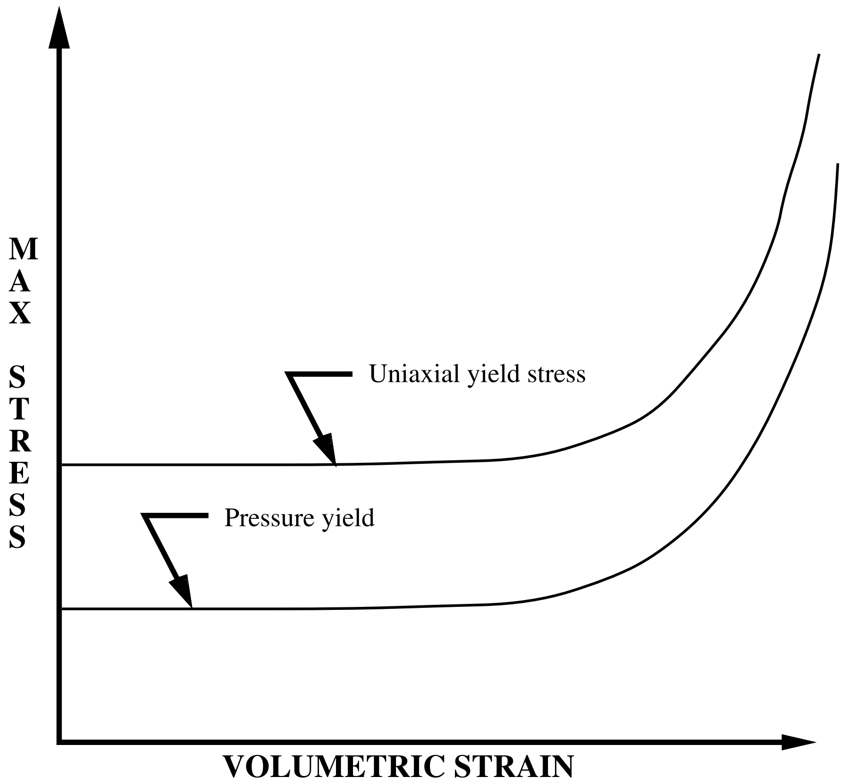


Figure 20.24. Behavior of crushable foam. Unloading is elastic.

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.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	BULK					
Type	I	F	F					

Insert a blank card here if constants are defined on cards 3,4,... below.

Card 2	1	2	3	4	5	6	7	8
Variable	LCID	NT	BSTART	TRAMP	LCIDK	NTK	BSTARTK	TRAMPK
Type	F	I	F	F	F	I	F	F

Card Format for viscoelastic constants. Up to 6 cards may be input. A keyword card (with a "*" in column 1) terminates this input if less than 6 cards are used. These cards are not needed if relaxation data is defined. The number of terms for the shear behavior may differ from that for the bulk behavior: simply insert zero if a term is not included.

Optional 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.
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 modulii. 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 accounted through linear viscoelasticity by a convolution integral of the form:

$$\sigma_{ij} = \int_0^t g_{ijkl}(t-\tau)\frac{\partial \varepsilon_{kl}}{\partial \tau}d\tau$$

where $g_{ijkl}(t-\tau)$ is the relaxation 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 modulii, G_i, and decay constants, β_i. An arbitrary number of terms, up to 6, may be used when applying the viscoelastic model.

For volumetric relaxation, the relaxation function is also represented by the Prony series in terms of bulk modulii:

$$k(t) = \sum_{m=1}^{N} K_m e^{-\beta_{k_m} t}$$

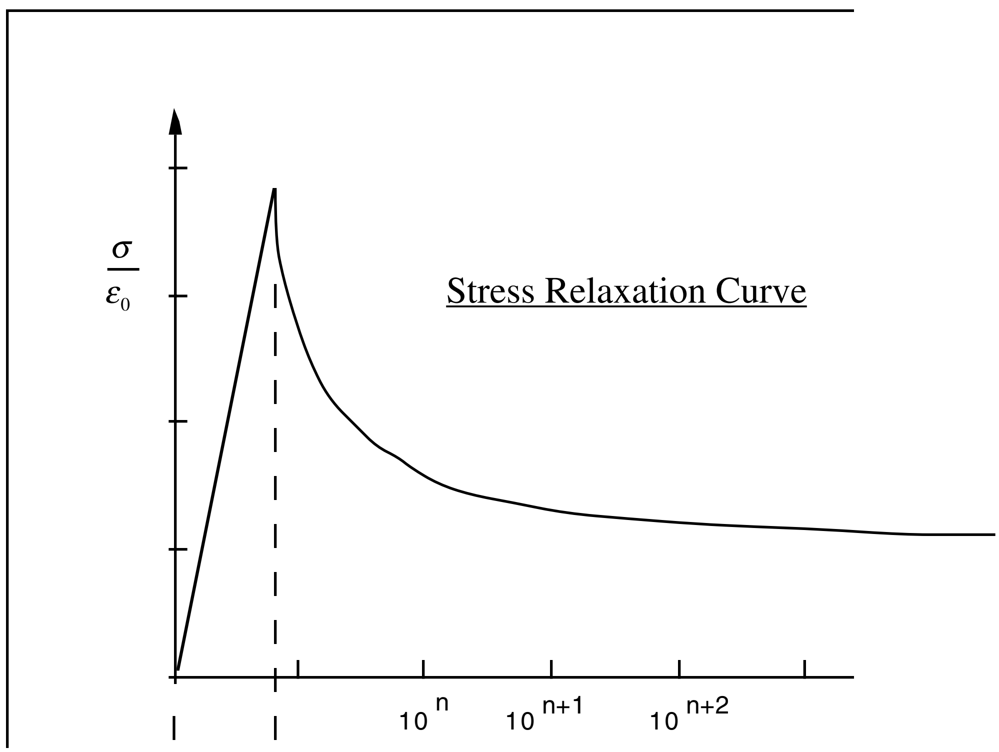


Figure 20.25. Relaxation curve. This curve defines stress versus time where time is defined on a logarithmic scale. For best results, the points defined in the load curve should be equally spaced on the logarithmic scale. Furthermore, the load curve should be smooth and defined in the positive quadrant. 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			
Type	I	F	F	I	I			

Card 2 if N > 0, a least squares fit is computed from unixial 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		

Card Format for Viscoelastic Constants. Up to 6 cards may be input. A keyword card (with a "*" in column 1) terminates this input if less than 6 cards are used.

Optional Cards	1	2	3	4	5	6	7	8
Variable	GI	BETAI						
Type	F	F						

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density
PR	Poissons ratio (>.49 is recommended, smaller values may not work and should not be used).
N	Number of constants to solve for: EQ.1: Solve for C10 and C01 EQ.2: Solve for C10, C01, C11, C20, and C02 EQ.3: Solve for C10, C01, C11, C20, C02, and C30
NV	Number of Prony series terms in fit. If zero, the default is 6. Currently, the maximum number is set to 6. Values less than 6, possibly 3-5 are recommended, since each term used adds significantly to the cost. Caution should be exercised when taking the results from the fit. Preferably, all generated coefficients should be positive. Negative values may lead to unstable results. Once a satisfactory fit has been achieved it is recommended that the coefficients which are written into the output file be input in future runs.

f N>0 test information from a uniaxial test are used:

VARIABLE	DESCRIPTION
SGL	Specimen gauge length
SW	Specimen width
ST	Specimen thickness
LCID1	Load curve ID giving the force versus actual change in the gauge length
DATA	Type of experimental data. EQ.0.0: uniaxial data (Only option for this model)

VARIABLE	DESCRIPTION
LCID2	Load curve ID of relaxation curve If constants $\beta\iota$ are determined via a least squares fit. This relaxation curve is shown in Figure 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	C_{10}
C01	C_{01}
C11	C_{11}
C20	C_{20}
C02	C_{02}
C30	C_{30}
GI	Optional shear relaxation modulus for the ith term
BETAI	Optional decay constant if ith term

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 invarients are modified as shown. This procedure is described in more detail by Sussman and Bathe [1987].

Rate effects are taken into account through linear viscoelasticity by a convolution integral of the form:

$$\sigma_{ij} = \int_0^t g_{ijkl}(t-\tau)\frac{\partial \varepsilon_{kl}}{\partial \tau}d\tau$$

or in terms of the second Piola-Kirchhoff stress, S_{ij}, and Green's strain tensor, E_{ij},

$$S_{ij} = \int_0^t G_{ijkl}(t-\tau)\frac{\partial E_{kl}}{\partial \tau}d\tau$$

where $g_{ijkl}(t-\tau)$ and $G_{ijkl}(t-\tau)$ are the relaxation functions for the different stress measures. This stress is added to the stress tensor determined from the strain energy functional.

If we wish to include only simple rate effects, the relaxation function is represented by six terms from the Prony series:

$$g(t) = \alpha_0 + \sum_{m=1}^{N} \alpha_m e^{-\beta t}$$

given by,

$$g(t) = \sum_{i=1}^{n} G_i e^{-\beta_i t}$$

This model is effectively a Maxwell fluid which consists of a dampers and springs in series. We characterize this in the input by shear modulii, G_i, and decay constants, β_i. The viscoelastic behavior is optional and an arbitrary number of terms may be used.

The Mooney-Rivlin rubber model (model 27) is obtained by specifying n=1. In spite of the differences in formulations with Model 27, we find that the results obtained with this model are nearly identical with those of Material 27 as long as large values of Poisson's ratio are used.

*MAT_OGDEN_RUBBER

This is also Material Type 77. This material model provides the Ogden [1984] rubber model combined optionally with linear viscoelasticity as outlined by [Christensen 1980].

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	PR	N	NV			
Type	I	F	F	I	I			

Card 2 if N > 0, a least squares fit is computed from unixial 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 Cards	1	2	3	4	5	6	7	8
Variable	GI	BETAI						
Type	F	F						

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density
PR	Poissons ratio (≥ .49 is recommended, smaller values may not work and should not be used).
N	Order of fit to the Ogden model, (currently <5, 2 generally works okay). The constants generated during the fit are printed in the output file and can be directly input in future runs, thereby, saving the cost of performing the nonlinear fit.
NV	Number of terms in fit. If zero, the default is 6. Currently, the maximum number is set to 6. Values less than 6, possibly 3-5 are recommended, since each term used adds significantly to the cost. Caution should be exercised when taking the results from the fit. Preferably, all generated coefficients should be positive. Negative values may lead to unstable results. Once a satisfactory fit has been achieved it is recommended that the coefficients which are written into the output file be input in future runs.

if N>0 test information from a uniaxial test are used:

VARIABLE	DESCRIPTION
SGL	Specimen gauge length
SW	Specimen width

VARIABLE	DESCRIPTION
ST	Specimen thickness
LCID1	Load curve ID giving the force versus actual change in the gauge length
DATA	Type of experimental data. EQ.0.0: uniaxial data EQ.1.0: biaxial data
LCID2	Load curve ID of relaxation curve If constants 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 MUi and ALPHAi have to be defined:
MUi	μ_i, the ith shear modulus, i varies up to 8. See discussion below.
ALPHAi	α_i, the ith exponent, i varies up to 8. See discussion below.
GI	Optional shear relaxation modulus for the ith term
BETAI	Optional decay constant if ith term

Remarks:

Rubber is generally considered to be fully incompressible since the bulk modulus greatly exceeds the shear modulus in magnitude. To model the rubber as an unconstrained material a hydrostatic work term is included in the strain energy functional which is function of the relative volume, J, [Ogden, 1984]:

$$W^* = \sum_{i=1}^{3}\sum_{j=1}^{n}\frac{\mu_j}{\alpha_j}\left(\lambda_i^{*\alpha_j}-1\right)+\frac{1}{2}K(J-1)^2$$

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:

$$\sigma_{ij} = \int_0^t g_{ijkl}(t-\tau)\frac{\partial \varepsilon_{kl}}{\partial \tau} d\tau$$

or in terms of the second Piola-Kirchhoff stress, S_{ij}, and Green's strain tensor, E_{ij},

$$S_{ij} = \int_0^t G_{ijkl}(t-\tau)\frac{\partial E_{kl}}{\partial \tau} d\tau$$

where $g_{ijkl}(t-\tau)$ and $G_{ijkl}(t-\tau)$ are the relaxation functions for the different stress measures. This stress is added to the stress tensor determined from the strain energy functional.

If we wish to include only simple rate effects, the relaxation function is represented by six terms from the Prony series:

$$g(t) = \alpha_0 + \sum_{m=1}^{N} \alpha_m e^{-\beta t}$$

given by,

$$g(t) = \sum_{i=1}^{n} G_i e^{-\beta_i t}$$

This model is effectively a Maxwell fluid which consists of a dampers and springs in series. We characterize this in the input by shear modulii, G_i, and decay constants, β_i. The viscoelastic behavior is optional and an arbitrary number of terms may be used.

The Mooney-Rivlin rubber model (model 27) is obtained by specifying n=1. In spite of the differences in formulations with Model 27, we find that the results obtained with this model are nearly identical with those of Material 27 as long as large values of Poisson's ratio are used.

*MAT_SOIL_CONCRETE

This is Material Type 78. This model permits concrete and soil to be efficiently modelled. See the explanations below.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	G	K	LCPV	LCYP	LCFP	LCRP
Type	I	F	F	F	F	F	F	F

Card 2	1	2	3	4	5	6	7	8
Variable	PC	OUT	B	FAIL				
Type	F	F	F	F				

VARIABLE	DESCRIPTION
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 logrithm of the relative volume.
LCYP	Load curve ID for yield versus pressure: GT.0: von Mises stress versus pressure, LT.0: Second stress invariant, J_2, versus pressure. This curve must be defined.
LCFP	Load curve ID for plastic strain at which fracture begins versus pressure. This load curve ID must be defined if B>0.0.

VARIABLE	DESCRIPTION
LCRP	Load curve ID for plastic strain at which residual strength is reached versus pressure. This load curve ID must be defined if B>0.0.
PC	Pressure cutoff for tensile fracture
OUT	Output option for plastic strain in database: EQ.0: volumetric plastic strain, EQ.1: deviatoric plastic strain.
B	Residual strength factor after cracking, see Figure 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 invarient, 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.

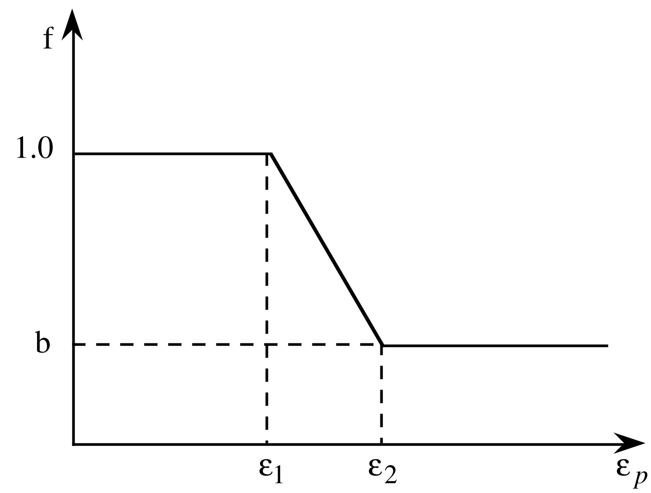


Figure 20.26. Strength reduction factor.

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.

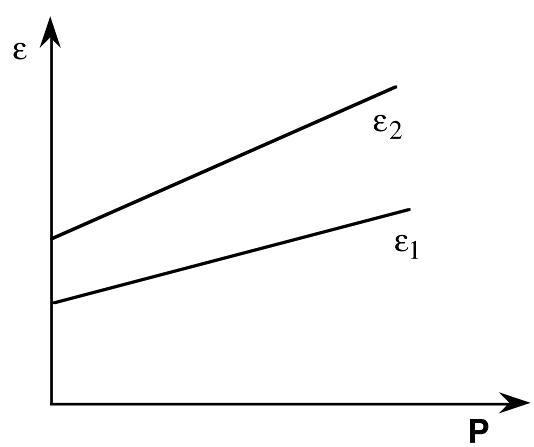


Figure 20.27. Cracking strain versus pressure.

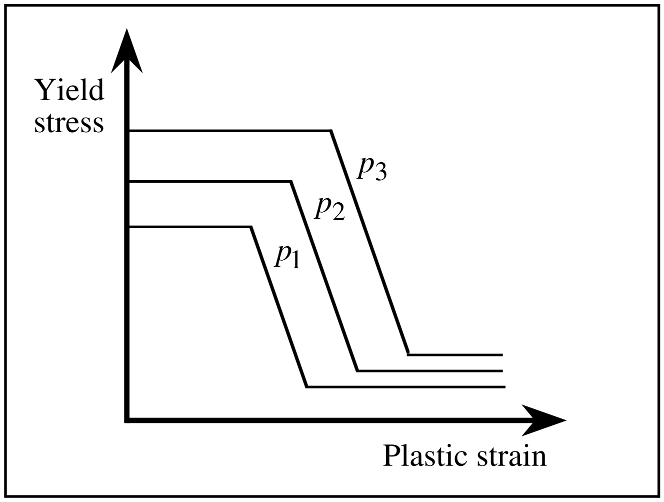


Figure 20.28.

*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 modulii 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≤ 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: $G = G_0(p - p_o)^b$, $K = K_0(p - p_o)^b$. b, must lie in the range 0≤b<1. Values close to 1 are not recommended because the pressure becomes indeterminate.
A0	Yield function constant a_o (Default = 1.0), see Mateiral Type 5.
A1	Yield function constant a_1 (Default = 0.0), see Material Type 5.
A2	Yield function constant a_2 (Default = 0.0), see Material Type 5.
DF	Damping factor. Must be in the range 0≤df≤1: EQ.0: no damping, EQ.1: maximum damping.
RP	Reference pressure for following input data.
LCID	Load curve ID defining shear strain verses shear stress. 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).

VARIABLE	DESCRIPTION
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 take from the stress-strain curve.

The stress strain pairs define a shear stress versus shear strain curve. The first point on the curve is assumed by default to be (0,0) and does not need to be entered. The slope of the curve must decrease with increasing γ. This curves applies at the reference pressure; at other pressures the curve 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.

$$G = G_0 (p - p_o)^b$$

$$K = K_0 (p - p_o)^b$$

where G_0 and K_0 are the input values, p is the current pressure, p_0 the cut-off or reference pressure (must be zero or negative). If p attempts to fall below p_0 (i.e., more tensile) the shear stresses are set to zero and the pressure is set to p_0. Thus, the material has no stiffness or strength in tension. The pressure in compression is calculated as follows:

$$p = \left[-K_0 \ln(V)\right]^{1/1-b}$$

where V is the relative volume, i.e., the ratio between the original and current volume.

*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:

$$\frac{\gamma}{\gamma_y} = \frac{\tau}{\tau_y} + \alpha \left| \frac{\tau}{\tau_y} \right|^r \quad if \quad \gamma \geq 0$$

$$\frac{\gamma}{\gamma_y} = \frac{\tau}{\tau_y} - \alpha \left| \frac{\tau}{\tau_y} \right|^r \quad if \quad \gamma < 0$$

where γ is the shear and τ is the stress. The model approaches perfect plasticity as the stress exponent $r \rightarrow \infty$. 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

$$\frac{(\gamma-\gamma_0)}{2\gamma_y} = \frac{(\tau-\tau_0)}{2\tau_y} + \alpha\left|\frac{(\tau-\tau_0)}{2\tau_y}\right|^r \quad if \quad \gamma \geq 0$$

$$\frac{(\gamma-\gamma_0)}{2\gamma_y} = \frac{(\tau-\tau_0)}{2\tau_y} - \alpha\left|\frac{(\tau-\tau_0)}{2\tau_y}\right|^r \quad if \quad \gamma < 0$$

where γ_0 and τ_0 represetn the values of strain and stress at the point of load reversal. Subsequent load reversals are detected by $(\gamma-\gamma_0)\dot{\gamma} < 0$.

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

This is Material Type 81. An elasto-visco-plastic material with an arbitrary stress versus strain curve and arbitrary strain rate dependency can be defined. Damage is considered before rupture occurs. Also, failure based on a plastic strain or a minimum time step size can be defined.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	SIGY	ETAN	EPPF	TDEL
Type	I	F	F	F	F	F	F	F
Default	none	none	none	none	none	0.0	10.E+20	10.E+20

Card 2								
Variable	C	P	LCSS	LCSR	EPPFR	VP	LCDM	
Type	F	F	F	F	F	F	F	
Default	0	0	0	0	0	0	0	

Card 3								
Variable	EPS1	EPS2	EPS3	EPS4	EPS5	EPS6	EPS7	EPS8
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

Card 4

Variable	ES1	ES2	ES3	ES4	ES5	ES6	ES7	ES8
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.
SIGY	Yield stress.
ETAN	Tangent modulus, ignored if (LCSS.GT.0) is defined.
EPPF	Plastic strain, *fs,* at which material softening begins (logrithmic).
TDEL	Minimum time step size for automatic element deletion.
C	Strain rate parameter, C, see formula below.
P	Strain rate parameter, P, see formula below.
LCSS	Load curve ID. Load curve ID defining effective stress versus effective plastic strain. If defined EPS1-EPS8 and ES1-ES8 are ignored.
LCSR	Load curve ID defining strain rate scaling effect on yield stress.
EPPFR	Plastic strain at which material ruptures (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.
EPS1-EPS8	Effective plastic strain values (optional if SIGY is defined). At least 2 points should be defined.

VARIABLE	DESCRIPTION
ES1-ES8	Corresponding yield stress values to EPS1 - EPS8.

Remarks:

The stress strain behavior may be treated by a bilinear stress strain curve by defining the tangent modulus, ETAN. Alternately, a curve similar to that shown in Figure 20.4 is expected to be defined by (EPS1,ES1) - (EPS8,ES8); however, an effective stress versus effective plastic strain curve (LCSS) may be input instead if eight points are insufficient. The cost is roughly the same for either approach. The most general approach is to use the table definition (LCSS) discussed below.

Two options to account for strain rate effects are possible.

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

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

where $\dot{\varepsilon}$ is the strain rate. $\dot{\varepsilon}=\sqrt{\dot{\varepsilon}_{ij}\dot{\varepsilon}_{ij}}$.

If the viscoplastic option is active, VP=1.0, and if SIGY is > 0 then the dynamic yield stress is computed from the sum of the static stress, $\sigma_y^s\left(\varepsilon_{eff}^p\right)$, which is typically given by a load curve ID, and the initial yield stress, SIGY, multiplied by the Cowper-Symonds rate term as follows:

$$\sigma_y\left(\varepsilon_{eff}^p,\dot{\varepsilon}_{eff}^p\right)=\sigma_y^s\left(\varepsilon_{eff}^p\right)+SIGY\cdot\left(\frac{\dot{\varepsilon}_{eff}^p}{C}\right)^{1/p}$$

where the plastic strain rate is used. With this latter approach similar results can be obtained between this model and material model: *MAT_ANISOTROPIC_VISCOPLASTIC. If SIGY=0, the following equation is used instead where the static stress, $\sigma_y^s\left(\varepsilon_{eff}^p\right)$, must be defined by a load curve:

$$\sigma_y\left(\varepsilon_{eff}^p,\dot{\varepsilon}_{eff}^p\right)=\sigma_y^s\left(\varepsilon_{eff}^p\right)\left[1+\left(\frac{\dot{\varepsilon}_{eff}^p}{C}\right)^{1/p}\right]$$

This latter equation is always used if the viscoplastic option is off.

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

The constitutive properties for the damaged material are obtained from the undamaged material properties. The amount of damage evolved is represented by the constant, ω, which varies from zero if no damage has occurred to unity for complete rupture. For uniaxial loading, the nominal stress in the damaged material is given by

$$\sigma_{nominal} = \frac{P}{A}$$

where P is the applied load and A is the surface area. The true stress is given by:

$$\sigma_{true} = \frac{P}{A - A_{loss}}$$

where A_{loss} is the void area. The damage variable can then be defined:

$$\omega = \frac{A_{loss}}{A} \qquad 0 \le \omega \le 1$$

In this model damage is defined in terms of plastic strain after the failure strain is exceeded:

$$\omega = \frac{\varepsilon^p_{eff} - \varepsilon^p_{failure}}{\varepsilon^p_{rupture} - \varepsilon^p_{failure}} \quad if \quad \varepsilon^p_{failure} \le \varepsilon^p_{eff} \le \varepsilon^p_{rupture}$$

After exceeding the failure strain softening begins and continues until the rupture strain is reached.

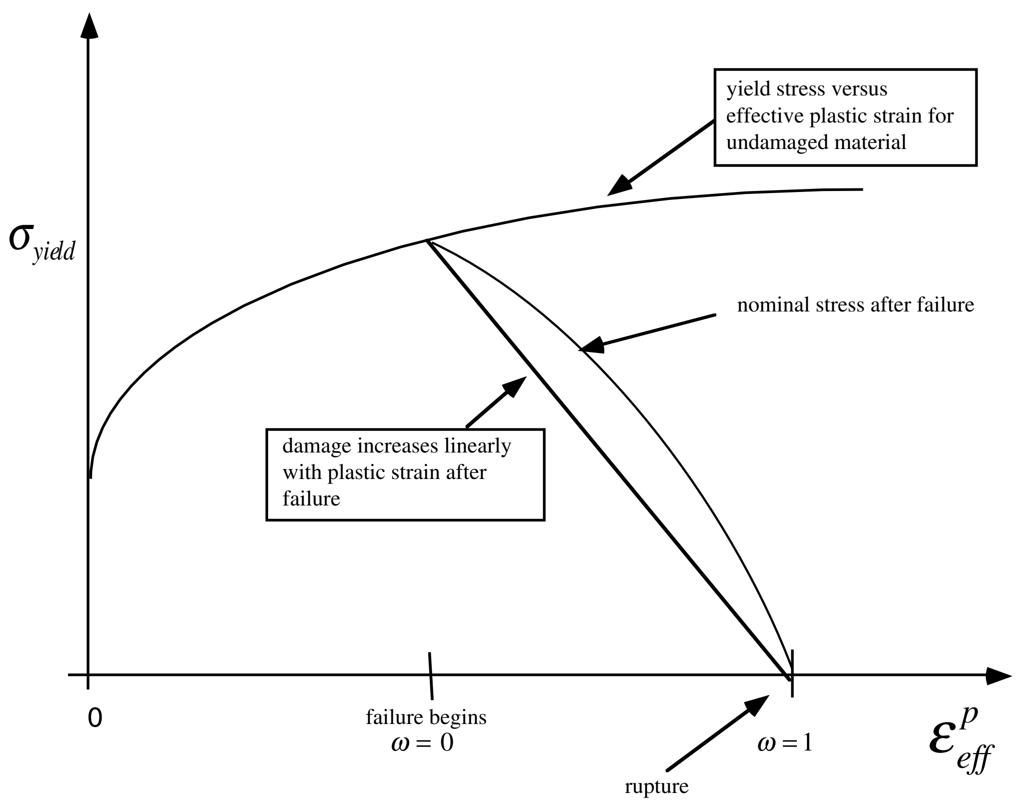


Figure 20.29. Stress strain behavior when damage is included.

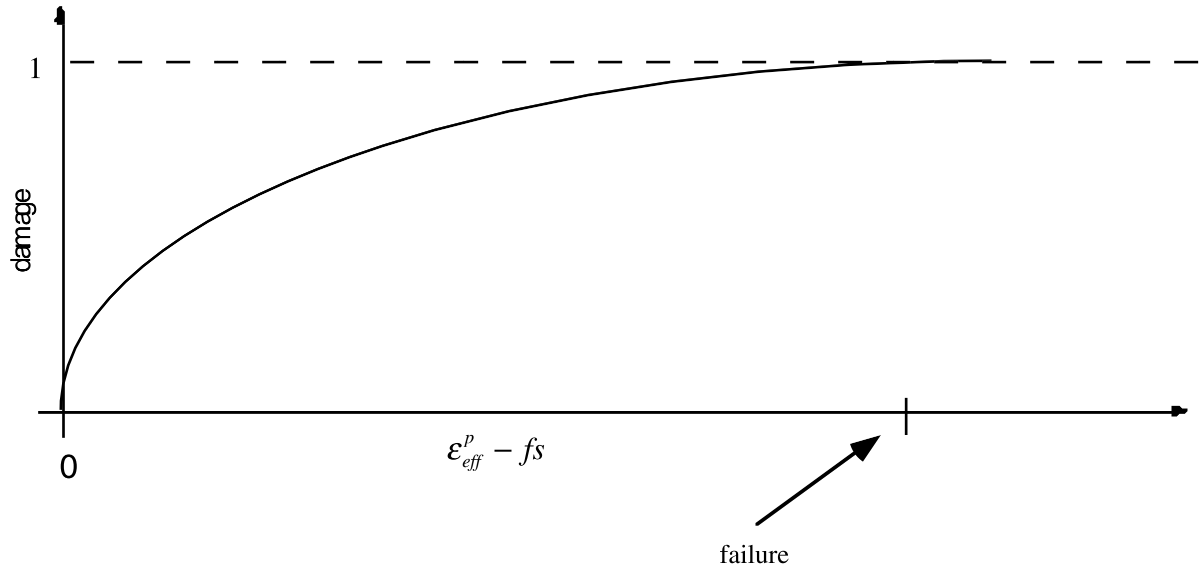


Figure 20.30. A nonlinear damage curve is optional. Note that the origin of the curve is at (0,0). It is permissible to input the failure strain, *fs*, as zero for this option. The nonlinear damage curve is useful for controlling the softening behavior after the failure strain is reached.

*MAT_FU_CHANG_FOAM

This is Material Type 83. Rate effects can be modelled in low and medium density foams, see Figure 20.31. Hysteretic unloading behavior in this model is a function of the rate sensitivity with the most rate sensitive foams providing the largest hystersis and visa versa. The unified constitutive equations for foam materials by Fu Chang [1995] provides the basis for this model. The mathematical description given below is excerpted from the reference. Further improvements have been incorporated based on work by Hirth, Du Bois, and Weimar [1998]. Their improvements permit: load curves generated by drop tower test to be directly input, a choice of principal or volumetric strain rates, load curves to be defined in tension, and the volumetric behavior to be specified by a load curve.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E	ED	TC	FAIL	DAMP	TBID
Type	I	F	F	F	F	F	F	F
Default	---	---	---	---	1.E+20			
Remarks	---	---	---	---	---			---

Card 2								
Variable	BVFLAG	SFLAG	RFLAG	TFLAG	PVID			
Type	F	F	F	F	F			
Default	1.0	0.0	0.0	0.0	0.0			
Remarks	1	2	3		4			

Card 3

Variable	D0	N0	N1	N2	N3	C0	C1	C2
Type	F	F	F	F	F	F	F	F
Default	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0

Card 4

Variable	C3	C4	C5	AIJ	SIJ	MINR	MAXR	
Type	F	F	F	F	F	F	F	
Default	0.0	0.0	0.0	0.0	0.0	0.0	0.0	

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density
E	Young's modulus
ED	Optional Young's relaxation modulus, E_d, for rate effects. See comments below. EQ.0.0: Maximum slope in stress vs. strain curve is used. When the maximum slope is taken for the contact, the time step size for this material is reduced for stability. In some cases 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.

VARIABLE	DESCRIPTION
TBID	Table ID, see *DEFINE_TABLE, for nominal stress strain data as a function of strain rate. If the table ID is provided, cards 3 and 4 may be left blank and the fit will be done internally.
BVFLAG	Bulk viscosity activation flag, see remark below: EQ.0.0: no bulk viscosity (recommended), EQ.1.0: bulk viscosity active.
D0	material constant, see equations below.
N0	material constant, see equations below.
N1	material constant, see equations below.
N2	material constant, see equations below.
N3	material constant, see equations below.
C0	material constant, see equations below.
C1	material constant, see equations below.
C2	material constant, see equations below.
C3	material constant, see equations below.
C4	material constant, see equations below.
C5	material constant, see equations below.
AIJ,	material constant, see equations below.
SIJ	material constant, see equations below.
MINR	Ratemin, minimum strain rate of interest.
MAXR	Ratemax, maximum strain rate of interest.
SFLAG	Strain rate flag (see remark 2 below): EQ.0.0: true constant strain rate, EQ.1.0: engineering strain rate.
RFLAG	Strain rate evaluation flag: EQ.0.0: first principal direction, EQ.1.0: principal strain rates for each principal direction, EQ.2.0: volumetric strain rate.
TFLAG	Tensile stress evaluation: EQ.0.0: linear in tension. EQ.1.0: input via load curves with the tensile response corresponds to negative values of stress and strain.

VARIABLE	DESCRIPTION
PVID	Optional load curve ID defining pressure versus volumetric strain.

Remarks:

The strain is divided into two parts: a linear part and a non-linear part of the strain

$$E(t) = E^L(t) + E^N(t)$$

and the strain rate become

$$\dot{E}(t) = \dot{E}^L(t) + \dot{E}^N(t)$$

$\dot{E}^N$ is an expression for the past history of E^N. A postulated constitutive equation may be written as:

$$\sigma(t) = \int_{\tau=0}^{\infty} \left[E_t^N(\tau), S(t) \right] d\tau$$

where $S(t)$ is the state variable and $\int_{.\tau=0}^{\infty}$ is a functional of all values of τ in $T_\tau : 0 \leq \tau \leq \infty$ and

$$E_t^N(\tau) = E^N(t - \tau)$$

where τ is the history parameter:

$$E_t^N(\tau = \infty) \Leftrightarrow \textit{the virgin material}$$

It is assumed that the material remembers only its immediate past, i.e., a neighborhood about $\tau = 0$. Therefore, an expansion of $E_t^N(\tau)$ in a Taylor series about $\tau = 0$ yields:

$$E_t^N(\tau) = E^N(0) + \frac{\partial E_t^N}{\partial t}(0) dt$$

Hence, the postulated constitutive equation becomes:

$$\sigma(t) = \sigma^*\left(E^N(t), \dot{E}^N(t), S(t)\right)$$

where we have replaced $\frac{\partial E_t^N}{\partial t}$ by $\dot{E}^N$, and σ^* is a function of its arguments.

For a special case,

$$\sigma(t) = \sigma^*\left(\dot{E}^N(t), S(t)\right)$$

we may write

$$\dot{E}_t^N = f\left(S(t), s(t)\right)$$

which states that the nonlinear strain rate is the function of stress and a state variable which represents the history of loading. Therefore, the proposed kinetic equation for foam materials is:

$$\dot{E}^N = \frac{\sigma}{\|\sigma\|} D_0 \exp\left[-c_0 \left(\frac{tr(\sigma S)}{(\|\sigma\|)^2} \right)^{2n_0} \right]$$

where D_0, c_0, and n_0 are material constants, and S is the overall state variable. If either $D_0 = 0$ or $c_0 \rightarrow \infty$ then the nonlinear strain rate vanishes.

$$\dot{S}_{ij} = \left[c_1 \left(a_{ij} R - c_2 S_{ij} \right) P + c_3 W^{n_1} \left(\left\| \dot{E}^N \right\| \right)^{n_2} I_{ij} \right] R$$

$$R = 1 + c_4 \left(\frac{\left\| \dot{E}^N \right\|}{c_5} - 1 \right)^{n_3}$$

$$P = tr\left(\sigma \dot{E}^N\right)$$

$$W = \int tr(\sigma\, dE)$$

where c_1, c_2, c_3, c_4, c_5, n_1, n_2, n_3, and a_{ij} are material constants and:

$$\|\sigma\| = \left(\sigma_{ij}\sigma_{ij}\right)^{\frac{1}{2}}$$

$$\left\|\dot{E}\right\| = \left(\dot{E}_{ij}\dot{E}_{ij}\right)^{\frac{1}{2}}$$

$$\left\|\dot{E}^N\right\| = \left(\dot{E}_{ij}^N \dot{E}_{ij}^N\right)^{\frac{1}{2}}$$

In the implementation by Fu Chang the model was simplified such that the input constants a_{ij} and the state variables S_{ij} are scalars.

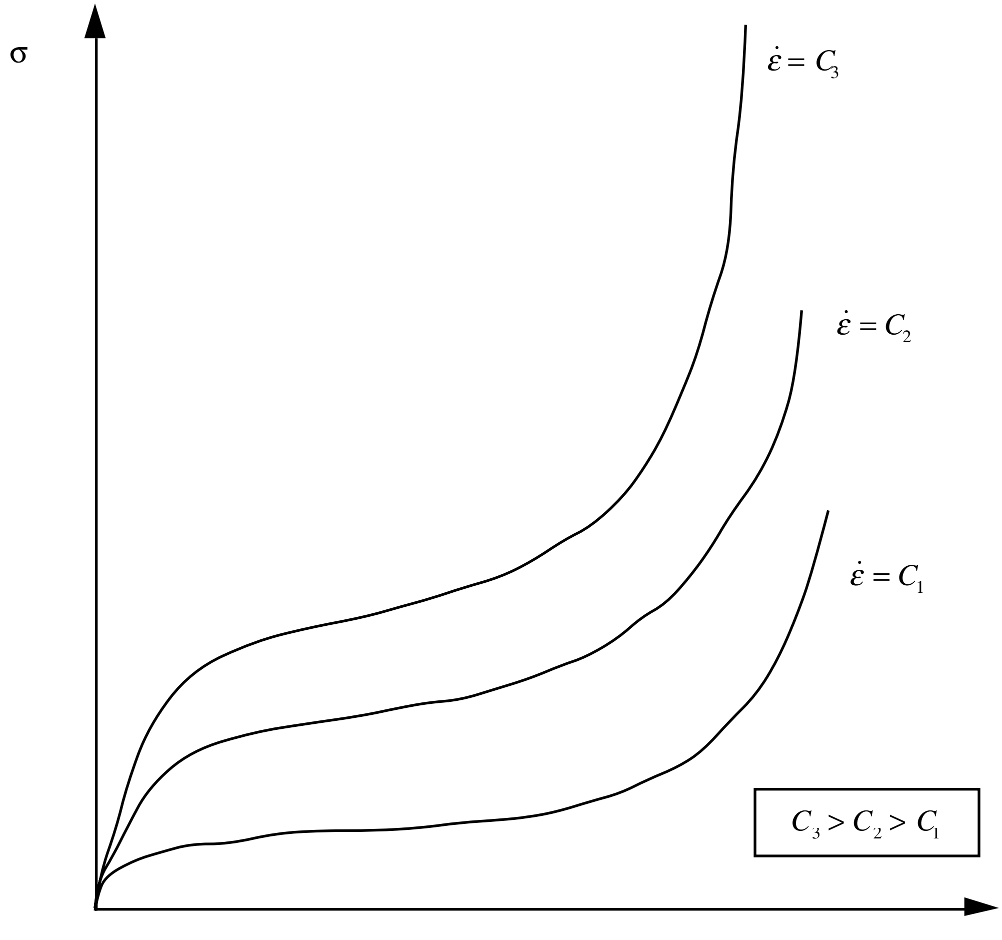


Figure 20.31. Rate effects in Fu Chang's foam model.

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 form the report by Broadhouse. The Fortran subroutines and quality assurance test problems were also provided to LSTC by the Winfrith Technology Center. The rebar is defined in the section: *MAT_WINFRITH_ CONCRETE_REINFORCEMENT which follows.

Card 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

VARIABLE	DESCRIPTION
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 competely 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).

The graphical post-processing code LS-TAURUS has been modified to 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.

*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		

VARIABLE	DESCRIPTION
EID1	First element ID in group.
EID2	Last element ID in group
INC	Element increment for generation.
XR	X-reinforcement quantity (for bars running parallel to global x-axis).
YR	Y-reinforcement quantity (for bars running parallel to global y-axis).
ZR	Z-reinforcement quantity (for bars running parallel to global z-axis).

VARIABLE	DESCRIPTION
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}

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, MANGLE, from a line in the plane of the element defined by the cross product of the vector **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 unixial 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]:

$$W(J_1, J_2, J) = \sum_{p,q=0}^{n} C_{pq}(J_1 - 3)^p (J_2 - 3)^q + W_H(J)$$

$$J_1 = I_1 I_3^{-1/3} \qquad \sigma_y = \left[3\left(a_0 + a_1 p + a_2 p^2\right)\right]^{1/2}$$

$$J_2 = I_2 I_3^{-2/3}$$

In order to prevent volumetric work from contributing to the hydrostatic work the first and second invarients are modified as shown. This procedure is described in more detail by Sussman and Bathe [1987].

The effects of confined air pressure in its overall response characteristics is included by augmenting the stress state within the element by the air pressure.

$$\sigma_{ij} = \sigma_{ij}^{sk} - \delta_{ij}\sigma^{air}$$

where σ_{ij}^{sk} is the bulk skeletal stress and σ^{air} is the air pressure computed from the equation:

$$\sigma^{air} = -\frac{p_0 \gamma}{1 + \gamma - \phi}$$

where p_0 is the initial foam pressure usually taken as the atmospheric pressure and γ defines the volumetric strain

$$\gamma = V - 1 + \gamma_0$$

where V is the relative volume of the voids and γ_0 is the initial volumetric strain which is typically zero. The rubber skeletal material is assumed to be incompressible.

Rate effects are taken into account through linear viscoelasticity by a convolution integral of the form:

$$\sigma_{ij} = \int_0^t g_{ijkl}(t-\tau)\frac{\partial \varepsilon_{kl}}{\partial \tau} d\tau$$

or in terms of the second Piola-Kirchhoff stress, S_{ij}, and Green's strain tensor, E_{ij},

$$S_{ij} = \int_0^t G_{ijkl}(t-\tau)\frac{\partial E_{kl}}{\partial \tau} d\tau$$

where $g_{ijkl}(t-\tau)$ and $G_{ijkl}(t-\tau)$ are the relaxation functions for the different stress measures. This stress is added to the stress tensor determined from the strain energy functional.

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.

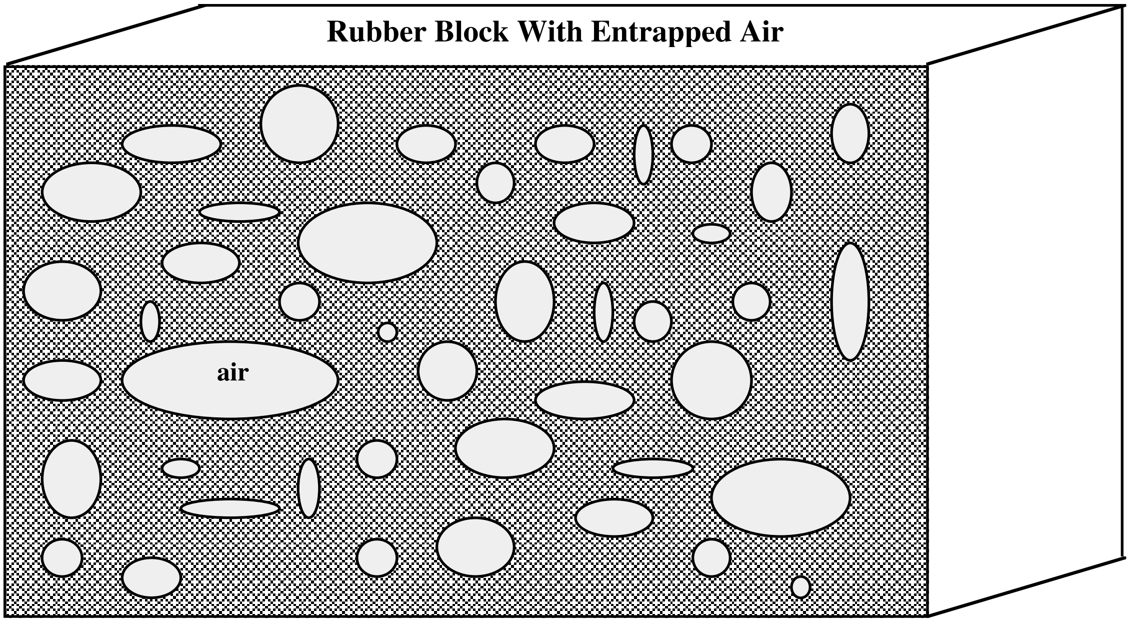


Figure 20.32. Cellular rubber with entrapped air. By setting the initial air pressure to zero, an open cell, cellular rubber can be simulated.

*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 value for the mechanical threshold stress, the flow stress is determined using either a thermal-activation-controlled or a drag-controlled kinetics relationship.. An equation-of-state is required for solid elements and a bulk modulus must be defined below for shell elements.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	SIGA	SIGI	SIGS	SIG0	BULK	
Type	I	F	F	F	F	F	F	

Card 2	1	2	3	4	5	6	7	8
Variable	HF0	HF1	HF2	SIGS0	EDOTS0	BURG	CAPA	BOLTZ
Type	F	F	F	F	F	F	F	F

Card 3	1	2	3	4	5	6	7	8
Variable	SM0	SM1	SM2	EDOT0	GO	PINV	QINV	EDOTI
Type	F	F	F	F	F	F	F	F

Card 4	1	2	3	4	5	6	7	8
Variable	G0I	PINVI	QINVI	EDOTS	G0S	PINVS	QINVS	
Type	F	F	F	F	F	F	F	

Card 5	1	2	3	4	5	6	7	8
Variable	RHOCPR	TEMPRF	ALPHA	EPS0				
Type	F	F						

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	$\hat{\sigma}_{\varepsilon so}$, saturation threshold stress at 0^{o} K (force/area).
BULK	Bulk modulus defined for shell elements only. Do not input for solid elements.

VARIABLE	DESCRIPTION
EDOTS0	$\dot{\varepsilon}_{\varepsilon so}$, 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	$\dot{\varepsilon}_o$, reference strain-rate (time $^{-1}$).
G0	g_0 , normalized activation energy for a .dislocation/dislocation interaction.
PINV	$\frac{1}{p}$, material constant.
QINV	$\frac{1}{q}$, material constant.
EDOTI	$\dot{\varepsilon}_{o,i}$, reference strain-rate (time $^{-1}$).
G0I	$g_{0,i}$, normalized activation energy for a dislocation/interstitial interaction.
PINVI	$\frac{1}{p_i}$, material constant.
QINVI	$\frac{1}{q_i}$, material constant.
EDOTS	$\dot{\varepsilon}_{o,s}$, reference strain-rate (time $^{-1}$).
G0S	$g_{0,s}$ normalized activation energy for a dislocation/solute interaction.
PINVS	$\frac{1}{p_s}$, material constant.
QINVS	$\frac{1}{q_s}$, material constant.

VARIABLE	DESCRIPTION
RHOCPR	ρc_p, product of density and specific heat.
TEMPRF	T_{ref}, initial element temperature in degrees K.
ALPHA	α, material constant (typical value is between 0 and 2).
EPS0	ε_o, factor to normalize strain rate in the calculation of Θ_o. (Use 1., 10^{-3}, or 10^{-6} for time units of seconds, milliseconds, or microseconds, respectively.)

Remarks:

The flow stress σ is given by:

$$\sigma = \hat{\sigma}_a + \frac{G}{G_0}\left[s_{th}\hat{\sigma} + s_{th,i}\hat{\sigma}_i + s_{th,s}\hat{\sigma}_s\right]$$

The first product in the equation for σ contains a micro-structure evolution variable, i.e,. $\hat{\sigma}$, called the *Mechanical Threshold Stress* (MTS), that is multiplied by a constant-structure deformation variable s_{th}: s_{th} is a function of absolute temperature T and the plastic strain-rates $\dot{\varepsilon}^P$. The evolution equation for $\hat{\sigma}$ is a differential hardening law representing dislocation-dislocation interactions:

$$\frac{\partial\hat{\sigma}}{\partial\varepsilon^p} \equiv \Theta_o\left[1 - \frac{\tanh\left(\alpha\dfrac{\hat{\sigma}}{\hat{\sigma}_{\varepsilon s}}\right)}{\tanh(\alpha)}\right]$$

The term, $\dfrac{\partial\hat{\sigma}}{\partial\varepsilon^p}$, represents the hardening due to dislocation generation and the stress ratio, $\dfrac{\hat{\sigma}}{\hat{\sigma}_{\varepsilon s}}$, represents softening due to dislocation recovery. The threshold stress at zero strain-hardening $\hat{\sigma}_{\varepsilon s}$ is called the saturation threshold stress. Relationships for Θ_o, $\hat{\sigma}_{\varepsilon s}$ are:

$$\Theta_o = a_o + a_1 \ln\left(\frac{\dot{\varepsilon}^p}{\varepsilon_0}\right) + a_2\sqrt{\frac{\dot{\varepsilon}^p}{\varepsilon_0}}$$

which contains the material constants, a_o, a_1, and a_2. The constant, $\hat{\sigma}_{\varepsilon s}$, is given as:

$$\hat{\sigma}_{\varepsilon s} = \hat{\sigma}_{\varepsilon so}\left(\frac{\dot{\varepsilon}^p}{\dot{\varepsilon}_{\varepsilon so}}\right)^{kT/Gb^3A}$$

which contains the input constants: $\hat{\sigma}_{\varepsilon so}$, $\dot{\varepsilon}_{\varepsilon so}$, b, A, and k. The shear modulus G appearing in these equations is assumed to be a function of temperature and is given by the correlation.

$$G = G_0 - b_1 / (e^{b_2 / T} - 1)$$

which contains the constants: G_0, b_1, and b_2. For thermal-activation controlled deformation s_{th} is evaluated via an Arrhenius rate equation of the form:

$$s_{th} = \left[1 - \left(\frac{kT1n\left(\frac{\dot{\varepsilon}_0}{\dot{\varepsilon}^p}\right)}{Gb^3 g_0} \right)^{\frac{1}{q}} \right]^{\frac{1}{p}}$$

The absolute temperature is given as:

$$T = T_{ref} + \rho c_p E$$

where E in the internal energy density per unit initial volume.

*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				

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, (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-propotional damping ratio. Typical values are 1e-3 or 1e-4. If set too high instabilites can result.
RATEFAC	Filtering factor for strain rate effects. Must be between 0 (no filtering) and 1 (infinite filtering) The filter is a simple low pass filter to remove high frequency oscillation from the strain rates before they are used in rate effect calculations. The cut off frequency of the filter is [(1 - RATEFAC) / timestep] rad/sec.
LCFAIL	Load curve ID giving variation of failure strain with strain rate. The points on the x-axis should be natural log of strain rate, the y-axis should be the true strain to failure. Typically this is measured by uniaxial tensile test, and the strain values converted to true strain.

Remarks:

1. Unlike other LS-DYNA material models, both the input stress-strain curve and the strain to failure are defined as total true strain, not plastic strain. The input can be defined from uniaxial tensile tests; nominal stress and nominal strain from the tests must be converted to true stress and true strain. The elastic component of strain must not be subtracted out.

2. The stress-strain curve is permitted to have sections steeper (i.e. stiffer) than the elastic modulus. When these are encountered the elastic modulus is increased to prevent spurious energy generation.

3. Sixty-four bit precision is recommended when using this material model, especially if the strains become high.

4. Invarient shell numbering is recommended when using this material model. See *CONTROL_ ACCURACY.

*MAT_ACOUSTIC

This is Material Type 90. This model is appropiate for tracking low pressure stress waves in an acoustic media such as air or water and can be used only with the acoustic pressure element formulation. The acoustic pressure element requires only one unknown per node. This element is very cost effective. 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

VARIABLE	DESCRIPTION
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)

VARIABLE	DESCRIPTION
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 (*OPTION*=**VISCO**)
T1 - T6	Characteristic times for Prony series relaxation kernel (*OPTION*=**VISCO**)

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:

$$W = C_1\left(\tilde{I}_1 - 3\right) + C_2\left(\tilde{I}_2 - 3\right) + F(\lambda) + \tfrac{1}{2}K\left[\ln(J)\right]^2$$

Here, $\tilde{I}_1$ and $\tilde{I}_2$ are the deviatoric invariants of the right Cauchy deformation tensor, λ is the deviatoric part of the stretch along the current fiber direction, and $J = \det \mathbf{F}$ is the volume ratio. The material coefficients C_1 and C_2 are the Mooney-Rivlin coefficients, while K is the effective bulk modulus of the material (input parameter XK).

The derivatives of the fiber term F are defined to capture the behavior of crimped collagen. The fibers are assumed to be unable to resist compressive loading - thus the model is isotropic when $\lambda < 1$. An exponential function describes the straightening of the fibers, while a linear function describes the behavior of the fibers once they are straightened past a critical fiber strestch level $\lambda \geq \lambda^*$ (input parameter XLAM):

$$\frac{\partial F}{\partial \lambda} = \begin{cases} 0 & \lambda < 1 \\ \frac{C_3}{\lambda}\left[\exp\left(C_4(\lambda - 1)\right) - 1\right] & \lambda < \lambda^* \\ \frac{1}{\lambda}\left(C_5\lambda + C_6\right) & \lambda \geq \lambda^* \end{cases}$$

Coefficients C_3, C_4, and C_5 must be defined by the user. C_6 is determined by LS-DYNA to ensure stress continuity at $\lambda = \lambda^*$. Sample values for the material coefficients $C_1 - C_5$ and λ^* for ligament tissue can be found in Quapp and Weiss [1998]. The bulk modulus K should be at least 3 orders of magnitude larger than C_1 to ensure near-incompressible material behavior.

Viscoelasticity is included via a convolution integral representation for the time-dependent second Piola-Kirchoff stress $\mathbf{S}(\mathbf{C},t)$:

$$\mathbf{S}(\mathbf{C},t) = \mathbf{S}^e(\mathbf{C}) + \int_0^t 2G(t-s)\frac{\partial W}{\partial \mathbf{C}(s)}ds$$

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 coefficeints to relaxation data tha tapproximates 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_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	

VARIABLE	DESCRIPTION
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. υ is the Poisson's ratio of the undamaged material also known as the virgin Poisson's ratio.

3. f_n is the initial principal tensile strength (stress) of the material. Once this stress has been reached at a point in the body a smeared crack is initiated there with a normal that is co-linear with the 1st principal direction. Once initiated, the crack is fixed at that location, though it will convect with the motion of the body. As the loading progresses the allowed tensile traction normal to the crack plane is progressively degraded to a small machine dependent constant.

   The degradation is implemented by reducing the material's modulus normal to the smeared crack plane according to a maximum dissipation law that incorporates exponential softening. The restriction on the normal tractions is given by

$$\phi_t = (\mathbf{n} \otimes \mathbf{n}):\sigma - f_n + (1-\varepsilon) f_n \left(1 - \exp[-H\alpha]\right) \le 0$$

   where $\mathbf{n}$ is the smeared crack normal, ε is the small constant, H is the softening modulus, and α is an internal variable. H is set automatically by the program; see g_c below. α measures the crack field intensity and is output in the equivalent plastic strain field, $\bar{\varepsilon}^p$, in a normalized fashion.

   The evolution of alpha is governed by a maximum dissipation argument. When the normalized value reaches unity it means that the material's strength has been reduced to 2% of its original value in the normal and parallel directions to the smeared crack. Note that for plotting purposes it is never output greater than 5.

4. f_s is the initial shear traction that may be transmitted across a smeared crack plane. The shear traction is limited to be less than or equal to $f_s(1-\beta)(1-\exp[-H\alpha])$, through the use of two orthogonal shear damage surfaces. Note that the shear degradation is coupled to the tensile degradation through the internal variable alpha which measures the intensity of the crack field. β is the shear retention factor defined below. The shear degradation is taken care of by reducing the material's shear stiffness parallel to the smeared crack plane.

5. g_c is the fracture toughness of the material. It should be entered as fracture energy per unit area crack advance. Once entered the softening modulus is automatically calculated based on element and crack geometries.

6. β is the shear retention factor. As the damage progresses the shear tractions allowed across the smeared crack plane asymptote to the product βf_s.

7. η represents the viscosity of the material. Viscous behavior is implemented as a simple Perzyna regularization method. This allows for the inclusion of first order rate effects. The use of some viscosity is recommend as it serves as regularizing parameter that increases the stability of calculations.

8. σ_y is a uniaxial compressive yield stress. A check on compressive stresses is made using the J2 yield function $\mathbf{s}:\mathbf{s}-\sqrt{\frac{2}{3}}\sigma_y \leq 0$, 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=3.15x10^6 psi, f_n=450 psi, f_s=2100 psi, ν = 0.2, g_c = 0.8 lbs/in, β =0.03, η = 0.0 psi-sec, σ_y = 4200 psi. For stability, values of η between 104 to 106 psi/sec are recommended. Our limited experience thus far has shown that many problems require nonzero valuies of η to run to avoid error terminations.

Remark: Various other internal variables such as crack orientations and degraded stiffness tensors are internally calculated but currently not available for output.

*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				
Type	I	F	F	F				
Default	none	none	none	none				

Card 2

Variable	A	B	N	C	PSFAIL	SIGMAX	SIGSAT	EPSO
Type	F	F	F	F	F	F	F	F
Default	none	0.0	0.0	0.0	1.0E+17	SIGSAT	1.0E+28	1.0

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density
E	Young's modulus

VARIABLE	DESCRIPTION
PR	Poisson's ratio
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.
SIGSAT	Saturation stress which limits the maximum value of effective stress which can develop after rate effects are added.
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:

Johnson and Cook express the flow stress as

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

where

A, B, C and n are input constants

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

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

The maximum stress is limited by *sigmax* and *sigsat* by:

$$\sigma_y = min\left\{min\left[A + B\,\bar{\varepsilon}^{p^n}\ ,sigmax\right]\left(1 + c\ \ln \dot{\varepsilon}^*\right),sigsat\right\}$$

Failure occurs when the effective plastic strain exceeds *psfail*.

*MAT_SPOTWELD_{*OPTION*}

This is Material Type 100. The material model applies to beam element type 9 for spot welds. These 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 the somewhate more expensive, *CONTACT_SPOTWELD_WITH_TORSION, contact will carry and transmit the torsional forces to and from the shell surfaces. This contact option can be used for deformable solid elements and thick, 8-node shells which lack rotational degrees-of-freedom since it computes the rotation from the kinematics of the surface.

Beam 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.

Options include:

<BLANK>

DAMAGE

The DAMAGE option 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.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	SIGY	ET	DT	TFAIL
Type	I	F	F	F	F	F	F	F

Card 2	1	2	3	4	5	6	7	8
Variable	EFAIL	NRR	NRS	NRT	MRR	MSS	MTT	NF
Type	F	F	F	F	F	F	F	F

Define this card if and only if the DAMAGE option is active.

Card 3	1	2	3	4	5	6	7	8
Variable	RS							
Type	F							

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
E	Young's modulus
PR	Poisson's ratio
SIGY	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.
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} at failure. If zero, failure due to this component is not considered.
NRS	Force resultant N_{rs_F} at failure. If zero, failure due to this component is not considered.

VARIABLE	DESCRIPTION
NRT	Force resultant N_{rt_F} at failure. If zero, failure due to this component is not considered.
MRR	Torsional moment resultant M_{rr_F} at failure. If zero, failure due to this component is not considered.
MSS	Moment resultant M_{ss_F} at failure. If zero, failure due to this component is not considered.
MTT	Moment resultant M_{tt_F} at failure. If zero, failure due to this component is not considered.
NF	Number of force vectors stored for filtering. The default value is set to zero which is generally recommended unless oscillatory resultant forces are observed in the time history databases. Even though these welds should not oscillate significantly, this option was added for consistency with the other spot weld options. NF affects the storage since it is necessary to store the resultant forces as history variables. When NF is nonzero, the resultants in the output databases are filtered.
RS	Rupture strain.

Remarks:

The weld material is modeled with isotropic hardening plasticity coupled to two failure models. The first model specifies a failure strain which fails each integration point in the spot weld independently. The second model fails the entire weld if the resultants are outside of the failure surface defined by:

$$\left(\frac{max(N_{rr},0)}{N_{rr_F}}\right)^2+\left(\frac{N_{rs}}{N_{rs_F}}\right)^2+\left(\frac{N_{rt}}{N_{rt_F}}\right)^2+\left(\frac{M_{rr}}{M_{rr_F}}\right)^2+\left(\frac{M_{ss}}{M_{ss_F}}\right)^2+\left(\frac{M_{tt}}{M_{tt_F}}\right)^2-1=0$$

where the *numerators* in the equation are the resultants calculated in the local coordinates of the cross section, and the **denominators** are the values specified in the input. If NF is nonzero the resultants are filtered before failure is checked.

If the failure strain is set to zero, the failure strain model is not used. In a similar manner, when the value of a resultant at failure is set to zero, the corresponding term in the failure surface is ignored. For example, if only N_{rr_F} is nonzero, the failure surface is reduced to $|N_{rr}|=N_{rr_F}$. None, either, or both of the failure models may be active depending on the specified input values.

The inertias of the spot welds are scaled during the first time step so that their stable time step size is Δt. A strong compressive load on the spot weld at a later time may reduce the length of the spot weld so that stable time step size drops below Δt. If the value of Δt is zero, mass scaling is not performed, and the spot welds will probably limit the time step size. Under most circumstances, the inertias of the spot welds are small enough that scaling them will have a negligible effect on the structural response and the use of this option is encouraged.

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:

$$\omega = \frac{A_{loss}}{A} \qquad 0 \le \omega \le 1$$

In this model damage is defined in terms of plastic strain after the failure strain is exceeded:

$$\omega = \frac{\varepsilon_{eff}^{p} - \varepsilon_{failure}^{p}}{\varepsilon_{rupture}^{p} - \varepsilon_{failure}^{p}} \quad if \quad \varepsilon_{failure}^{p} \le \varepsilon_{eff}^{p} \le \varepsilon_{rupture}^{p}$$

After exceeding the failure strain softening begins and continues until the rupture strain is reached.

*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				

VARIABLE	DESCRIPTION
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.

VARIABLE	DESCRIPTION
LCFSIG	Load curve ID that defines the Maximum principal failure stress as a function of strain rate.
LCE	Load curve ID that defines the Unloading moduli as a function of plastic strain.

Remarks:

The constitutive model for this approach is:

$$\dot{\varepsilon}_p = \dot{\varepsilon}_0 \exp(A\{\sigma - S(\varepsilon_p)\}) \times \exp(-p\alpha A)$$

where $\dot{\varepsilon}_0$ and A are rate dependent yield stress parameters, $S(\varepsilon_p)$ internal resistance (strain hardening) and α is a pressure dependence parameter.

In this material characteristic, 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 thermoplactics, GE Plastics have constants available for use in a wide range of commercially available grades of their engineering thermoplastics.

*MAT_INV_HYPERBOLIC_SIN

This is Material Type 102. It allows the modeling of temperature and rate dependent plasticity, Sheppard and Wright [1979].

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	T	HC	VP	
Type	I	F	F	F	F	F	F	

Card 2	1	2	3	4	5	6	7	8
Variable	ALPHA	N	A	Q	G	EPS0		
Type	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
T	Initial Temperature.
HC	Heat generation coefficient.
VP	Formulation for rate effects: EQ.0.0: Scale yield stress (default) EQ.1.0: Viscoplastic formulation.
ALPHA	See remarks.
N	See remarks.

VARIABLE	DESCRIPTION
A	See remarks.
Q	See remarks.
G	See remarks.

Remarks:

Resistance to deformation is both temperature and strain rate dependent. The flow stress equation is:

$$\sigma = \frac{1}{\alpha}\sinh^{-1}\left(\left[\frac{Z}{A}\right]^{\frac{1}{N}}\right)$$

where Z, the Zener-Holloman temperature compensated strain rate, is:

$$Z = \dot{\varepsilon}\exp\left(\frac{Q}{GT}\right)$$

where the constant with example units are:

The material constitutive constants, 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 which allows us to compute the temperature change during the deformation. In the absence of a couples thermo-mechanical finite element code we assume adiabatic temperature change and follow the empirical assumption that 90-95% of the plastic work is dissipated as heat. Thus the heat generation coefficient is

$$HC \approx \frac{0.9}{\rho C_v}$$

where ρ is the density of the material and C_v is the specific heat.

*MAT_ANISOTROPIC_VISCOPLASTIC

This is Material Type 103. This anisotropic-viscoplastic material model applies to shell and brick elements. The material constants may be fit directly or, if desired, stress versus strain data may be input and a least squares fit will be performed by LS-DYNA to determine the constants. Kinematic or isotopic or a combination of kinematic and isotropic hardening may be used.. A detailed describtion of this model can be found in the following references: Berstad, Langseth, and Hopperstad [1994]; Hopperstad and Remseth [1995]; and Berstad [1996].

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	SIGY	FLAG	LCSS	ALPHA
Type	I	F	F	F	F	F	F	F

Card 2	1	2	3	4	5	6	7	8
Variable	QR1	CR1	QR2	CR2	QX1	CX1	QX2	CX2
Type	F	F	F	F	F	F	F	F

Card 3	1	2	3	4	5	6	7	8
Variable	VK	VM	R00 or F	R45 or G	R90 or H	L	M	N
Type	F	F	F	F	F	F	F	F

Card 4	1	2	3	4	5	6	7	8
Variable	AOPT							
Type	F							

Card 5

Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card 6

Variable	V1	V2	V3	D1	D2	D3	BETA	
Type	F	F	F	F	F	F	F	

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
E	Young's modulus
PR	Poisson's ratio
SIGY	Initial yield stress
FLAG	Flag EQ.0: Give all material parameters EQ.1: Material parameters are fit in LS-DYNA to Load curve or Table given below. The parameters Q_{r1}, C_{r1}, Q_{r2}, and C_{r2} for isotropic hardening are determined by the fit and those for kinematic hardening are found by scaling those for isotropic hardening by $(1-\alpha)$ where 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 effectiveplastic strain for that rate. If the load curve only is used, then the coefficients V_k and V_m must be given if viscoplastice behavior is desired. If a Table ID is given these coefficients are determined internally during initialization.

VARIABLE	DESCRIPTION
ALPHA	α distribution of hardening used in the curve-fitting. $\alpha = 0$ pure kinematic hardening and $\alpha = 1$ provides pure isotropic hardening
QR1	Isotropic hardening parameter Q_{r1}
CR1	Isotropic hardening parameter C_{r1}
QR2	Isotropic hardening parameter Q_{r2}
CR2	Isotropic hardening parameter C_{r2}
QX1	Kinematic hardening parameter $Q_{\chi 1}$
CX1	Kinematic hardening parameter $C_{\chi 1}$
QX2	Kinematic hardening parameter $Q_{\chi 2}$
CX2	Kinematic hardening parameter $C_{\chi 2}$
VK	Viscous material parameter V_k
VM	Viscous material parameter V_m
R00	R_{00} for shell (Default=1.0)
R45	R_{45} for shell (Default=1.0)
R90	R_{90} for shell (Default=1.0)
F	F for brick (Default =1/2)
G	G for brick (Default =1/2)
H	H for brick (Default =1/2)
L	L for brick (Default =3/2)
M	M for brick (Default =3/2)
N	N for brick (Default =3/2)

VARIABLE	DESCRIPTION
AOPT	Material axes option (see MAT_*OPTION* TROPIC_ELASTIC for a more complete description): EQ. 0.0: locally orthotropic with material axes determined by element nodes 1, 2, and 4, as with *DEFINE_ COORDINATE_NODES. EQ. 1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center; this is the a-direction. This option is for solid elements only. EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR. EQ. 3.0: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle, BETA, from a line in the plane of the element defined by the cross product of the vector **v** with the element normal. EQ. 4.0: locally orthotropic in cylindrical coordinate system with the material axes determined by a vector **v**, and an originating point, P, which define the centerline axis. This option is for solid elements only.
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

$$\begin{aligned}\sigma(\varepsilon_{eff}^{p},\dot{\varepsilon}_{eff}^{p}) = \sigma_0 &+ Q_{r1}(1-\exp(-C_{r1}\varepsilon_{eff}^{p})) + Q_{r2}(1-\exp(-C_{r2}\varepsilon_{eff}^{p})) \\ &+ Q_{\chi 1}(1-\exp(-C_{\chi 1}\varepsilon_{eff}^{p})) + Q_{\chi 2}(1-\exp(-C_{\chi 2}\varepsilon_{eff}^{p})) \\ &+ V_k \dot{\varepsilon}_{eff}^{p\,V_m}\end{aligned}$$

For bricks the following yield criteria is used

$$\begin{aligned}F(\sigma_{22}-\sigma_{33})^2 + G(\sigma_{33}-\sigma_{11})^2 + H(\sigma_{11}-\sigma_{22})^2 \\ + 2L\sigma_{23}^2 + 2M\sigma_{31}^2 + 2N\sigma_{12}^2 = \sigma(\varepsilon_{eff}^{p},\dot{\varepsilon}_{eff}^{p})\end{aligned}$$

where ε_{eff}^{p} is the effective plastic strain and $\dot{\varepsilon}_{eff}^{p}$ is the effective plastic strain rate. For shells the anisotropic behavior is given by R_{00}, R_{45} and R_{90}. The model will work when the three first parameters in card 3 are given values. When $V_k = 0$ the material will behave elasto-plastically. Default values are given by:

$$F = G = H = \frac{1}{2}$$

$$L = M = N = \frac{3}{2}$$

$$R_{00} = R_{45} = R_{90} = 1$$

Strain rate of accounted for using the Cowper and Symonds model which, e.g., model 3, scales the yield stress with the factor:

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

To convert these constants set the viscoelastic constants, V_k and V_m, to the following values:

$$V_k = \sigma \left(\frac{1}{C} \right)^{\frac{1}{p}}$$

$$V_m = \frac{1}{p}$$

This model properly treats rate effects and should provide superior results to models 3 and 24.

*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 describtion of this model can be found in the paper by Berstad, Hopperstad, Lademo, and Malo[1999].

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	SIGY			
Type	I	F	F	F	F			

Card 2	1	2	3	4	5	6	7	8
Variable	Q1	C1	Q2	C2	EPSD	S	DC	FLAG
Type	F	F	F	F	F	F	F	F

Card 3	1	2	3	4	5	6	7	8
Variable	VK	VM	R00 or F	R45 or G	R90 or H	L	M	N
Type	F	F	F	F	F	F	F	F

Card 4	1	2	3	4	5	6	7	8
Variable	AOPT							
Type	F							

Card 5

Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card 6

Variable	V1	V2	V3	D1	D2	D3	BETA	
Type	F	F	F	F	F	F	F	

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
E	Young's modulus
PR	Poisson's ratio
SIGY	Initial yield stress, σ_0.
Q1	Isotropic hardening parameter Q_1
C1	Isotropic hardening parameter C_1
Q2	Isotropic hardening parameter Q_2
C2	Isotropic hardening parameter C_2
EPSD	Damage threshold r_d Damage effective plastic strain when material softening begin.(Default=0.0)
S	Damage material constant S. (Default= $\frac{\sigma_0}{200}$)
DC	Critical damage value D_C. When the damage value D reaches this value, the elment is deleted from the calculation. (Default=0.5)

VARIABLE	DESCRIPTION
FLAG	Flag EQ.0.No calculation of localization EQ.1.The model flags element where strain localization occur
VK	Viscous material parameter V_k
VM	Viscous material parameter V_m
R00	R_{00} for shell (Default=1.0)
R45	R_{45} for shell (Default=1.0)
R90	R_{90} for shell (Default=1.0)
F	F for brick (Default =1/2)
G	G for brick (Default =1/2)
H	H for brick (Default =1/2)
L	L for brick (Default =3/2)
M	M for brick (Default =3/2)
N	N for brick (Default =3/2)
AOPT	Material axes option (see MAT_*OPTION* TROPIC_ELASTIC for a more complete description): EQ. 0.0: locally orthotropic with material axes determined by element nodes 1, 2, and 4, as with *DEFINE_ COORDINATE_NODES. EQ. 1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center; this is the a-direction. This option is for solid elements only. EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR. EQ. 3.0: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle, BETA, from a line in the plane of the element defined by the cross product of the vector **v** with the element normal. EQ. 4.0: locally orthotropic in cylindrical coordinate system with the material axes determined by a vector **v**, and an originating point, P, which define the centerline axis. This option is for solid elements only.

VARIABLE	DESCRIPTION
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 Continium Damage Mecahnics (CDM) model is based on a CDM model proposed by Lemaitre [1992]. The effective stress $\tilde{\sigma}$, which is the stress calculated over the section that effectively resist the forces and reads.

$$\tilde{\sigma} = \frac{\sigma}{1-D}$$

where D is the damage variable. The evolution equation for the damage variable is defined as

$$\dot{D} = \begin{cases} 0 & \text{for} \quad r \le r_D \\ \dfrac{Y}{S(1-D)}\dot{r} & \text{for} \quad r > r_D \quad \text{and} \quad \sigma_1 > 0 \end{cases}$$

where r_D is the damage threshold, S is a positive material constant , S is the so-called strain energy release rate and σ_1 is the maxiaml principal stress. The strain energy density release rate is

$$Y = \frac{1}{2}\mathbf{e_e} : \mathbf{C} : \mathbf{e_e} = \frac{\sigma_{vm}^2 R_v}{2E(1-D)^2}$$

where σ_{vm} is the equivalent von Mises stress. The triaxiality function R_v is defined as

$$R_v = \frac{2}{3}(1+\nu) + 3(1-2\nu)\left(\frac{\sigma_H}{\sigma_{vm}}\right)^2$$

The uniaxial stress-strain curve is given in the following form

$$\sigma(r, \dot{\varepsilon}_{eff}^p) = \sigma_0 + Q_1(1-\exp(-C_1 r)) + \ Q_2(1-\exp(-C_2 r)) \\ + V_k \dot{\varepsilon}_{eff}^{p\ V_m}$$

where r is the damage accumalted plastic strain, which can be calculated by

$$\dot{r} = \dot{\varepsilon}_{eff}^{p}(1 - D)$$

For bricks the following yield criteria is used

$$F(\tilde{\sigma}_{22} - \tilde{\sigma}_{33})^2 + G(\tilde{\sigma}_{33} - \tilde{\sigma}_{11})^2 + H(\tilde{\sigma}_{11} - \tilde{\sigma}_{22})^2 \\ + 2L\tilde{\sigma}_{23}^2 + 2M\tilde{\sigma}_{31}^2 + 2N\tilde{\sigma}_{12}^2 = \sigma(r, \dot{\varepsilon}_{eff}^{p})$$

where r is the damage effective viscoplastic strain and $\dot{\varepsilon}_{eff}^{p}$ is the effective viscoplastic strain rate. For shells the anisotropic behavior is given by the R-values: R_{00}, R_{45}, and R_{90}. When $V_k = 0$ the material will behave as an elastoplastic material without rate effects. Default values for the anisotropic constants are given by:

$$F = G = H = \frac{1}{2}$$

$$L = M = N = \frac{3}{2}$$

$$R_{00} = R_{45} = R_{90} = 1$$

so that isotropic behavior is obtained.

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

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

To convert these constants, set the viscoelastic constants, V_k and V_m, to the following values:

$$V_k = \sigma\left(\frac{1}{C}\right)^{\frac{1}{p}}$$

$$V_m = \frac{1}{p}$$

*MAT_DAMAGE_2

This is Material Type 105. This is 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

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.
SIGY	Yield stress.
ETAN	Tangent modulus, ignored if (LCSS.GT.0) is defined.
FAIL	Failure flag. 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.

VARIABLE	DESCRIPTION
C	Strain rate parameter, C, see formula below.
P	Strain rate parameter, P, see formula below.
LCSS	Load curve ID or Table ID. Load curve ID defining effective stress versus effective plastic strain. If defined EPS1-EPS8 and ES1-ES8 are ignored. The table ID defines for each strain rate value a load curve ID giving the stress versus effectiveplastic strain for that rate, See Figure 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 elment 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

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

where $\dot{\varepsilon}$ is the strain rate. $\dot{\varepsilon} = \sqrt{\dot{\varepsilon}_{ij}\dot{\varepsilon}_{ij}}$

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

III. If different stress versus strain curves can be provided for various strain rates, the option using the reference to a table (LCSS) can be used. Then the table input in *DEFINE_TABLE has to be used, see Figure 20.7.

A fully viscoplastic formulation is used in this model.

*MAT_ELASTIC_VISCOPLASTIC_THERMAL

This is Material Type 106. This is an elastic viscoplastic material with thermal effects.

Card 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	VK	VM	LCE	LCPR	LCSIGY	LCR	LCX	LCALPH
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
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.
ALPHA	Coefficient of thermal expansion.

VARIABLE	DESCRIPTION
QR1	Isotropic hardening parameter Q_{r1}
CR1	Isotropic hardening parameter C_{r1}
QR2	Isotropic hardening parameter Q_{r2}
CR2	Isotropic hardening parameter C_{r2}
QX1	Kinematic hardening parameter $Q_{\chi 1}$
CX1	Kinematic hardening parameter $C_{\chi 1}$
QX2	Kinematic hardening parameter $Q_{\chi 2}$
CX2	Kinematic hardening parameter $C_{\chi 2}$
VK	Viscous material parameter V_k
VM	Viscous material parameter V_m
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.

Remarks:

The uniaxial stress-strain curve has the form

$$\sigma(\varepsilon_{eff}^{p},\dot{\varepsilon}_{eff}^{p}) = \sigma_0 + Q_{r1}(1-\exp(-C_{r1}\varepsilon_{eff}^{p})) + Q_{r2}(1-\exp(-C_{r2}\varepsilon_{eff}^{p})) + Q_{\chi 1}(1-\exp(-C_{\chi 1}\varepsilon_{eff}^{p})) + Q_{\chi 2}(1-\exp(-C_{\chi 2}\varepsilon_{eff}^{p})) + V_k \dot{\varepsilon}_{eff}^{p\,V_m}$$

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

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

To convert from these constants, set the viscoelastic constants, V_k and V_m, to the following values:

$$V_k = \sigma\left(\frac{1}{C}\right)^{\frac{1}{p}}$$

$$V_m = \frac{1}{p}$$

*MAT_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 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

$$\sigma_i^* = a(p^* + t^*)^n(1 + c\ln\dot{\varepsilon})$$

represents the intact, undamaged behaviour,

$$D = \sum \Delta\varepsilon^p / \varepsilon_f^p$$

represents the accumulated damage based upon the increase in plastic strain per computational cycle and the plastic strain to fracture

$$\varepsilon_f^p = d_1(p^* + t^*)^{d_2}$$

and

$$\sigma_f^* = b(p^*)^m (1 + c \ln \dot{\varepsilon}) \leq sfma$$

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 functiion 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		

VARIABLE	DESCRIPTION
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 coeffisient.

VARIABLE	DESCRIPTION
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

$$\sigma^* = \left[A(1-D) + BP^{*N}\right]\left[1 - c\ln\left(\dot{\varepsilon}^*\right)\right]$$

where D is the damage parameter, $P^* = P / f_c^{'}$ is the normalized pressure and $\dot{\varepsilon}^* = \dot{\varepsilon} / \dot{\varepsilon}_0$ is the dimensionless strain rate. The model 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

$$P = K_1\overline{\mu} + K_2\overline{\mu}^2 + K_3\overline{\mu}^3$$

where K_1 , K_2 and K_3 are material constants and the modified volumteric strain is defined as

$$\overline{\mu} = \frac{\mu - \mu_{lock}}{1 + \mu_{lock}}$$

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 caluculations will be saved. GT.0.0: Plastic strain to failure. When the plastic strain reaches this value, the element is deleted from the calculation.
TDEL	Minimum time step size for automatic element deletion.
C	Strain rate parameter, C, see formula below.
P	Strain rate parameter, P, see formula below.
LCSS	Load curve ID or Table ID. Load curve ID defining effective stress versus effective plastic strain. If defined EPS1-EPS8 and ES1-ES8 are ignored. The table ID defines for each strain rate value a load curve ID giving the stress versus effectiveplastic strain for that rate, See Figure 20.7. The stress versus effective plastic strain curve for the lowest value of strain rate is used if the strain rate falls below the minimum value. Likewise, the stress versus effective plastic strain curve for the highest value of strain rate is used if the strain rate exceeds the maximum value. The strain rate parameters: C and P;

VARIABLE	DESCRIPTION
	the curve ID, LCSR; EPS1-EPS8 and ES1-ES8 are ignored if a Table ID is defined.
LCSR	Load curve ID defining strain rate scaling effect on yield stress.
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

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

where $\dot{\varepsilon}$ is the strain rate. $\dot{\varepsilon}=\sqrt{\dot{\varepsilon}_{ij}\dot{\varepsilon}_{ij}}$.

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

III. If different stress versus strain curves can be provided for various strain rates, the option using the reference to a table (LCSS) can be used. Then the table input in *DEFINE_TABLE has to be used, see Figure 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

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
E	Young’s modulus.
PR	Poisson’s ratio.
SIGY	Yield stress.
ETAN	Tangent modulus, ignored if (LCSS.GT.0) is defined.
FAIL	Failure flag. LT.0.0: User defined failure subroutine is called to determine failure EQ.0.0: Failure is not considered. This option is recommended if failure is not of interest since many caluculations will be saved. GT.0.0: Plastic strain to failure. When the plastic strain reaches this value, the element is deleted from the calculation.
TDEL	Minimum time step size for automatic element deletion.
C	Strain rate parameter, C, see formula below.
P	Strain rate parameter, P, see formula below.
LCSS	Load curve ID or Table ID. Load curve ID defining effective stress versus effective plastic strain. If defined EPS1-EPS8 and ES1-ES8 are ignored. The table ID defines for each strain rate value a load curve ID giving the stress versus effectiveplastic strain for that rate, See Figure 20.7. The stress versus effective plastic strain curve for the lowest value of strain rate is used if the strain rate falls below the minimum value. Likewise, the stress versus effective plastic strain curve for the highest value of strain rate is used if the strain rate exceeds the maximum value. The strain rate parameters: C and P;

VARIABLE	DESCRIPTION
	the curve ID, LCSR; EPS1-EPS8 and ES1-ES8 are ignored if a Table ID is defined.
LCSR	Load curve ID defining strain rate scaling effect on yield stress.
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

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

where $\dot{\varepsilon}$ is the strain rate. $\dot{\varepsilon} = \sqrt{\dot{\varepsilon}_{ij}\dot{\varepsilon}_{ij}}$.

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

III. If different stress versus strain curves can be provided for various strain rates, the option using the reference to a table (LCSS) can be used. Then the table input in *DEFINE_TABLE has to be used, see Figure 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	none	none	none	none	none	none	

VARIABLE	DESCRIPTION
MID	Material identification. A unique number must be chosen.
RO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.
SIGY	Stress coefficient .
N	Stress exponent.
M	Time exponent.

Remarks:

The effective creep strain, $\bar{\varepsilon}^c$, given as:

$$\bar{\varepsilon}^c = A\bar{\sigma}^n \bar{t}^m$$

where A, n, and m are constants and $\bar{t}$ is the effective time. The effective stress, $\bar{\sigma}$, is defined as:

$$\bar{\sigma} = \sqrt{\frac{3}{2}\sigma_{ij}\sigma_{ij}}$$

The creep strain, therefore, is only a function of the deviatoric stresses. The volumetric behavior for this material is assumed to be elastic. By varying the time constant m primary creep ($m<1$), secondary creep ($m=1$), and tertiary creep ($m>1$) can be modeled. This model is described by Whirley and Henshall (1992).

*MAT_COMPOSITE_LAYUP

This is Material Type 116. This material is for modeling the elastic responses of composite layups that have an arbitrary number of layers through the shell thickness. A pre-integration is used to compute the extensional, bending, and coupling 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.

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

Card 2	1	2	3	4	5	6	7	8
Variable	GAB	GBC	GCA	AOPT				
Type	F	F	F	F				

Card 3

Card 3	1	2	3	4	5	6	7	8
Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card 4

Card 4	1	2	3	4	5	6	7	8
Variable	V1	V2	V3	D1	D2	D3	BETA	
Type	F	F	F	F	F	F	F	

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: 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.
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:

$$\begin{Bmatrix} N_x \\ N_y \\ N_{xy} \end{Bmatrix} = \begin{bmatrix} A_{11} & A_{12} & A_{16} \\ A_{21} & A_{22} & A_{26} \\ A_{16} & A_{26} & A_{66} \end{bmatrix} \begin{Bmatrix} \varepsilon_x^0 \\ \varepsilon_y^0 \\ \varepsilon_z^0 \end{Bmatrix} + \begin{bmatrix} B_{11} & B_{12} & B_{16} \\ B_{21} & B_{22} & B_{26} \\ B_{16} & B_{26} & B_{66} \end{bmatrix} \begin{Bmatrix} \kappa_x \\ \kappa_y \\ \kappa_{xy} \end{Bmatrix}$$

$$\begin{Bmatrix} M_x \\ M_y \\ M_{xy} \end{Bmatrix} = \begin{bmatrix} B_{11} & B_{12} & B_{16} \\ B_{21} & B_{22} & B_{26} \\ B_{16} & B_{26} & B_{66} \end{bmatrix} \begin{Bmatrix} \varepsilon_x^0 \\ \varepsilon_y^0 \\ \varepsilon_z^0 \end{Bmatrix} + \begin{bmatrix} D_{11} & D_{12} & D_{16} \\ D_{21} & D_{22} & D_{26} \\ D_{16} & D_{26} & D_{66} \end{bmatrix} \begin{Bmatrix} \kappa_x \\ \kappa_y \\ \kappa_{xy} \end{Bmatrix}$$

where A_{ij} is the extensional stiffness, D_{ij} is the bending stiffnes, and B_{ij} is the coupling stiffness which is a null matrix for symmetric lay-ups. The mid-surface stains and curvatures are denoted by ε_{ij}^0 and κ_{ij}, respectively. Since these stiffness matrices are symmetric, 18 terms are needed per shell element in addition to the shell resulants 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	

VARIABLE	DESCRIPTION
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 **v** with the element normal. The plane of a solid element is the midsurface between the inner surface and outer surface defined by the first four nodes and the last four nodes of the connectivity of the element, respectively.

VARIABLE	DESCRIPTION
	EQ. 4.0: locally orthotropic in cylindrical coordinate system with the material axes determined by a vector **v**, and an originating point, P, which define the centerline axis. This option is for solid elements only.
XP YP ZP	Define coordinates of point p for AOPT = 1 and 4.
A1 A2 A3	Define components of vector a for AOPT = 2.
V1 V2 V3	Define components of vector v for AOPT = 3 and 4.
D1 D2 D3	Define components of vector d for AOPT = 2:
BETA	Material angle in degrees for AOPT = 3, may be overridden on the element card, see *ELEMENT_SHELL_BETA.

Remarks:

The calculation of the force, N_{ij}, and moment, M_{ij}, stress resultants is given in terms of the membrane strains, ε_i^0, and shell curvatures, κ_i, as:

$$\begin{Bmatrix} N_x \\ N_y \\ N_{xy} \\ M_x \\ M_y \\ M_{xy} \end{Bmatrix} = \begin{bmatrix} C_{11} & C_{12} & C_{13} & C_{14} & C_{15} & C_{16} \\ C_{21} & C_{22} & C_{23} & C_{24} & C_{25} & C_{26} \\ C_{31} & C_{32} & C_{33} & C_{34} & C_{35} & C_{36} \\ C_{41} & C_{42} & C_{43} & C_{44} & C_{45} & C_{46} \\ C_{51} & C_{52} & C_{53} & C_{54} & C_{55} & C_{56} \\ C_{61} & C_{62} & C_{63} & C_{64} & C_{65} & C_{66} \end{bmatrix} \begin{Bmatrix} \varepsilon_x^0 \\ \varepsilon_y^0 \\ \varepsilon_z^0 \\ \kappa_x \\ \kappa_y \\ \kappa_{xy} \end{Bmatrix}$$

where $C_{ij} = C_{ji}$.. In this model this symmetric matrix is transformed into the element local system and the coefficients are stored as element history variables. In model type *MAT_COMPOSITE _DIRECT below, the resultants are already assumed to be given in the element local system which reduces the storage since the 21 coefficients are not stored as history variables as part of the element data.

The shell thickness is built into the coefficient matrix and, consequently, within the part ID, which references this material ID, the thickness must be uniform.

*MAT_COMPOSITE_DIRECT

This is Material Type 118. This material is used for modeling the elastic responses of composites where a pre-integration is used to compute the extensional, bending, and coupling stiffness coefficients for use with the Belytschko-Tsay resultant shell formulation. Since the stresses are not computed in the resultant formulation, the stresses output to the binary databases for the resultant elements are zero.

Card 1	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			

VARIABLE	DESCRIPTION
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:

$$\begin{Bmatrix} N_x \\ N_y \\ N_{xy} \\ M_x \\ M_y \\ M_{xy} \end{Bmatrix} = \begin{bmatrix} C_{11} & C_{12} & C_{13} & C_{14} & C_{15} & C_{16} \\ C_{21} & C_{22} & C_{23} & C_{24} & C_{25} & C_{26} \\ C_{31} & C_{32} & C_{33} & C_{34} & C_{35} & C_{36} \\ C_{41} & C_{42} & C_{43} & C_{44} & C_{45} & C_{46} \\ C_{51} & C_{52} & C_{53} & C_{54} & C_{55} & C_{56} \\ C_{61} & C_{62} & C_{63} & C_{64} & C_{65} & C_{66} \end{bmatrix} \begin{Bmatrix} \varepsilon_x^0 \\ \varepsilon_y^0 \\ \varepsilon_z^0 \\ \kappa_x \\ \kappa_y \\ \kappa_{xy} \end{Bmatrix}$$

where $C_{ij} = C_{ji}$. In this model the stiffness coefficients are already assumed to be given in the element local system which reduces the storage. Great care in the element orientation and choice of the local element system, see *CONTROL_ACCURACY, must be observed if this model is used.

The shell thickness is built into the coefficient matrix and, consequently, within the part ID, which references this material ID, the thickness must be uniform.

*MAT_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 folowing 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					

VARIABLE	DESCRIPTION
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 nucleatiing 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.
LCSS	Load curve ID defining effective stress versus effective plastic strain. ATYP is ignored with this option.

VARIABLE	DESCRIPTION
LCLF	Load curve ID definingfailure 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 defeined as:

$$\Phi = \frac{\sigma_M^2}{\sigma_Y^2} + 2q_1 f^* \cosh\left(\frac{3q_2\sigma_H}{2\sigma_M}\right) - 1 - \left(q_1 f^*\right)^2 = 0$$

where σ_M is the equivalent von Mises stress, σ_Y is the Yield stress, σ_H is the mean hydrostatic stress. The effective void volume fraction is defined as

$$f^*(f) = \begin{cases} f & f \le f_c \\ f_c + \dfrac{1/q_1 - f_c}{f_F - f_c}\left(f - f_c\right) & f > f_c \end{cases}$$

The grow of void volume fraction is defined as

$$\dot{f} = \dot{f}_G + \dot{f}_N$$

where the crow of existing voids is deined as

$$\dot{f}_G = (1 - f)\dot{\varepsilon}_{kk}^p$$

and nucleation of new voids is defined as

$$\dot{f}_N = A\dot{\varepsilon}_P$$

where is defined as

$$A = \frac{f_N}{S_N\sqrt{2\pi}} \exp\left(-\frac{1}{2}\left(\frac{\varepsilon_p - \varepsilon_N}{S_N}\right)^2\right)$$

*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 effecitve plastic strain, plastic thinning, the major principal in plane strain component, or a minimum time step size. See the discussion under the model description for MAT_PIECEWISE_LINEAR_PLASTICITY if more information is desired.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	SIGY	ETAN	FAIL	TDEL
Type	I	F	F	F	F	F	F	F
Default	none	none	none	none	none	0.0	10.E+20	0

Card 2								
Variable	C	P	LCSS	LCSR	VP	EPSTHIN	EPSMAJ	NUMINT
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

Card 3								
Variable	EPS1	EPS2	EPS3	EPS4	EPS5	EPS6	EPS7	EPS8
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

Card 4

Variable	ES1	ES2	ES3	ES4	ES5	ES6	ES7	ES8
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.
SIGY	Yield stress.
ETAN	Tangent modulus, ignored if (LCSS.GT.0) is defined.
FAIL	Failure flag. LT.0.0: User defined failure subroutine is called to determine failure EQ.0.0: Failure is not considered. This option is recommended if failure is not of interest since many caluculations will be saved. GT.0.0: Plastic strain to failure. When the plastic strain reaches this value, the element is deleted from the calculation.
TDEL	Minimum time step size for automatic element deletion.
C	Strain rate parameter, C, see formula below.
P	Strain rate parameter, P, see formula below.
LCSS	Load curve ID or Table ID. Load curve ID defining effective stress versus effective plastic strain. If defined EPS1-EPS8 and ES1-ES8 are ignored. The table ID defines for each strain rate value a load curve ID giving the stress versus effectiveplastic strain for that rate, See Figure 20.7. The stress versus effective plastic strain curve for the lowest value of strain rate is used if the strain rate falls below the minimum value. Likewise, the stress versus effective plastic strain curve for the highest value of strain rate is used if the strain rate exceeds the maximum value. The strain rate parameters: C and P;

VARIABLE	DESCRIPTION
	the curve ID, LCSR; EPS1-EPS8 and ES1-ES8 are ignored if a Table ID is defined.
LCSR	Load curve ID defining strain rate scaling effect on yield stress.
VP	Formulation for rate effects (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 caluculations will be saved. GT.0.0: Plastic strain to failure. When the plastic strain reaches this value, the element is deleted from the calculation.
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

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

where $\dot{\varepsilon}$ is the strain rate. $\dot{\varepsilon}=\sqrt{\dot{\varepsilon}_{ij}\dot{\varepsilon}_{ij}}$

*MAT_MODIFIED_HONEYCOMB

This is Material Type 126. The major use of this material model is for honeycomb and foam materials with real anisotropic behavior. A nonlinear elastoplastic material behavior can be defined separately for all normal and shear stresses. These are considered to be fully uncoupled. See notes below.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	SIGY	VF	MU	BULK
Type	I	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	.05	0.0

Card 2								
Variable	LCA	LCB	LCC	LCS	LCAB	LCBC	LCCA	LCSR
Type	F	F	F	F	F	F	F	F
Default	none	LCA	LCA	LCA	LCS	LCS	LCS	optional

Card 3	1	2	3	4	5	6	7	8
Variable	EAAU	EBBU	ECCU	GABU	GBCU	GCAU	AOPT	
Type	F	F	F	F	F	F		

Card 4								
Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card 5

Variable	D1	D2	D3	TSEF	SSEF	VREF	TREF	
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 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 μ=0 This will give results identical to previous versions of LS-DYNA.
LCA	Load curve ID, see *DEFINE_CURVE, for sigma-aa versus normal strain component aa. For the corotational solid elements, types 0 and 9, engineering strain is expected, but for all other solid element formulations a logarithmic strain is expected. See notes below.
LCB	Load curve ID, see *DEFINE_CURVE, for sigma-bb versus normal strain component bb. For the corotational solid elements, types 0 and 9, engineering strain is expected, but for all other solid element formulations a logarithmic strain is expected. Default LCB=LCA. See notes below.
LCC	Load curve ID, see *DEFINE_CURVE, for sigma-cc versus normal strain component cc. For the corotational solid elements, types 0 and 9, engineering strain is expected, but for all other solid element formulations a logarithmic strain is expected. Default LCC=LCA. See notes below.

VARIABLE	DESCRIPTION
LCS	Load curve ID, see *DEFINE_CURVE, for shear stress versus shear strain. For the corotational solid elements, types 0 and 9, engineering strain is expected, but for all other solid element formulations a shear strain based on the deformed configuration is used.. Default LCS=LCA. Each component of shear stress may have its own load curve. See notes below.
LCAB	Load curve ID, see *DEFINE_CURVE, for sigma-ab versus shear strain-ab. For the corotational solid elements, types 0 and 9, engineering strain is expected, but for all other solid element formulations a shear strain based on the deformed configuration is used.. Default LCAB=LCS. See notes below.
LCBC	Load curve ID, see *DEFINE_CURVE, for sigma-bc versus shear strain-bc. For the corotational solid elements, types 0 and 9, engineering strain is expected, but for all other solid element formulations a shear strain based on the deformed configuration is used.. Default LCBC=LCS. See notes below.
LCCA	Load curve ID, see *DEFINE_CURVE, or sigma-ca versus shear strain-ca. For the corotational solid elements, types 0 and 9, engineering strain is expected, but for all other solid element formulations a shear strain based on the deformed configuration is used.. Default LCCA=LCS. See notes below.
LCRS	Load curve ID, see *DEFINE_CURVE, for strain-rate effects defining the scale factor versus strain rate $\dot{\bar{\varepsilon}} = \sqrt{\ }$. 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.

VARIABLE	DESCRIPTION
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 abcissa. If this recommendation is followed the cost of the table lookup is insignificant. Conversely, the cost increases significantly if the abcissa strain values are not consistent between load curves.

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 modulii vary from their initial values to the fully compacted values linearly with the relative volume:

$$E_{aa} = E_{aau} + \beta(E - E_{aau}) \qquad G_{ab} = G_{abu} + \beta(G - G_{abu})$$

$$E_{bb} = E_{bbu} + \beta(E - E_{bbu}) \qquad G_{bc} = G_{bcu} + \beta(G - G_{bcu})$$

$$E_{cc} = E_{ccu} + \beta(E - E_{ccu}) \qquad G_{ca} = G_{cau} + \beta(G - G_{cau})$$

where

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

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

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

The relative volume, V, is defined as the ratio of the current volume over the initial volume, and typically, V=1 at the beginning of a calculation.

For corotational solid elements, types 0 and 9, the components of the stress tensor remain uncoupled and the uncompressed elastic modulii are used, that is, the fully compacted elastic modulii 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 modulii according to:

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

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

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

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

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

then

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

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

For fully compacted material (element formulations 1 and 2), we assume that the material behavior is elastic-perfectly plastic and updated the stress components according to:

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

where the deviatoric strain increment is defined as

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

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

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

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

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

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

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

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

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.

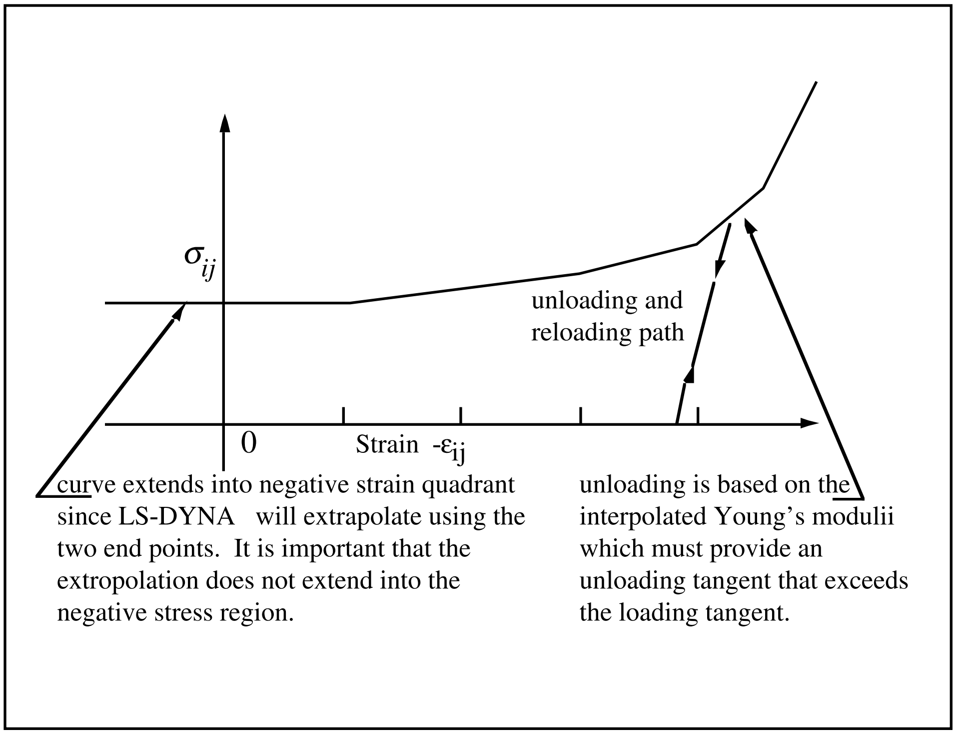


Figure 20.33 Stress quantity versus strain. Note that the "yield stress" at a strain of zero is nonzero. In the load curve definition the "time" value is the directional strain and the "function" value is the yield stress. Note that for element types 0 and 9 engineering strains are used, but for all 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 $\beta\iota$ 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]:

$$W(J_1,J_2,J) = nk\theta\left[\tfrac{1}{2}(J_1-3)+\tfrac{1}{20N}(J_1^2-9)+\tfrac{11}{1050N^2}(J_1^3-27)\right]$$

$$+nk\theta\left[\tfrac{19}{7000N^3}(J_1^4-81)+\tfrac{519}{673750N^4}(J_1^5-243)\right]+W_H(J)$$

$$J_1 = I_1 J^{-1/3}$$

$$J_2 = I_2 J$$

where the hydrostatic work term is in terms of the bulk modulus, K, and the third invarient J, as:

$$W_H(J) = \frac{K}{2}(J-1)^2$$

Rate effects are taken into accounted through linear viscoelasticity by a convolution integral of the form:

$$\sigma_{ij} = \int_0^t g_{ijkl}(t-\tau)\frac{\partial \varepsilon_{kl}}{\partial \tau}d\tau$$

or in terms of the second Piola-Kirchhoff stress, S_{ij}, and Green's strain tensor, E_{ij},

$$S_{ij} = \int_0^t G_{ijkl}(t-\tau)\frac{\partial E_{kl}}{\partial \tau}d\tau$$

where $g_{ijkl}(t-\tau)$ and $G_{ijkl}(t-\tau)$ are the relaxation functions for the different stress measures. This stress is addedto the stress tensor determined from the strain energy functional.

If we wish to include only simple rate effects, the relaxation function is represented by six terms from the Prony series:

$$g(t) = \alpha_0 + \sum_{m=1}^{N} \alpha_m e^{-\beta t}$$

given by,

$$g(t) = \sum_{i=1}^{n} G_i e^{-\beta_i t}$$

This model is effectively a Maxwell fluid which consists of a dampers and springs in series. We characterize this in the input by shear modulii, G_i, and decay constants, β_i. The viscoelastic behavior is optional and an arbitrary number of terms may be used.

*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 **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.
XP,YP,ZP	x_p y_p z_p, define coordinates of point **p** for AOPT = 1 and 4.
A1,A2,A3	a_1 a_2 a_3, define components of vector **a** for AOPT = 2.
D1,D2,D3	d_1 d_2 d_3, define components of vector **d** for AOPT = 2.
V1,V2,V3	v_1 v_2 v_3, define components of vector **v** for AOPT = 3 and 4.
BETA	Material angle in degrees for AOPT = 3, may be overridden on the element card, see *ELEMENT_SOLID_ORTHO.

Remarks:

The tissue model is described in terms of the energy functional in terms of the Green strain components, E_{ij},

$$W(E) = \tfrac{C}{2}(e^{Q} - 1) + \frac{1}{2}P\left(I_3 - 1\right)$$

$$Q = b_1 E_{11}^2 + b_2(E_{22}^2 + E_{33}^2 + E_{23}^2 + E_{32}^2) + b_3(E_{12}^2 + E_{21}^2 + E_{13}^2 + E_{31}^2)$$

The Green components are modified to eliminate any effects of volumetric work following the procedures of Ogden.

*MAT_LUNG_TISSUE

This is Material Type 129. This material model provides a hyperelastic model for heart tissue, see [Vawter, 1980] combined optionally with linear viscoelasticity as outlined by [Christensen 1980].

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	K	C	DELTA	ALPHA	BETA	
Type	I	F	F	F	I			

Card 2	1	2	3	4	5	6	7	8
Variable	C1	C2	LCID	TRAMP	NT			
Type	F	F	F	F	F			

Card Format for Viscoelastic Constants. Up to 6 cards may be input. A keyword card (with a "*" in column 1) terminates this input if less than 6 cards are used.

Optional Cards	1	2	3	4	5	6	7	8
Variable	GI	BETAI						
Type	F	F						

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density
K	Bulk modulus
C	Material coefficient.
DELTA	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 $\beta\iota$ 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:

$$W(I_1,I_2)=\frac{C}{2\Delta}e^{(\alpha I_1^2+\beta I_2)}+\frac{12C_1}{\Delta(1+C_2)}\left[A^{(1+C_2)}-1\right]$$

$$A^2=\tfrac{4}{3}\left(I_1+I_2\right)-1$$

where the hydrostatic work term is in terms of the bulk modulus, K, and the third invariant J, as:

$$W_H(J) = \frac{K}{2}(J-1)^2$$

Rate effects are taken into accounted through linear viscoelasticity by a convolution integral of the form:

$$\sigma_{ij} = \int_0^t g_{ijkl}(t-\tau)\frac{\partial \varepsilon_{kl}}{\partial \tau}d\tau$$

or in terms of the second Piola-Kirchhoff stress, S_{ij}, and Green's strain tensor, E_{ij},

$$S_{ij} = \int_0^t G_{ijkl}(t-\tau)\frac{\partial E_{kl}}{\partial \tau}d\tau$$

where $g_{ijkl}(t-\tau)$ and $G_{ijkl}(t-\tau)$ are the relaxation functions for the different stress measures. This stress is addedto the stress tensor determined from the strain energy functional.

If we wish to include only simple rate effects, the relaxation function is represented by six terms from the Prony series:

$$g(t) = \alpha_0 + \sum_{m=1}^{N} \alpha_m e^{-\beta t}$$

given by,

$$g(t) = \sum_{i=1}^{n} G_i e^{-\beta_i t}$$

This model is effectively a Maxwell fluid which consists of a dampers and springs in series. We characterize this in the input by shear modulii, G_i, and decay constants, β_i. The viscoelastic behavior is optional and an arbitrary number of terms may be used.

*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.

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	

VARIABLE	DESCRIPTION
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	ν_{12p}, for in plane behavior.
V11P	ν_{21p}, for in plane behavior.
G12P	G_{12p}, for in plane behavior.
G23P	G_{23p}, for in plane behavior.
G31P	G_{31p}, for in plane behavior.
E11B	E_{11b}, for bending behavior.
E22B	E_{22b}, for bending behavior.
V12B	ν_{12b}, for bending behavior.
V21B	ν_{21b}, for bending behavior.
G12B	G_{12b}, for bending behavior.

VARIABLE	DESCRIPTION
AOPT	Material axes option (see MAT_*OPTION* TROPIC_ELASTIC for a more complete description): EQ. 0.0: locally orthotropic with material axes determined by element nodes 1, 2, and 4, as with *DEFINE_ COORDINATE_NODES. EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR. EQ. 3.0: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle, BETA, from a line in the plane of the element defined by the cross product of the vector **v** with the element normal.
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 trasmitted 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.

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)
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

$$M_{r_{upper}} = MAX\left(M_r,\frac{M_{r_{yield}}}{2}\right)$$

and similar for M_s and M_t.

Thereafter the plastic moments will be given by

M_{rp}, = min (M_{rupper}, M_{rcurve}) and similar for s and t

where

M_{rp} = current plastic moment

M_{rcurve} = moment taken from load curve at the current rotation scaled according to the scale factor.

The effect of this is to provide an upper limit to the moment that can be generated; it represents the softening effect of local buckling at a hinge site. Thus if a member is bent about is local s-axis it will then be weaker in torsion and about its local t-axis. For moments-softening curves, the effect is to trim off the initial peak (although if the curves subsequently harden, the final hardening will also be trimmed off).

It is not possible to make the plastic moment vary with the current axial load, but it is possible to make hinge formation a function of axial load and subsequent plastic mement 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}.

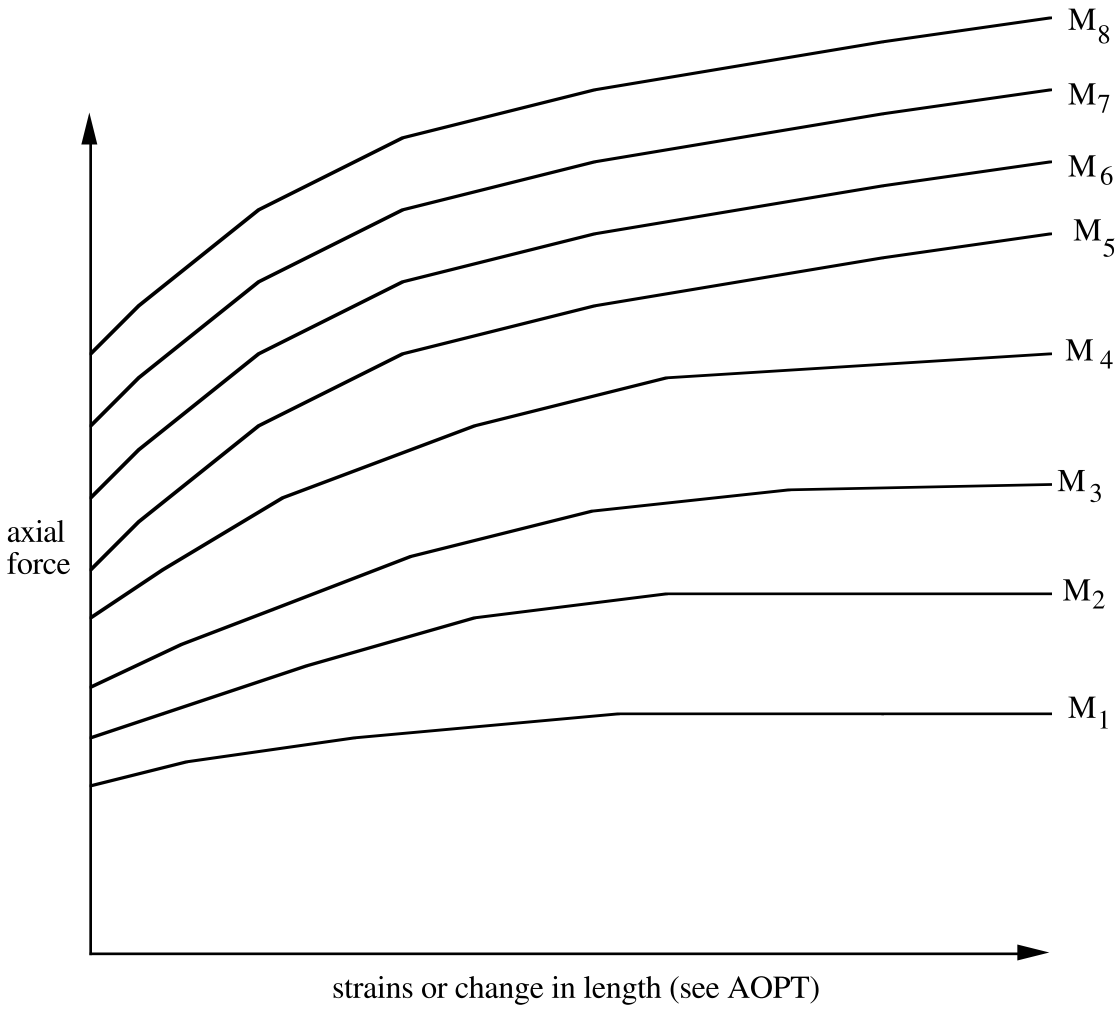


Figure 20.12. The force magnitude is limited by the applied end moment. For an intermediate value of the end moment LS-DYNA interpolates between the curves to determine the allowable force value.

*MAT_COMPOSITE_MSC

This is Material Type 161. This model may be used to model the progressive failure analysis for composite materials consisting of unidirectional and woven fabric layers. The progressive layer failure criteria has been established by adopting the methodology developed by Hashin [1980] with a generalization to include the effect of highly constrained pressure on composite failure. This failure model can be used to effectively simulate fiber failure, matrix damage, and delamination behavior under all conditions-opening, closure, and sliding of failure surfaces. Furthermore, this progressive failure modeling approach is advantageous as it enables one to predict delamination when locations of delamination sites cannot be anticipated. This model is implemented for single integration point brick elements only. This model requires an additional license from Material Sciences Corporation, who have developed and support this model with material data.

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	EEXPN	CERATE				
Type	F	F	F	F				

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density
EA	E_a, Young's modulus - longitudinal direction
EB	E_b, Young's modulus - transverse direction
EC	E_c, Young's modulus – through thickness direction
PRBA	ν_{ba}, Poisson's ratio ba

VARIABLE	DESCRIPTION
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.
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

VARIABLE	DESCRIPTION
AMODEL	Material models: EQ. 1: Unidirectional layer model EQ. 2: Fabric layer model
PHIC	Coulomb friction angle for matrix and delamination failure
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
CERATE	Coefficient for strain rate dependent strength properties

Material Models

The unidirectional layer failure criteria and the associated property degradation models are described as follows. Note that all failure criteria are expressed in terms of stress components based on ply level stresses $(\sigma_a, \sigma_b, \sigma_c, \tau_{ab}, \tau_{bc}, \tau_{ca})$ with a, b and c denoting the fiber, in-plane transverse and out-of-plane directions, respectively.

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):

$$f_5 = S\left\{\left(\frac{\langle\sigma_c\rangle}{S_{bT}}\right)^2 + \left(\frac{\tau_{bc}}{S_{bc}^{''}}\right)^2 + \left(\frac{\tau_{ca}}{S_{ca}}\right)^2\right\} - 1 = 0$$

where S_{bT} is the transverse tensile. 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 j is a material constant as tan(j) 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 s_c, t_{bc} and t_{ca} to the fractured material strength surface. For tensile mode, $s_c > 0$, these stress components are reduced to zero. For compressive mode, $s_c < 0$, the normal stress s_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 t_{bc} and t_{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, $s_b>0$, s_b, t_{ab} and t_{bc} are zero. For compressive mode, $s_b <0$, s_b is assumed to be elastic, while t_{ab} and t_{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 t_{bc} is governed by both the failure functions and should lie within or on each of these two strength surfaces.

Failure criteria based on the 3D stresses in a plain weave composite layer with improved progressive failure modeling capability have also been established following the same approach as that for the unidirectional model. Note that the fabric failure criteria are expressed in terms of stress components based on ply level stresses $\left(\sigma_a,\sigma_b,\sigma_c,\tau_{ab},\tau_{bc},\tau_{ca}\right)$ with a, b and c denoting the in-plane fill, in-plane warp and out-of-plane directions, respectively.

The fill and warp fiber tensile/shear failure are given by the quadratic interaction between the associated axial and shear stresses, i.e.

$$f_6=\left(\frac{\langle\sigma_a\rangle}{S_{aT}}\right)^2+\frac{\left(\tau_{ab}^2+\tau_{ca}^2\right)}{S_{aFS}^2}-1=0$$

$$f_7=\left(\frac{\langle\sigma_b\rangle}{S_{bT}}\right)^2+\frac{\left(\tau_{ab}^2+\tau_{bc}^2\right)}{S_{bFS}^2}-1=0$$

where S_{aT} and S_{bT} are the axial tensile strengths in the fill and warp directions, respectively, and S_{aFS} and S_{bFS} are the layer shear strengths due to fiber shear failure in the fill and warp directions. These failure criteria are applicable when the associated σ_a or σ_b is positive. It is assumed S_{aFS}= SFS, and

$$S_{bFS}=SFS*S_{bT}/S_{aT}.$$

When σ_a or σ_b is compressive, it is assumed that the in-plane compressive failure in both the fill and warp directions are given by the maximum stress criterion, i.e.

$$f_8 = \left[\frac{\langle \sigma_a^{'} \rangle}{S_{aC}} \right]^2 - 1 = 0, \quad \sigma_a^{'} = -\sigma_a + \langle -\sigma_c \rangle$$

$$f_9 = \left[\frac{\langle \sigma_b^{'} \rangle}{S_{bC}} \right]^2 - 1 = 0, \quad \sigma_b^{'} = -\sigma_b + \langle -\sigma_c \rangle$$

where S_{aC} and S_{bC} are the axial compressive strengths in the fill and warp directions, respectively. The crush failure under compressive pressure is

$$f_{10} = \left(\frac{\langle p \rangle}{S_{FC}} \right)^2 - 1 = 0, \quad p = -\frac{\sigma_a + \sigma_b + \sigma_c}{3}$$

A plain weave layer can fail under in-plane shear stress without the occurrence of fiber breakage. This in-plane matrix failure mode is given by

$$f_{11} = \left(\frac{\tau_{ab}}{S_{ab}} \right)^2 - 1 = 0$$

where S_{ab} is the layer shear strength due to matrix shear failure.

Another failure mode, which is due to the quadratic interaction between the thickness stresses, is expected to be mainly a matrix failure. This through the thickness matrix failure criterion is

$$f_{12} = S^2 \left\{ \left(\frac{\langle \sigma_c \rangle}{S_{cT}} \right)^2 + \left(\frac{\tau_{bc}}{S_{bc}} \right)^2 + \left(\frac{\tau_{ca}}{S_{ca}} \right)^2 \right\} - 1 = 0$$

where S_{cT} is the through the thickness tensile strength, and S_{bc}, and S_{ca} are the shear strengths assumed to depend on the compressive normal stress sc, i.e.,

$$\begin{Bmatrix} S_{ca} \\ S_{bc} \end{Bmatrix} = \begin{Bmatrix} S_{ca}^{(0)} \\ S_{bc}^{(0)} \end{Bmatrix} + \tan(\varphi) \langle -\sigma_c \rangle$$

When failure predicted by this criterion occurs within elements that are adjacent to the ply interface, the failure plane is expected to be parallel to the layering planes, and, thus, can be referred to as the delamination mode. Note that a scale factor S is introduced to provide better correlation of delamination area with experiments. The scale factor S can be determined by fitting the analytical prediction to experimental data for the delamination area.

Similar to the unidirectional model, when fiber tensile/shear failure is predicted in a layer by f_6 or f_7, the load carrying capacity of that layer in the associated direction is completely eliminated. For compressive fiber failure due to by f_8 or f_9, the layer is assumed to carry a residual axial load in the failed direction, while the load carrying capacity transverse to the failed direction is assumed unchanged. When the compressive axial stress in a layer reaches the compressive axial strength S_{aC} or S_{bC}, the axial layer stress is assumed to be reduced to the residual strength S_{aRC} or S_{bRC} where $S_{aRC} = SFFC * S_{aC}$ and $S_{bRC} = SFFC * S_{bC}$. The axial stress is assumed to remain constant, i.e., $\sigma_a = -S_{aCR}$ or $\sigma_b = -S_{bCR}$, for continuous compressive loading, while the subsequent unloading curve follows a reduced axial modulus. When the fiber crush failure is occurred, the material is assumed to behave elastically for compressive pressure, p > 0, and to carry no load for tensile pressure, p < 0.

When the in-plane matrix shear failure is predicted by f_{11} the axial load carrying capacity within a failed element is assumed unchanged, while the in-plane shear stress is assumed to be reduced to zero.

For through the thickness matrix (delamination) failure given by equations f_{12}, the in-plane load carrying capacity within the element is assumed to be elastic, while the strength values for the tensile mode, $S_A^{(0)}$ and $S_T^{(0)}$, are set to zero. For tensile mode, s_c >0, the through the thickness stress components are reduced to zero. For compressive mode, s_c <0, s_c is assumed to be elastic, while t_{bc} and t_{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 multiplying the associated strength values by a scale factor S_{RT} as

$$S_{RT} = 1 + C_{rate} \ln \frac{\dot{\bar{\varepsilon}}}{\dot{\varepsilon}_0}$$

where $\dot{\bar{\varepsilon}}$ is the effective strain rate for $\dot{\varepsilon}_0 = 1s^{-1}$.

Element Erosion

A failed element is eroded in any of four 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 volume strain in a failed element is smaller than ECRSH.

3. If tensile volume strain 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 1-failed	8	88
2. efb(I)	Fiber mode in b		9	89
3. efp(I)	Fiber crush mode		10	90
4. em(I)	Perpendicular matrix mode		11	91
5. ed(I)	Parallel matrix/ delamination mode		12	92

*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	see note	see note	see note	see note	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			

VARIABLE	DESCRIPTION
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.

VARIABLE	DESCRIPTION
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 <u>all</u> 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 initilization 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

VARIABLE	DESCRIPTION
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 tolenrance 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 maxuimum over buden 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

VARIABLE	DESCRIPTION
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(1psi in the model stress units). This is needed because the ultimate tensile stress of concrete is expessed as SQRT(fc=) where fc= is in psi. Therefore a unit conversion factor of sqrt(psi/stress unit) is required. Examples: UCONV = 0.083 if stress unit is MN/m2 or N/mm2 UCONV = 83.3 if stress unit is N/m2
ALPHA	Shear span factor - see below.
FT	Cracking stress in direct tension - see notes below. Default is 8% of the cylinder strength.

VARIABLE	DESCRIPTION
ERIENF	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:

max shear stress (UBC) = Vu/Acv = uconv.alpha.sqrt(fc=) + ro.fy

where,

uconv = unit conversion factor, 0.083 for SI units (MN)

Alpha = aspect ratio, = 2.0 unless ratio h/l < 2.0 in which case alpha varies linearly from 2.0 at h/l=2.0 to 3.0 at h/l=1.5.
Fc = unconfined compressive strength of concrete
ro = fraction of reiforcement = percent reinforcement/100
fy = yield stress of reinforcement

To this we add shear stress due to the overburden to obtain the ultimate shear stress:

tmax = max shear stress (UBC) + sig0

where

sig0 = in-plane compressive stress under static equilibrium conditions

The UBC formula for ultimate shear stress is generally conservative (predicts that the wall is weaker than shown in test), sometimes by 50% or more. A less conservative formula is that of Fukuzawa:

tmax = a1*2.7*(1.9-M/LV)*UCONV*sqrt(fc=) + ro*fy*0.5 + sig0

where

a1 = max((0.4 + Ac/Aw),1.0)
Ac/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 0.3tu, strain=0.3tu/G where G is the elastic shear modulus given by E/2(1+v)

Second point (reinforcement yield) at 0.5yu, 0.8tmax

Third point (ultimate strength) at yu, tmax

Fourth point (onset of strength reduction) at 2yu, tmax

Fifth point (failure) at 3yu, 0.6tmax.

After failure, the shear stress drops to zero. The curve points can be entered by the user if desired, in which case they over-ride the automatically calculated curve. However, it is anticipated that in most cases the default curve will be preferred due to ease of input.

Hysteresis follows the algorithm of Shiga as for the squat 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 vales 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 ft (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 enetered, no cracking occurs in the model. As a minimum, fc=, % and fy 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					

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.
SIGY	Yield stress.
ETAN	Tangent modulus, ignored if (LCSS.GT.0) is defined.
FAIL	Failure flag. LT.0.0: user defined failure subroutine is called to determine failure EQ.0.0: failure is not considered. This option is recommended if failure is not of interest since many caluculations will be saved. GT.0.0: plastic strain to failure. When the plastic strain reaches this value, the element is deleted from the calculation.
TDEL	Minimum time step size for automatic element deletion.
C	Strain rate parameter, C, see formula below.
P	Strain rate parameter, P, see formula below.
LCSS	Load curve ID or Table ID. Load curve ID defining effective stress versus effective plastic strain. If defined EPS1-EPS8 and ES1-ES8 are ignored. The table ID defines for each strain rate value a load curve ID giving the stress versus effectiveplastic strain for that rate, See Figure 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

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

where $\dot{\varepsilon}$ is the strain rate. $\dot{\varepsilon}=\sqrt{\dot{\varepsilon}_{ij}\dot{\varepsilon}_{ij}}$.

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

III. If different stress versus strain curves can be provided for various strain rates, the option using the reference to a table (LCSS) can be used.

*MAT_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

VARIABLE	DESCRIPTION
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. (Opitonal).
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 SCOOR variable in *SECTION_BEAM).

For null stiffness coefficients, no forces corresponding to these null values will develop. The viscous damping coefficients are optional.

*MAT_NONLINEAR_ELASTIC_DISCRETE_BEAM

This is Material Type 67. This material model is defined for simulating the effects of nonlinear elastic and nonlinear viscous beams by using six springs each acting about one of the six local degrees-of-freedom. The two nodes defining a beam may be coincident to give a zero length beam, or offset to give a finite length beam. For finite length discrete beams the absolute value of the variable SCOOR in the SECTION_BEAM input should be set to a value of 2.0, which causes the local r-axis to be aligned along the two nodes of the beam to give physically correct behavior. The distance between the nodes of a beam should not affect the behavior of this material model. A triad is used to orient the beam for the directional springs. Arbitrary curves to model transitional/ rotational stiffness and damping effects are allowed. See notes below.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	LCIDTR	LCIDTS	LCIDTT	LCIDRR	LCIDRS	LCIDRT
Type	I	F	F	F	F	F	F	F

Card 2								
Variable	LCIDTDR	LCIDTDS	LCIDTDT	LCIDRDR	LCIDRDS	LCIDRDT		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density, see also volume in *SECTION_BEAM definition.
LCIDTR	Load curve ID defining translational force resultant along local r-axis versus relative translational displacement, see Remarks and Figure 20.19.
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.

VARIABLE	DESCRIPTION
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].

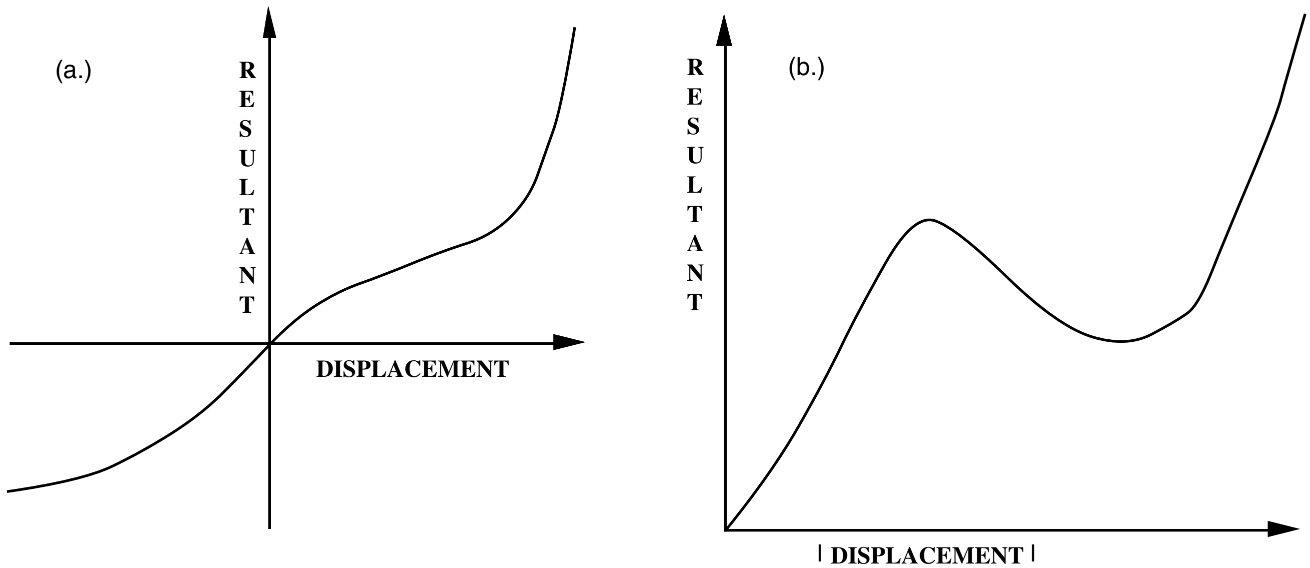


Figure 20.19. The resultant forces and moments are determined by a table lookup. If the origin of the load curve is at [0,0] as in (b.) and tension and compression responses are symmetric.

*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	none	none	none	none	none	none	none

Card 2								
Variable	TDR	TDS	TDT	RDR	RDS	RDT		
Type	F	F	F	F	F	F		
Default	none	none	none	none	none	none		

Card 3

Variable	LCPDR	LCPDS	LCPDT	LCPMR	LCPMS	LCPMT		
Type	F	F	F	F	F	F		
Default	0.0	0.0	0.0	0.0	0.0	0.0		

Card 4	1	2	3	4	5	6	7	8
Variable	FFAILR	FFAILS	FFAILT	MFAILR	MFAILS	MFAILT		
Type	F	F	F	F	F	F		
Default	0.0	0.0	0.0	0.0	0.0	0.0		

Card 5

Variable	UFAILR	UFAILS	UFAILT	TFAILR	TFAILS	TFAILT		
Type	F	F	F	F	F	F		
Default	0.0	0.0	0.0	0.0	0.0	0.0		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density, see also volume on *SECTION_BEAM definition.
TKR	Translational stiffness about local r-axis
TKS	Translational stiffness about local s-axis
TKT	Translational stiffness about local t-axis
RKR	Rotational stiffness about the local r-axis

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.

VARIABLE	DESCRIPTION
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.

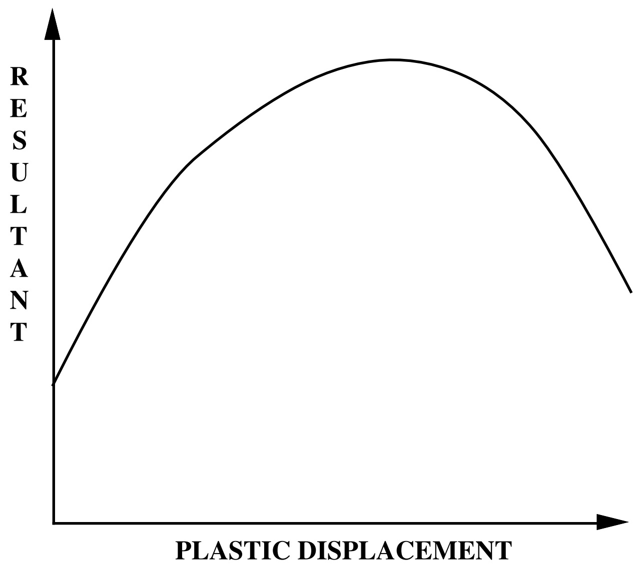


Figure 20.20. The resultant forces and moments are limited by the yield definition. The initial yield point corresponds to a plastic displacement of zero.

*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 parmeters 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 absolute value of the relative velocity.
C	C, discharge coefficient
C3	Coefficient for fluid inertia term
STF	k, stiffness coefficient if piston bottoms out
RHOF	ρ_{fluid}, fluid density
C1	C_1, coefficient for linear velocity term
C2	C_2, coefficient for quadratic velocity term
LCIDF	Load curve number ID defining force versus piston displacement, s, i.e., term $f(s+s_0)$. Compressive behavior is defined in the positive quadrant of the force displacement curve. Displacements falling outside of the defined force displacement curve are extrapolated. Care must be taken to ensure that extrapolated values are reasonable.
LCIDD	Load curve number ID defining damping coefficient versus piston displacement, s, i.e., $g(s+s_0)$. Displacements falling outside the defined curve are extrapolated. Care must be taken to ensure that extrapolated values are reasonable.
S0	Initial displacement s_0, typically set to zero. A positive displacement corresponds to compressive behavior.
ORFLOC	d_i, orifice location of ith orifice relative to the fixed end.
ORFRAD	r_i, orifice radius of ith orifice, if zero the default radius is used.

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,

$$F = SF\left\{KA_pV_p\left\{\frac{C_1}{A_0^t} + C_2\left|V_p\right|\rho_{fluid}\left[\left(\frac{A_p}{CA_0^t}\right)^2 - 1\right]\right\} - f\left(s+s_0\right) + V_p g(s+s_0)\right\}$$

where K is a user defined constant or a tabulated function of the absolute value of the relative velocity, V_p is the piston velocity, C is the discharge coefficient, A_p is the piston area, A_0^t is the total open areas of orifices at time t, ρ_{fluid} is the fluid density, C_1 is the coefficient for the linear term, and C_2 is the coefficient for the quadratic term.

In the implementation, the orifices are assumed to be circular with partial covering by the orifice controller. As the piston closes, the closure of the orifice is gradual. This gradual closure is properly taken into account to insure a smooth response. If the piston stroke is exceeded, the stiffness value, k, limits further movement, i.e., if the damper bottoms out in tension or compression the damper forces are calculated by replacing the damper by a bottoming out spring and damper, *k* and *c*, respectively. The piston stroke must exceed the initial length of the beam element. The time step calculation is based in part on the stiffness value of the bottoming out spring. A typical force versus displacement curve at constant relative velocity is shown in Figure 20.22.

The factor, *SF*, which scales the force defaults to 1.0 and is analogous to the adjusting ring on the damper.

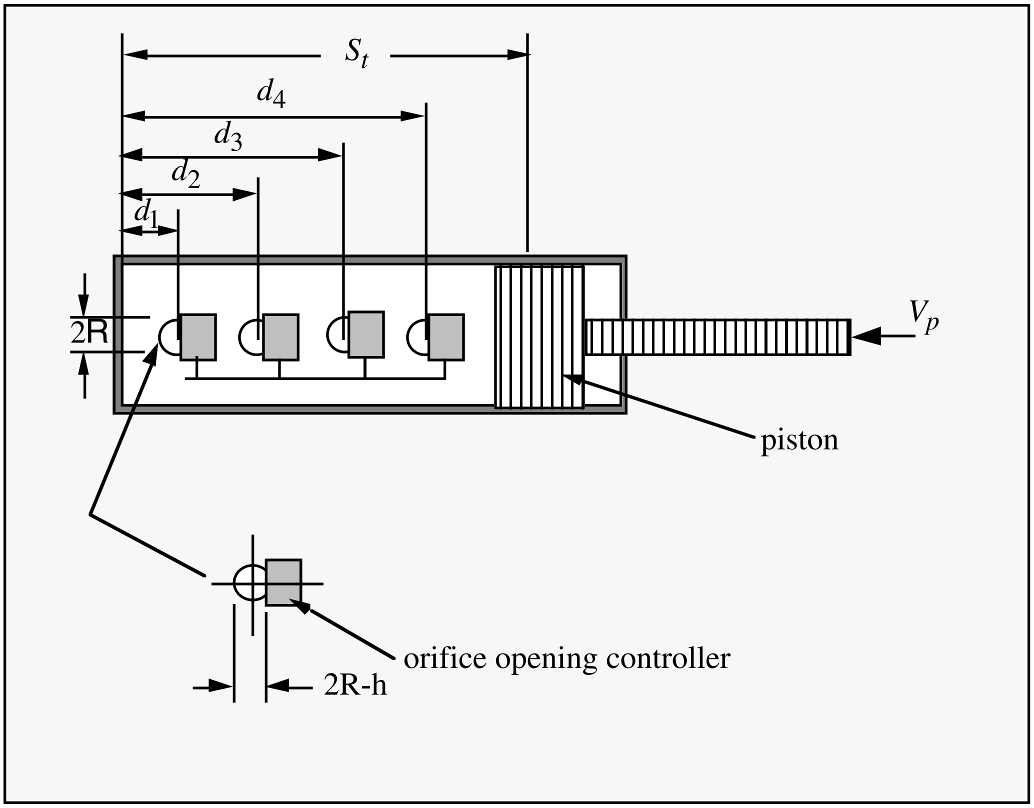


Figure 20.21. Mathematical model for the Side Impact Dummy damper.

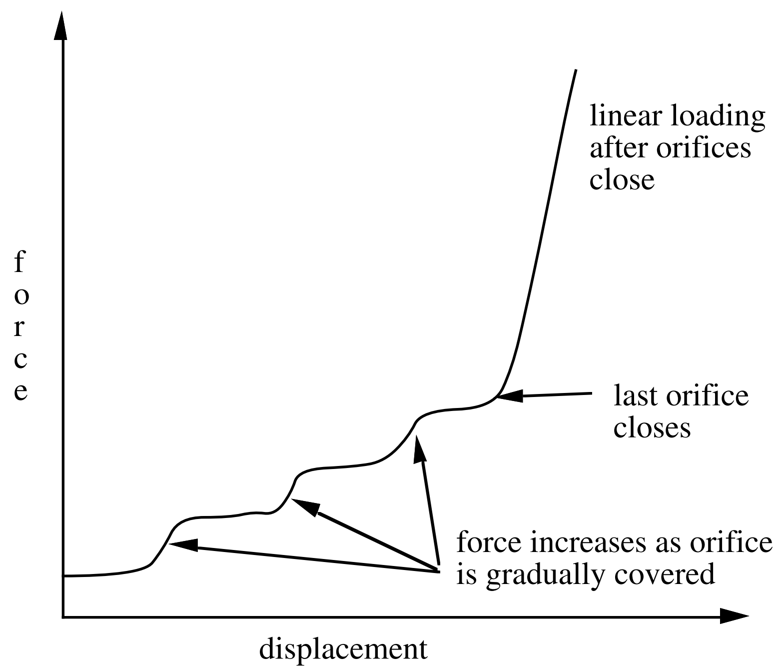


Figure 20.22. Force versus displacement as orifices are covered at a constant relative velocity. Only the linear velocity term is active.

*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				

VARIABLE	DESCRIPTION
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_o
N	Adiabatic constant
P0	Initial gas pressure, P_0
PA	Atmospheric pressure, P_a
AP	Piston cross sectional area, A_p
KH	Hydraulic constant, K

VARIABLE	DESCRIPTION
LCID	Load curve ID, see *DEFINE_CURVE, defining the orifice area, a_0, versus element deflection.
FR	Return factor on orifice force. This acts as a factor on the hydraulic force only and is applied when unloading. It is intended to represent a valve that opens when the piston unloads to relieve hydraulic pressure. Set it to 1.0 for no such relief.
SCLF	Scale factor on force. (Default = 1.0)
CLEAR	Clearance (if nonzero, no tensile force develops for positive displacements and negative forces develop only after the clearance is closed.

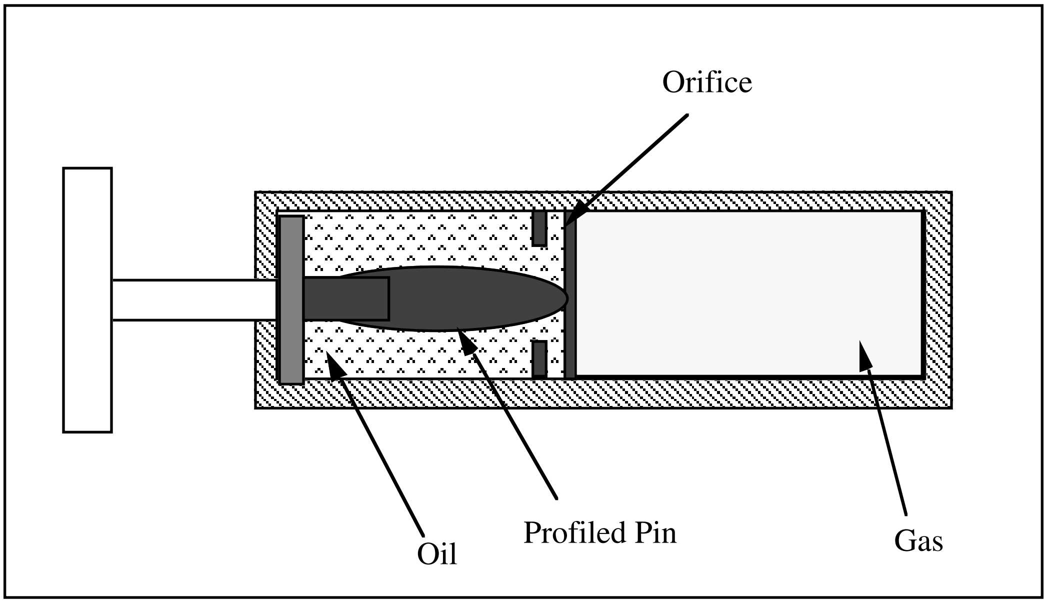


Figure 20.23. Schematic of Hydraulic/Gas damper.

Remarks:

As the damper is compressed two actions contribute to the force which developes. First, the gas is adiabatically compressed into a smaller volume. Secondly, oil is forced through an orifice. A profiled pin may occupy some of the cross-sectional area of the orifice; thus, the orifice area available for the oil varies with the stroke. The force is assumed proportional to the square of the velocity and inversely proportional to the available area.

The equation for this element is:

$$F = SCLF \cdot \left\{ K_h \left(\frac{V}{a_0} \right)^2 + \left[P_0 \left(\frac{C_0}{C_0 - S} \right)^n - P_a \right] \cdot A_p \right\}$$

where S is the element deflection and V is the relative velocity across the element.

*MAT_CABLE_DISCRETE_BEAM

This is Material Type 71. This model permits elastic cables to be realistically modelled; thus, no force will develop in compression.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E	LCID	F0			
Type	I	F	F	F	F			

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density, see also volume in *SECTION_BEAM definition.
E	GT. 0.0: Young's modulus LT. 0.0: Stiffness
LCID	Load curve ID, see *DEFINE_CURVE, defining the stress versus engineering strain. (Optional).
F0	Initial tensile force. If F0 is defined, an offset is not needed for an initial tensile force.

Remarks:

The force, F, generated by the cable is nonzero if and only if the cable is tension. The force is given by:

$$F = \max(F_0 + K\Delta L, 0.)$$

where ΔL is the change in length

$$\Delta L = current\ length - \left(initial\ length - offset\right)$$

and the stiffness (E > 0.0 only) is defined as:

$$K = \frac{E \cdot area}{\left(initial\ length - offset\right)}$$

Note that a constant force element can be obtained by setting:

$$F_0 > 0 \quad and \;\; K = 0$$

although the application of such an element is unknown.

The area and offset are defined on either the cross section or element cards. For a slack cable the offset should be input as a negative length. For an initial tensile force the offset should be positive.

If a load curve is specified the Young's modulus will be ignored and the load curve will be used instead. The points on the load curve are defined as engineering stress versus engineering strain, i.e., the change in length over the initial length. The unloading behavior follows the loading.

*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			
Type	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.
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 coefficien for nonlinear behavior (optional).
DLE	Factor to scale time units. The default is unity.

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:

$$F = F_0 + K\,f(\Delta L)\left[1 + C1\cdot\Delta\dot{L} + C2\cdot sgn\left(\Delta\dot{L}\right)\ln\left(\max\left\{1.,\frac{\left|\Delta\dot{L}\right|}{DLE}\right\}\right)\right] + D\Delta\dot{L} + h\left(\Delta\dot{L}\right)$$

In these equations, ΔL is the change in length

$$\Delta L = current\ length - initial\ length$$

The cross sectional area is defined on the section card for the discrete beam elements, See *SECTION_BEAM. The square root of this area is used as the contact thickness offset if these elements are included in the contact treatment.

*MAT_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

VARIABLE	DESCRIPTION
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			
Type	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.
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.

Remarks:

The yield force is taken from the load cuve:

$$F^Y = F_y\left(\Delta L^{plastic}\right)$$

where $L^{plastic}$ is the plastic deflection. A trial force is computed as:

$$F^T = F^n + K\Delta\dot{L}(\Delta t)$$

and is checked against the yield force to determine F:

$$F = \begin{cases} F^Y \; if \; F^T > F^Y \\ F^T \; if \; F^T \le F^Y \end{cases}$$

The final force, which includes rate effects and damping, is given by:

$$F^{n+1} = F \cdot \left[1 + C1 \cdot \Delta\dot{L} + C2 \cdot sgn\left(\Delta\dot{L}\right) \ln\left(\max\left\{ 1., \frac{\left|\Delta\dot{L}\right|}{DLE} \right\} \right) \right] + D\Delta\dot{L} + h\left(\Delta\dot{L}\right)$$

Unless the origin of the curve starts at (0,0), the negative part of the curve is used when the spring force is negative where the negative of the plastic displacement is used to interpolate, F_y. The positive part of the curve is used whenever the force is positive. In these equations, ΔL is the change in length

$$\Delta L = current\ length - initial\ length$$

The cross sectional area is defined on the section card for the discrete beam elements, See *SECTION_BEAM. The square root of this area is used as the contact thickness offset if these elements are included in the contact treatment.

*MAT_INELASTIC_6DOF_SPRING_DISCRETE_BEAM

This is Material Type 95. This material model is defined for simulating the effects of nonlinear inelastic and nonlinear viscous beams by using six springs each acting about one of the six local degrees-of-freedom. The input consists of part ID's that reference material type, *MAT_ INELASTIC_SPRING_DISCRETE_BEAM above (type 94). Generally, these referenced parts are used only for the definition of this material model and are not referenced by any elements. The two nodes defining a beam may be coincident to give a zero length beam, or offset to give a finite length beam. For finite length discrete beams the absolute value of the variable SCOOR in the SECTION_BEAM input should be set to a value of 2.0, which causes the local r-axis to be aligned along the two nodes of the beam to give physically correct behavior. The distance between the nodes of a beam should not affect the behavior of this material model. A triad must be used to orient the beam for zero length beams.

Card 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

VARIABLE	DESCRIPTION
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_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						

VARIABLE	DESCRIPTION
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				

VARIABLE	DESCRIPTION
MID	Material number. A unique number has to be chosen.
K	Elastic stiffness (force/displacement) or (moment/rotation).
KT	Tangent stiffness (force/displacement) or (moment/rotation).
FY	Yield (force) or (moment).

*MAT_SPRING_NONLINEAR_ELASTIC

This 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					

VARIABLE	DESCRIPTION
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		

VARIABLE	DESCRIPTION
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 20.34.
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.

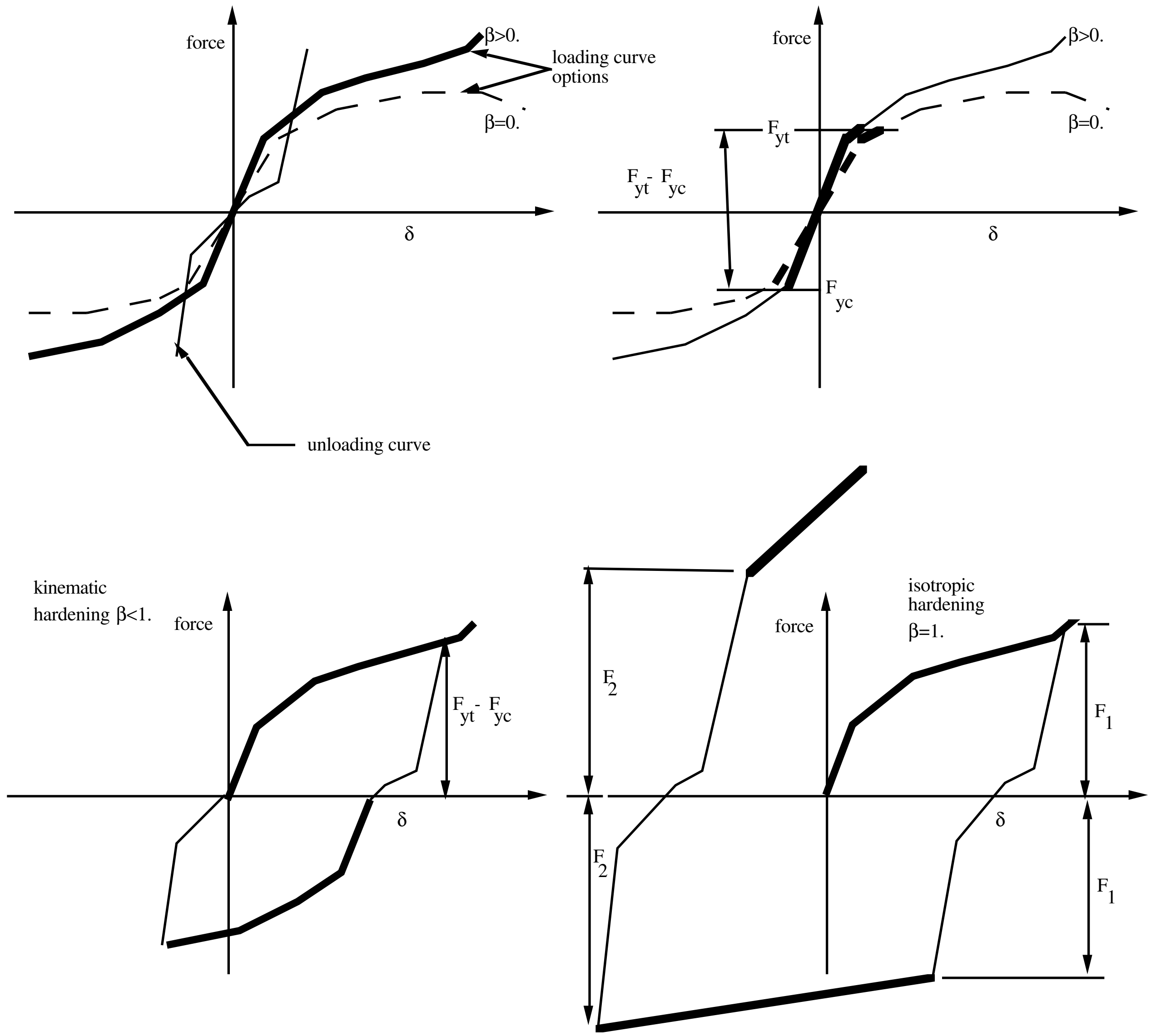


Figure 20.34. General nonlinear material for discrete elements.

*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	

VARIABLE	DESCRIPTION
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

$$K(t) = K_\infty + (K_0 - K_\infty)e^{-\beta t} \quad .$$

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 continous function of time implementation has the disadvantage of the energy absorber's resistance decaying with increasing time even without deformation. The advantage of the incremental implementation is that 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				

VARIABLE	DESCRIPTION
MID	Material identification. A uniques 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

VARIABLE	DESCRIPTION
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

VARIABLE	DESCRIPTION
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							

VARIABLE	DESCRIPTION
MID	Material identification. A uniques number has to be chosen.
L0	Initial muscle length, *Lo.*
VMAX	Maximum CE shortening velocity, *Vmax*.
SV	Scale factor, *Sv,* for *Vmax* vs. active state. LT.0: absolute value gives load curve ID GE.0: constant value of 1.0 is used
A	Activation level vs. time function. LT.0: absolute value gives load curve ID GE.0: constant value of *A* is used
FMAX	Peak isometric force, *Fmax*.

VARIABLE	DESCRIPTION
TL	Active tension vs. length function. LT.0: absolute value gives load curve ID GE.0: constant value of 1.0 is used
TV	Active tension vs. velocity function. LT.0: absolute value gives load curve ID GE.0: constant value of 1.0 is used
FPE	Force vs. length function, *Fpe*, for parallel elastic element. LT.0: absolute value gives load curve ID EQ.0: exponential function is used (see below) GT.0: constant value of 0.0 is used
LMAX	Relative length when *Fpe* reaches *Fmax*. Required if *Fpe*=0 above.
KSH	Constant, *Ksh*, governing the exponential rise of *Fpe*. Required if *Fpe*=0 above.

Remarks:

The material behavior of the muscle model is adapted from the original model proposed by Hill (1938). Reviews of this model and extensions can be found in Winters (1990) and Zajac (1989). The most basic Hill-type muscle model consists of a contractile element (CE) and a parallel elastic element (PE) (Figure 20.35). 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.

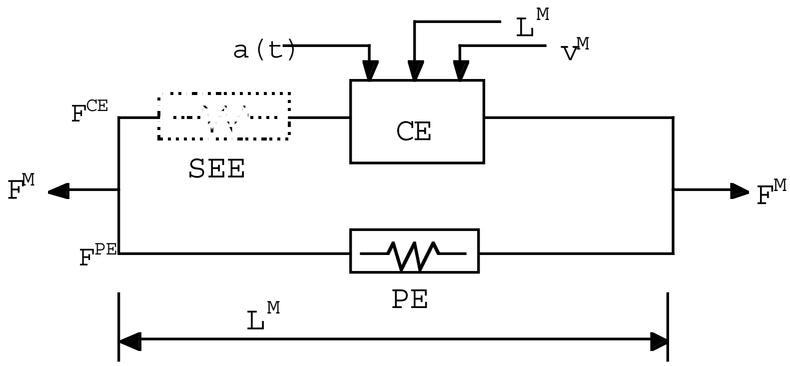


Figure 20.35 Discrete model for muscle contraction dynamics, based on a Hill-type representation. The total force is the sum of passive force F^{PE} and active force F^{CE}. The passive element (PE) represents energy storage from muscle elasticity, while the contractile element (CE) represents force generation by the muscle. The series elastic element (SEE), shown in dashed lines, is often neglected when a series tendon compliance is included. Here, a(t) is the activation level, L^M is the length of the muscle, and v^M is the shortening velocity of the muscle.

When the contractile element (CE) of the Hill model is inactive, the entire resistance to elongation is provided by the PE element and the tendon load-elongation behavior. As activation is increased, force then passes through the CE side of the parallel Hill model, providing the contractile dynamics. The original Hill model accommodated only full activation - this limitation is circumvented in the present implementation by using the modification suggested by Winters (1990). The main features of his approach were to realize that the CE force-velocity input force equals the CE tension-length output force. This yields a three-dimensional curve to describe the force-velocity-length relationship of the CE. If the force-velocity y-intercept scales with activation, then given the activation, length and velocity, the CE force can be determined.

Without the SEE, the total force in the muscle FM is the sum of the force in the CE and the PE because they are in parallel:

$$F^M = F^{PE} + F^{CE}$$

The relationships defining the force generated by the CE and PE as a function of L^M, V^M and a(t) are often scaled by F_{max}, the peak isometric force (p. 80, Winters 1990), L_0, the initial length of the muscle (p. 81, Winters 1990), and V_{max}, the maximum unloaded CE shortening velocity (p. 80, Winters 1990). From these, dimensionless length and velocity can be defined:

$$L = \frac{L^M}{L_o},$$

$$V = \frac{V^M}{V_{max} * S_V(a(t))}$$

Here, S_V scales the maximum CE shortening velocity V_{max} and changes with activation level a(t). This has been suggested by several researchers, i.e. Winters and Stark (1985). The activation level specifies the level of muscle stimulation as a function of time. Both have values between 0 and 1. The functions $S_V(a(t))$ and a(t) are specified via load curves in LS-DYNA, or default values of S_V=1 and a(t)=0 are used. Note that L is always positive and that V is positive for lengthening and negative for shortening.

The relationship between F^{CE}, V and L was proposed by Bahler et al. (1967). A three-dimensional relationship between these quantities is now considered standard for computer implementations of Hill-type muscle models (i.e., eqn 5.16, p. 81, Winters 1990). It can be written in dimensionless form as:

$$F^{CE} = a(t) * F_{max} * f_{TL}(L) * f_{TV}(V)$$

Here, f_{TL} and f_{TV} are the tension-length and tension-velocity functions for active skeletal muscle. Thus, if current values of L^M, V^M, and a(t) are known, then F^{CE} can be determined (Figure 20.35).

The force in the parallel elastic element F^{PE} is determined directly from the current length of the muscle using an exponential relationship (eqn 5.5, p. 73, Winters 1990):

$$f_{PE} = \frac{F^{PE}}{F_{MAX}} = 0, \quad L \le 1$$

$$f_{PE} = \frac{F^{PE}}{F_{MAX}} = \frac{1}{\exp(K_{sh}) - 1}\left[\exp\left(\frac{K_{sh}}{L_{max}}(L-1)\right) - 1\right], \quad L > 1$$

Here, L_{max} is the relative length at which the force F_{max} occurs, and K_{sh} is a dimensionless shape parameter controlling the rate of rise of the exponential. Alternatively, the user can define a custom f_{PE} curve giving tabular values of normalized force versus dimensionless length as a load curve.

For computation of the total force developed in the muscle F^M, the functions for the tension-length f_{TL} and force-velocity f_{TV} relationships used in the Hill element must be defined. These relationships have been available for over 50 years, but have been refined to allow for behavior such as active lengthening. The active tension-length curve f_{TL} describes the fact that isometric muscle force development is a function of length, with the maximum force occurring at an optimal length. According to Winters, this optimal length is typically around L=1.05, and the force drops off for shorter or longer lengths, approaching zero force for L=0.4 and L=1.5. Thus the curve has a bell-shape. Because of the variability in this curve between muscles, the user must specify the function f_{TL} via a load curve, specifying pairs of points representing the normalized force (with values between 0 and 1) and normalized length L (Figure 20.36).

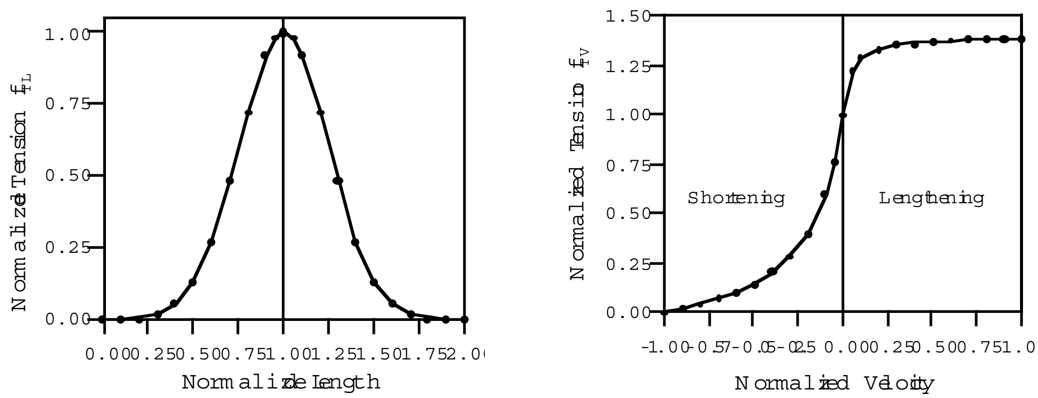


Figure 20.36 Typical normalized tension-length (TL) and tension-velocity (TV) curves for skeletal muscle.

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 Vis 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 Vis less than zero, muscle shortening occurs and the classic Hill equation hyperbola is used to drop the normalized tension to 0 (Figure 20.34). 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 forced 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 sliprings and retractors.

The user inputs a mass per unit length that is used to calculate nodal masses on initialization.

A ‘minimum length’ is also input. This controls the shortest length allowed in any element and determines when an element passes through sliprings or are absorbed into the retractors. Onetenth of a typical initial element length is usually a good choice.

MAT_CFD_*OPTION

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:

CONSTANT

This is material type 150. It allows constant, isotropic fluid properties to be defined for the incompressible/low-Mach CFD solver.

Card Format (1 of 3)

	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 Format (2 of 3)

	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 Format (3 of 3)

	1	2	3	4	5	6	7	8
Variable	DIFF6	DIFF7	DIFF8	DIFF9	DIFF10			
Type	F	F	F	F	F			
Default	-	-	-	-	-			

VARIABLE	DESCRIPTION
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_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						

VARIABLE	DESCRIPTION
TMID	Thermal material identification, a unique number has to be chosen.
TRO	Thermal density: EQ 0.0 default to structural density.
TGRLC	Thermal generation rate curve number, see *DEFINE_CURVE: GT.0: function versus time, EQ.0: use constant multiplier value, TGMULT, LT.0: function versus temperature.
TGMULT	Thermal generation rate multiplier: EQ.0.0: no heat generation.
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					

VARIABLE	DESCRIPTION
TMID	Thermal material identification, a unique number has to be chosen.
TRO	Thermal density: EQ 0.0 default to structural density.
TGRLC	Thermal generation rate curve number, see *DEFINE_CURVE: GT.0: function versus time, EQ.0: use constant multiplier value, TGMULT, LT.0: function versus temperature.
TGMULT	Thermal generation rate multiplier: EQ.0.0: no heat generation.
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
K_1	Thermal conductivity K_1 in local x-direction
K_2	Thermal conductivity K_2 in local y-direction
K_3	Thermal conductivity K_3 in local z-direction
XP, YP, ZP	Define coordinate of point **p** for AOPT = 1
A1, A2, A3	Define components of vector **a** for AOPT = 2
D1, D2, D3	Define components of vector **v** for AOPT = 2

*MAT_THERMAL_ISOTROPIC_TD

This is thermal material property type 3. It allows temperture 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 tempertaure 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

VARIABLE	DESCRIPTION
TMID	Thermal material identification, a unique number has to be chosen.
TRO	Thermal density: EQ 0.0 default to structural density.
TGRLC	Thermal generation rate curve number, see *DEFINE_CURVE: GT.0: function versus time, EQ.0: use constant multiplier value, TGMULT, LT.0: function versus temperature.
TGMULT	Thermal generation rate multiplier: EQ.0.0: no heat generation.
T1 ... T8	Temperatures (T1 ... T8)
C1 ... C8	Heat capacity at T1 ... T8
K1 ... K8	Thermal conductivity at T1 ... T8

*MAT_THERMAL_ORTHOTROPIC_TD

This is thermal material property type 4. It allows temperture 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 tempertaure 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					

VARIABLE	DESCRIPTION
TMID	Thermal material identification, a unique number has to be chosen.
TRO	Thermal density: EQ 0.0 default to structural density.
TGRLC	Thermal generation rate curve number, see *DEFINE_CURVE: GT.0: function versus time, EQ.0: use constant multiplier value, TGMULT, LT.0: function versus temperature.
TGMULT	Thermal generation rate multiplier: EQ.0.0: no heat generation.
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 temperture 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 tempertaure range that the material will see in the analysis.

Card Format (1 of 5)

	1	2	3	4	5	6	7	8
Variable	TMID	TRO	TGRLC	TGMULT				
Type	I	F	F	F				

Card Format (2 of 5)

	1	2	3	4	5	6	7	8
Variable	T1	T2	T3	T4	T5	T6	T7	T8
Type	F	F	F	F	F	F	F	F

Card Format (3 of 5)

	1	2	3	4	5	6	7	8
Variable	C1	C2	C3	C4	C5	C6	C7	C8
Type	F	F	F	F	F	F	F	F

Card Format (4 of 5)

	1	2	3	4	5	6	7	8
Variable	K1	K2	K3	K4	K5	K6	K7	K8
Type	F	F	F	F	F	F	F	F

Card Format (5 of 5)

	1	2	3	4	5	6	7	8
Variable	SOLT	LIQT	LH					
Type	F	F	F					

VARIABLE	DESCRIPTION
TMID	Thermal material identification, a unique number has to be chosen.
TRO	Thermal density: EQ 0.0 default to structural density.
TGRLC	Thermal generation rate curve number, see *DEFINE_CURVE: GT.0: function versus time, EQ.0: use constant multiplier value, TGMULT, LT.0: function versus temperature.
TGMULT	Thermal generation rate multiplier: EQ.0.0: no heat generation.
T1 ... T8	Temperatures (T1 ... T8)
C1 ... C8	Heat capacity at T1 ... T8
K1 ... K8	Thermal conductivity at T1 ... T8
SOLT	Solidus temperature, T_S (must be $< T_L$)
LIQT	Liquidus temperature, T_L (must be $> T_S$)
LH	Latent heat

Remarks:

During phase change, that is between the solidus and liquidus temperatures, the heat capacity of the material will be enhanced to account for the latent heat as follows:

$$c(t) = m\left[1 - \cos 2\pi\left(\frac{T - T_S}{T_L - T_S}\right)\right] \qquad T_S < T < T_L$$

Where

T_L = liquidus termperature

T_S = solidus termperature

T = termperature

m = multiplier such that $\lambda = \int_{T_S}^{T_L} C(T)dT$

λ = latent heat

c = hear capcity

*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 tempertaure 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						

VARIABLE	DESCRIPTION
TMID	Thermal material identification, a unique number has to be chosen.
TRO	Thermal density: EQ 0.0 default to structural density.
TGRLC	Thermal generation rate curve number, see *DEFINE_CURVE: GT.0: function versus time, EQ.0: use constant multiplier value, TGMULT, LT.0: function versus temperature.
TGMULT	Thermal generation rate multiplier: EQ.0.0: no heat generation.
HCLC	Load curve ID specifying heat capacity vs. temperature.
TCLC	Load curve ID specifying thermal conductivity vs. temperature.

REFERENCES

Allman, D.J., "A Compatible Triangular Element Including Vertex Rotations for Plane Elasticity Analysis," Comp. Struct., 19,1-8, (1984).

Arruda, E. and M. Boyce, "A Three-Dimensional Constitutive Model for the Large Stretch Behavior of Rubber Elastic Materials," published in the Journal of the Mechanics and Physics of Solids, Vol. 41, No. 2, pp. 389-412, (1993).

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 Approapriate for Material Acceleration and Overdriven Detonation: Theoretical Background and Fourmulation," Technical Report ARAED-TR-911013, U.S. Army Armament Research, Development and Engineering Center, Picatinney Arsenal, New Jersey, 1991.

Baker, E.L. and J. Orosz, J., "Advanced Warheads Concepts: An Advanced Equation of State for Overdriven Detonation," Technical Report ARAED-TR-911007, U.S. Army Armament Research, Development and Engineering Center, Picatinney Arsenal, New Jersey, 1991.

Baker, E.L. and L.I. Stiel, "Improved Quantitative Explosive Performance Prediction Using Jaguar," 1997 Insensitive Munitions and Energetic Materials Technology Symposium, Tampa, FL, (1997).

Bammann, D.J. and E.C. Aifantis, "A Model for Finite-Deformation Plasticity," Acta Mechanica, 70, 1-13 (1987).

Bammann, D.J. and G. Johnson, "On the Kinematics of Finite-Deformation Plasticity," Acta Mechanica*, 69,* 97-117 (1987).

Bammann, D.J., "Modeling the Temperature and Strain Rate Dependent Large Deformation of Metals," Proceedings of the 11th US National Congress of Applied Mechanics, Tuscon, AZ, (1989).

Bammann, D.J., M.L. Chiesa, A. McDonald, W.A. Kawahara, J.J. Dike, and V.D. Revelli, "Predictions of Ductile Failure in Metal Structures," in AMD-Vol. 107, Failure Criteria and Analysis in Dynamic Response, Edited by. H.E. Lindberg, 7-12, (1990).

Bandak, F.A., private communications, U.S. Dept. of Trans., Division of Biomechanics Research, 400 7th St., S.W. Washington, D.C. 20590 (1991).

REFERENCES

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^o Triangular Plate Element with One-Point Quadrature," Int. J. Num. Meth. Eng., 20, 787-802 (1984).

Belytschko, T.B., L. Schwer, and M.J. Klein, "Large Displacement Transient Analysis of Space Frames," Int. J. Num. Eng., 11, 65-84 (1977).

Benson, D.J. and J.O. Hallquist, "A Simple Rigid Body Algorithm for Structural Dynamics Programs," Int. J. Numer. Meth. Eng., 22, (1986).

Benson, D.J. and J.O. Hallquist, "A Single Surface Contact Algorithm for the Postbuckling Analysis of Shell Structures," Comp. Meths. Appl. Mech. Eng., 78, 141-163 (1990).

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).

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).

REFERENCES

Couch, R., E. Albright, and N. Alexander, “The Joy Computer Code,” Lawrence Livermore National Laboratory, Internal Document Rept. UCID-19688, (January, 1983).

CRAY-1 Computer System CFT Reference Manual, Cray Research Incorporated, Bloomington, NM., Publication No. 2240009 (1978).

DeRuntz, J.A. Jr., “Reference Material for USA, The Underwater Shock Analysis Code, USA-STAGS, and USA-STAGS-CFA,” Report LMSC-P032568, Computational Mechanics Laboratory, Lockheed Palo Alto Research Laboratory, Palo Alto, CA. (1993).

Desai, C.S., and H.J. Siriwardane, Constitutive Laws for Engineering Materials With Emphasis On Geologic Materials, Prentice-Hall, Chapter 10, (1984).

Dick, R.E., and W.H. Harris, "Full Automated Rezoning of Evolving Geometry Problems," Numerical Methods in Industrial Forming Processes, Chenot, Wood, and Zienkiewicz, Editors, Bulkema, Rotterdam, 243-248, (1992)

Dilger, W.H., R. Koch, and R. Kowalczyk, "Ductility of Plain and Confined Concrete Under Different Strain Rates," ACI Journal, January-February, (1984).

Dobratz, B.M., “LLNL Explosives Handbook, Properties of Chemical Explosives and Explosive Simulants,” University of California, Lawrence Livermore National Laboratory, Rept. UCRL-52997 (1981).

Englemann, B. E., R.G. Whirley, and G.L. Goudreau, “A Simple Shell Element Formulation for Large-Scale Elastoplastic Analysis,” CED-Vol. 3. Analytical and Computational Models of Shells, A.K. Noor, T. Belytschko, and J.C. Simo, Editors, 1989, pp. 399-416.

Faßnacht, W., "Simulation der Rißbildung in Aluminiumgußbauteilen," Dissertation, Technishe Universität Darmstadt, (1999).

Feucht, M., "Ein gradientenabhängiges Gursonmodell zur Beshreibung duktiler Schädigung mit Entfestigung," Dissertation, Technishe 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)

Ginsberg, M. and J. Johnson, “Benchmarking the Performance of Physical Impact Simulation Software on Vector and Parallel Computers,” Applications Track of Supercomputing, IEEE monograph, Computer Society Press, March, 1989.

Giroux, E.D. “HEMP User’s Manual,” University of California, Lawrence Livermore National Laboratory, Rept. UCRL-51079 (1973).

Goudreau, G.L. and J.O. Hallquist, “Recent Developments in Large Scale Finite Element Lagrangian Hydrocode Technology,” J. Comp. Meths. Appl. Mechs. Eng., 30 (1982).

Goldak, J., Chakravarti, A., and Bibby, M., “A New Finite Element Model for Welding Heat Sources,” Metallurgical Transactions B, vol. 15B, pp. 299-305, June, 1984

Govindjee, S., Kay, J.G., and Simo, J.C. [1994], "Anisotropic Modeling and Numerical Simulation of Brittle Damage in Concrete," Report No. UCB/SEMM-94/18, Department of Civil Engineering, University of California, Berkeley, CA 94720.

Govindjee, S., Kay, J.G., and Simo, J.C. [1995], "Anisotropic Modeling and Numerical Simulation of Brittle Damage in Concrete," *Int. J. Numer. Meth. Engng*, **38**, 3611-3633.

Graefe, H., W. Krummheuer, and V. Siejak, “Computer Simulation of Static Deployment Tests for Airbags, Air Permeability of Uncoated Fabrics and Steady State Measurements of the Rate of Volume Flow Through Airbags,” SAE Technical Paper Series, 901750, Passenger Car Meeting and Expositition, Dearborn, Michigan, September 17-20, 1990.

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**]

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).

REFERENCES

Hallquist, J.O., “Theoretical Manual for DYNA3D,” University of California, Lawrence Livermore National Laboratory, Rept. UCID-19501 (March, 1983).

Hallquist, J.O., “DYNA3D User's Manual (Nonlinear Dynamic Analysis of Solids in Three Dimensions),” University of California, Lawrence Livermore National Laboratory, Rept. UCID-19156 (1988, Rev. 4).

Hallquist, J.O., “LS-DYNA User's Manual (Nonlinear Dynamic Analysis of Solids in Three Dimensions),” Livermore Software Technology Corporation, Rept. 1007 (1990).

Hallquist, J.O., D.J. Benson, and G.L. Goudreau, “Implementation of a Modified Hughes-Liu Shell into a Fully Vectorized Explicit Finite Element Code,” Proceedings of the International Symposium on Finite Element Methods for Nonlinear Problems, University of Trondheim, Trondheim, Norway (1985).

Hallquist, J.O. and D.J. Benson, “A Comparison of an Implicit and Explicit Implementation of the Hughes-Liu Shell,” Finite Element Methods for Plate and Shell Structures, T.J.R. Hughes and E. Hinton, Editors, 394-431, Pineridge Press Int., Swanea, U.K. (1986).

Hallquist, J.O. and D.J. Benson, “DYNA3D User’s Manual (Nonlinear Dynamic Analysis of Solids in Three Dimensions),” University of California, Lawrence Livermore National Laboratory, Rept. UCID-19156 (Rev. 2: 1986; Rev. 3: 1987).

Hallquist, J.O., D.W. Stillman, T.J.R. Hughes, C. and Tarver,”Modeling of Airbags Using MVMA/DYNA3D,” LSTC Report (1990).

Herrmann, L.R. and F.E. Peterson, “A Numerical Procedure for Viscoelastic Stress Analysis,” Seventh Meeting of ICRPG Mechanical Behavior Working Group, Orlando, FL, CPIA Publication No. 177, 1968.

Hill A.V., "The heat of shortening and the dynamic constants of muscle," Proc Roy Soc B126:136-195, (1938).

Hill, R., “A Theory of the Yielding and Plastic Flow of Anisotropic Metals,” Proceedings of the Royal Society of London, Series A., Vol. 193, 1948, pp. 281-197.

Hill, R., “Constitutive Modelling of Orthotropic Plasticity in Sheet Metals,” J. Mech. Phys. Solids, Vol. 38, No. 3, 1989, pp. 405-417.

Hirth, A., P. Du Bois, and K. Weimar, "Improvement of LS-DYNA Material Law 83 (Fu Chang) for the Industrial Simulation of Reversible Energy-Absorbing Foams," CAD-FEM User's Meeting, Bad Neuenahr - Ahrweiler, Germany, October 7-9, Paper 2-40, (1998).

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).

Kenchington, G.J., “A Non-Linear Elastic Material Model for DYNA3D,” Proceedings of the DYNA3D Users Group Conference, September 1988, published by Boeing Computer Services (Europe) Limited.

Key, S.W. “HONDO – A Finite Element Computer Program for the Large Deformation Dynamic Response of Axisymmetric Solids,” Sandia National Laboratories, Albuquerque, N.M., Rept. 74-0039 (1974).

Krieg, R.D. and S.W. Key, “Implementation of a Time Dependent Plasticity Theory into Structural Computer Programs,” Vol. 20 of Constitutive Equations in Viscoplasticity: Computational and Engineering Aspects (American Society of Mechanical Engineers, New York, N.Y., 1976), pp. 125-137.

Lee, E.L. and C.M. Tarver, “Phenomenological Model of Shock Initiation in Heterogenous Explosives,” PHYS. Fluids, Vol. 23, p. 2362 (1980).

Lemaitre, J., A Course on Damage Mechanics, Springer-Verlag, (1992).

MADYMO3D USER’S MANUAL, Version 4.3, TNO Road-Vehicles Research Institute, Department of Injury Prevention, The Hague, The Netherlands, (1990).

Maker, B.N., Private communication Lawrence Livermore National Laboratory, Dr. Maker programmed and implemented the compressible Mooney Rivlin rubber model (1987).

Matzenmiller, A. and J. K. Schm, "Crashworthiness Considerations of Composite Structures – A First Step with Explicit Time Integration in Nonlinear Computational Mechanics–State-of-the-Art," Ed. P. Wriggers, W. Wagner, Springer Verlay, (1991).

Neilsen, M.K., H.S. Morgan, and R.D. Krieg, "A Phenomenological Constitutive Model for Low Density Polyurethane Foams," Rept. SAND86-2927, Sandia National Laboratories, Albuquerque, N.M., (1987)

Nusholtz, G., W. Fong, and J. Wu, "Air Bag Wind Blast Phenomena Evaluation," Experimental Techniques, Nov.-Dec. (1991).

Nusholtz, G., D. Wang, and E.B. Wylie, "Air Bag Momentum Force Including Aspiration," Preprint, Chrysler Corporation, (1996).

Nusholz, private communication, (1996).

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)

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).

Sackett, S.J., "Geological/Concrete Model Development," Private Communication (1987).

Sandler, I.S. and D. Rubin, "An Algorithm and a Modular Subroutine for the Cap Model," Int. J. Numer. Analy. Meth. Geomech., 3, pp. 173-186 (1979).

Schwer, L.E., W. Cheva, and J.O. Hallquist, "A Simple Viscoelastic Model for Energy Absorbers Used in Vehicle-Barrier Impacts," in preparation.

Simo, J.C., J.W. Ju, K.S. Pister, and R.L. Taylor, "An Assessment of the Cap Model: Consistent Return Algorithms and Rate-Dependent Extension," J. Eng. Mech., Vol. 114, No. 2, 191-218 (1988a).

Simo, J.C., J.W. Ju, K.S. Pister, and R.L. Taylor, "Softening Response, Completeness Condition, and Numerical Algorithms for the Cap Model," Int. J. Numer. Analy. Meth. Eng., (in press) (1988b).

Steinberg, D.J. and M.W. Guinan, "A High-Strain-Rate Constitutive Model for Metals," University of California, Lawrence Livermore National Laboratory, Rept. UCRL-80465 (1978).

Steinberg, D.J. and C.M. Lund, "A Constitutive Model for Strain Rates form 10^{-4} to 10^{6} S^{-1}," J. Appl. Phys., 65, p. 1528 (1989).

Stillman, D.W. and J.O. Hallquist, "INGRID: A Three-Dimensional Mesh Generator for Modeling Nonlinear Systems," University of California, Lawrence Livermore National Laboratory, Rept. UCID-20506. (1985).

Storakers, B., "On Material Representation and Constitutive Branching in Finite Compressible Elasticity", Royal Institute of Technology, Stockholm, Sweden, (1985).

Stout, M.G., D.E. Helling, T.L. Martin, and G.R. Canova, Int. J. Plasticity, Vol. 1, pp. 163-174, 1985.

Taylor, L.M. and D.P. Flanagan, "PRONTO3D A Three-Dimensional Transient Solid Dynamics Program," Sandia Report: SAND87-1912, UC-32, (1989).

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).

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)

Wilson, E.L. *Three Dimensional Static and Dynamic Analysis of Structures,* Computers and Structures, Inc., Berkeley CA, (2000).

REFERENCES

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).

Zajac F.E., "Muscle and tendon: Properties, models, scaling, and application to biomechanics and motor control, "CRC Critical Reviews in Biomedical Engineering 17(4):359-411, (1989).

APPENDIX A: User Defined Materials

The addition of user material subroutine into LS-DYNA is relatively simple. A control card, starting with card 14 in the control section, is required for each user subroutine. The number of history variables is arbitrary and can be any number greater than or equal to 0. When the material requires the deformation gradient, the number of history variables must be increased by 9 for its storage. The coordinate system definition is optional but is probably necessary if the model involves material that have directional properties such as composites and anisotropic plasticity models. When the coordinate system option is used then all data passed to the constitutive model is in the local system. A bulk modulus and shear modulus are required for transmitting boundaries, contact interfaces, rigid body constraints, and time step size calculations. The number of constants read in columns 6-10 include the eight values for the coordinate system option if it is nonzero and two values for the bulk and shear modulus.Up to ten user subroutines can currently be implemented simultaneously to update the stresses is solids, shells, thick shells, and beam elements. A sample subroutine is given in this Appendix for treating an elastic material.

The deformation gradient matrix is stored in 9 of the history variables requested on the control cards. To compute the deformation gradient matrix for 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        .
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
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
       CHARACTER*(*) ETYPE
       DIMENSION CM(*),EPS(*),SIG(*),HISV(*)
C
C      COMPUTE SHEAR MODULUS, G
C
       G2=CM(1)/(1.+CM(2))
       G =.5*G
C
       IF (ETYPE.EQ.'BRICK') THEN
       DAVG=(-EPS(1)-EPS(2)-EPS(3))/3.
       P=-DAVG*CM(1)/((1.-2.*CM(2)))
       SIG(1)=SIG(1)+P+G2*(EPS(1)+DAVG)
       SIG(2)=SIG(2)+P+G2*(EPS(2)+DAVG)
       SIG(3)=SIG(3)+P+G2*(EPS(3)+DAVG)
       SIG(4)=SIG(4)+G*EPS(4)
       SIG(5)=SIG(5)+G*EPS(5)
       SIG(6)=SIG(6)+G*EPS(6)
C
       ELSEIF (ETYPE.EQ.'SHELL') THEN
C
       GC =CAPA*G
       Q1     =CM(1)*CM(2)/((1.0+CM(2))*(1.0-2.0*CM(2)))
       Q3     =1./(Q1+G2)
       EPS(3)=-Q1*(EPS(1)+EPS(2))*Q3
       DAVG   =(-EPS(1)-EPS(2)-EPS(3))/3.
       P      =-DAVG*CM(1)/((1.-2.*CM(2)))
       SIG(1)=SIG(1)+P+G2*(EPS(1)+DAVG)
       SIG(2)=SIG(2)+P+G2*(EPS(2)+DAVG)
       SIG(3)=0.0
       SIG(4)=SIG(4)+G *EPS(4)
       SIG(5)=SIG(5)+GC*EPS(5)
       SIG(6)=SIG(6)+GC*EPS(6)
C
       ELSEIF (ETYPE.EQ.'BEAM') THEN
       Q1     =CM(1)*CM(2)/((1.0+CM(2))*(1.0-2.0*CM(2)))
       Q3     =Q1+2.0*G
       GC     =CAPA*G
       DETI   =1./(Q3*Q3-Q1*Q1)
       C22I   = Q3*DETI
       C23I   =-Q1*DETI
       FAC    =(C22I+C23I)*Q1
       EPS(2)=-EPS(1)*FAC-SIG(2)*C22I-SIG(3)*C23I
       EPS(3)=-EPS(1)*FAC-SIG(2)*C23I-SIG(3)*C22I
       DAVG   =(-EPS(1)-EPS(2)-EPS(3))/3.
       P      =-DAVG*CM(1)/(1.-2.*CM(2))
       SIG(1)=SIG(1)+P+G2*(EPS(1)+DAVG)
       SIG(2)=0.0
       SIG(3)=0.0
       SIG(4)=SIG(4)+GC*EPS(4)
       SIG(5)=0.0
       SIG(6)=SIG(6)+GC*EPS(6)
       ENDIF
C
       RETURN
       END
```

APPENDIX B: User Defined Airbag Sensor

The addition of a user sensor subroutine into LS-DYNA is relatively simple. The sensor is mounted on a rigid body which is attached to the structure. The motion of the sensor is provided in the local coordinate system defined for the rigid body in the definition of material model 20–the rigid material. When the user defined criterion is met for the deployment of the airbag, a flag is set and the deployment begins. All load curves relating to the mass flow rate versus time are then shifted by the initiation time. The user subroutine is given below with all the necessary information contained in the comment cards.

```
      SUBROUTINE AIRUSR (RBU,RBV,RBA,TIME,DT1,DT2,PARAM,HIST,ITRNON,
      . RBUG,RBVG,RBAG)
C*******************************************************************
C|     LIVERMORE SOFTWARE TECHNOLOGY CORPORATION (LSTC)            |
C|     ------------------------------------------------------------|
C|     COPYRIGHT 1987, 1988, 1989 JOHN O. HALLQUIST, LSTC          |
C|     ALL RIGHTS RESERVED                                         |
C*******************************************************************
C
C     USER SUBROUTINE TO INITIATE THE INFLATION OF THE AIRBAG
C
C     VARIABLES
C
C     DISPLACEMENTS ARE DEFINED AT TIME N+1 IN LOCAL SYSTEM
C     VELOCITIES ARE DEFINED AT TIME N+1/2 IN LOCAL SYSTEM
C     ACCELERATIONS ARE DEFINED AT TIME N IN LOCAL SYSTEM
C
C           RBU(1-3) TOTAL DISPLACEMENTS IN THE LOCAL XYZ DIRECTIONS
C           RBU(3-6) TOTAL ROTATIONS ABOUT THE LOCAL XYZ AXES
C           RBV(1-3) VELOCITIES IN THE LOCAL XYZ DIRECTIONS
C           RBV(3-6) ROTATIONAL VELOCITIES ABOUT THE LOCAL XYZ AXES
C           RBA(1-3) ACCELERATIONS IN THE LOCAL XYZ DIRECTIONS
C           RBA(3-6) ROTATIONAL ACCELERATIONS ABOUT THE LOCAL XYZ AXES
C           TIME IS THE CURRENT TIME
C           DT1 IS TIME STEP SIZE AT N-1/2
C           DT2 IS TIME STEP SIZE AT N+1/2
C           PARAM IS USER DEFINED INPUT PARAMETERS (MAX 25)
C           HIST IS USER DEFINED HISTORY VARIABLES (MAX 25)
C           ITRNON IS FLAG TO TURN ON THE AIRBAG INFLATION
C           RBUG,RBVG,RBAG, ARE SIMILAR TO RBU,RBV,RBA BUT ARE DEFINED
C           GLOBALLY.
C
C     THE USER SUBROUTINE SETS THE VARIABLE ITRNON TO:
C
C              ITRNON=0 BAG IS NOT INFLATED
C              ITRNON=1 BAG INFLATION BEGINS AND THIS SUBROUTINE IN NOT
C                       CALLED AGAIN
C
      DIMENSION RBU(6),RBV(6),PARAM(25),HIST(25),
      . RBUG(6),RBVG(6),RBAG(6)
      RETURN
      END
```

APPENDIX C: User Defined Solution Control

This subroutine may be provided by the user to control the I/O, monitor the energies and other solution norms of interest, and to shut down the problem whenever he pleases. The arguments are defined in the listing provided below. This subroutine is call each time step and does not need any control card to operate.

```
      SUBROUTINE UCTRL1 (NUMNP,NDOF,TIME,DT1,DT2,PRTC,PLTC,FRCI,PRTO,
     . PLTO,FRCO,VT,VR,AT,AR,UT,UR,XMST,XMSR,IRBODY,RBDYN,USRHV,
     . MESSAG,TOTALM,CYCL,IDRINT)
C*******************************************************************
C|     LIVERMORE SOFTWARE TECHNOLOGY CORPORATION (LSTC)            |
C|     ------------------------------------------------------------ |
C|     COPYRIGHT 1987, 1988, 1989 JOHN O. HALLQUIST, LSTC          |
C|     ALL RIGHTS RESERVED                                         |
C*******************************************************************
C
      CHARACTER*(*) MESSAG
      INTEGER CYCLE
C
C
C     USER SUBROUTINE FOR SOLUTION CONTROL
C
C     NOTE:  LS-DYNA USED AN INTERNAL NUMBERING SYSTEM TO
C            ACCOMODATE ARBITRARY NODE NUMBERING.  TO ACCESS
C            INFORMATION FOR USER NODE N, ADDRESS ARRAY LOCATION M,
C            M=LQF(N,1).  TO OBTAIN USER NODE NUMBER, N,
C            CORRESPONDING TO ARRAY ADDRESS M, SET N=LQFINV(M,1)
C
C     ARGUMENTS:
C        NUMNP=NUMBER OF NODAL POINTS
C        NDOF=NUMBER OF DEGREES IF FREEDOM PER NODE
C        TIME=CURRENT SOLUTION TIME
C        PRTC=OUTPUT INTERVAL FOR TAURUS TIME HISTORY DATA
C        PLTC=OUTPUT INTERVAL FOR TAURUS STATE DATA
C        FRCI=OUTPUT INTERVAL FOR TAURUS INTERFACE FORCE DATA
C        PRTO=OUTPUT TIME FOR TIME HISTORY FILE
C        PLTO=OUTPUT TIME FOR STATE DATA
C        FRCO=OUTPUT TIME FOR FORCE DATA
C        VT(3,NUMNP)  =NODAL TRANSLATIONAL VELOCITY VECTOR
C        VR(3,NUMNP)  =NODAL ROTATIONAL VELOCITY VECTOR.  THIS ARRAY
C                      IS DEFINED IF AND ONLY IF NDOF=6
C        AT(3,NUMNP)  =NODAL TRANSLATIONAL ACCELERATION VECTOR
C        AR(3,NUMNP)  =NODAL ROTATIONAL ACCELERATION VECTOR.  THIS
C                      ARRAY IS DEFINED IF AND ONLY IF NDOF=6
C        UT(3,NUMNP)  =NODAL TRANSLATIONAL DISPLACEMENT VECTOR
C        UR(3,NUMNP)  =NODAL ROTATIONAL DISPLACEMENT VECTOR.  THIS ARRAY
C                      IS DEFINED IF AND ONLY IF NDOF=6
C        XMST(NUMNP)  =RECIPROCAL OF NODAL TRANSLATIONAL MASSES
C        XMSR(NUMNP)  =RECIPROCAL OF NODAL ROTATIONAL MASSES.  THIS
C                      ARRAY IS DEFINED IF AND ONLY IF NDOF=6
C        IRBODY       =FLAG FOR RIGID BODY NODAL POINTS
C                      IF DEFORMABLE NODE THEN SET TO 1.0
C                      IF RIGID BODY NODE THEN SET TO 0.0
C                      DEFINED IF AN ONLY IF RIGID BODY ARE PRESENT
C                      I.E.,IRBODY.NE.0 IF NO RIGID BODY ARE PRESENT
C        USRHV(LENHV)  =USER DEFINED HISTORY VARIABLES THAT ARE STORED
```

```
C                       IN THE RESTART FILE.  LENHV=100+U*NUMMAT WHERE
C                       NUMMAT IS THE # OF MATERIALS IN THE PROBLEM.
C                       ARRAY USRHV IS UPDATED ONLY IN THIS SUBROUTINE.
C        MESSAG        =FLAG FOR DYNA3D WHICH MAY BE SET TO:
C                       'SW1.' LS-DYNA TERMINATES WITH RESTART FILE
C                       'SW3.' LS-DYNA WRITES A RESTART FILE
C                       'SW4.' LS-DYNA WRITES A PLOT STATE
C        TOTALM        =TOTAL MASS IN PROBLEM
C           CYCLE =CYCLE NUMBER
C           IDRINT      =FLAG FOR DYNAMIC RELAXATION PHASE
C                       .NE.0:  DYNAMIC RELAXATION IN PROGRESS
C                       .EQ.0:  SOLUTION PHASE
C
      COMMON/PTIMES/  PRTIMS(32),PRTLST(32),IGMPRT
C
C     PRTIMS(32)=OUTPUT INTERVALS FOR ASCII FILES
C
C     ASCII FILES
C           ( 1)=CROSS SECTION FORCES
C           ( 2)=RIGID WALL FORCES
C           ( 3)=NODAL DATA
C           ( 4)=ELEMENT DATA
C           ( 5)=GLOBAL DATA
C           ( 6)=DISCRETE ELEMENTS
C           ( 7)=MATERIAL ENERGIES
C           ( 8)=NODAL INTERFACE FORCES
C           ( 9)=RESULTANT INTERFACE FORCES
C           (10)=SMUG ANIMATOR
C           (11)=SPC REACTION FORCES
C           (12)=NODAL CONSTRAIN RESULTANT FORCES
C           (13)=AIRBAG STATISTICS
C           (14)=AVS DATABASE
C           (15)=NODAL FORCE GROUPS
C           (16)=OUTPUT INTERVALS FOR NODAL BOUNDARY CONDITIONS
C           (17)-(32)=UNUSED AT THIS TIME
C
C     PRTLST(32)=OUTPUT TIMES FOR ASCII FILES ABOVE.  WHEN SOLUTION TIME
C             EXCEEDS THE OUTPUT TIME A PRINT STATE IS DUMPED.
C
      COMMON/RBKENG/ENRBDY,RBDYX,RBDYY,RBDYZ
C
C     TOTAL RIGID BODY ENERGIES AND MOMENTUMS:
C             ENRBDY=RIGID BODY KINETIC ENERGY
C             RBDYX =RIGID BODY X-MOMENTUM
C             RBDYY =RIGID BODY Y-MOMENTUM
C             RBDYZ =RIGID BODY Z-MOMENTUM
C
      COMMON/RBKENG/ENRBDY,RBDYX,RBDYY,RBDYZ
C
C     TOTAL RIGID BODY ENERGIES AND MOMENTUMS:
C             SWXMOM=STONEWALL X-MOMENTUM
C             SWYMOM=STONEWALL Y-MOMENTUM
C             SWZMOM=STONEWALL Z-MOMENTUM
C             ENRBDY=STONEWALL KINETIC ENERGY
C
      COMMON/DEENGS/DEENG
C
C     DEENG=TOTAL DISCRETE ELEMENT ENERGY
```

```
C
      COMMON/ENERGY/XPE
C
C     XPE   =TOTAL INTERNAL ENERGY IN THE FINITE ELEMENTS
C
      DIMENSION VT(3,*),VR(3,*),AT(3,*),AR(3,*),UT(3,*),UR(3,*)
      XMST(*),XMSR(*),RBDYN(*),USRHV(*)
C
C     SAMPLE MOMENTUM AND KINETIC ENERGY CALCULATIONS
C
C     REMOVE ALL COMMENTS IN COLUMN 1 BELOW TO ACTIVATE
CC
CC
CC    INITIALIZE KINETIC ENERGY, XKE, AND X,Y,Z MOMENTUMS.
CC
C     XKE=2.*SWKENG+2.*ENRBDY
C     XM-SWXMOM+RBDYX
C     YM=SWYMOM+RBDYY
C     ZM=SWZMOM+RBDYZ
CC
C     NUMNP2=NUMNP
C     IF (NDOF.EQ.6) THEN
C     NUMNP2=NUMNP+NUMNP
C     ENDIF
C     PRINT *,NDOF
C     IF(IRBODY.EQ.0) THEN
CC
CC
CC    NO RIGID BODIES PRESENT
CC
CC    NOTE IN BLANK COMMENT VR FOLLOWS VT.  THIS FACT IS USED BELOW.
C     DO 10 N=1,NUMNP2
C     XMSN=1./XMST(N)
C     VN1=VT(1,N)
C     VN2=VT(2,N)
C     VN3=VT(3,N)
C     XM=XM+XMSN*VN1
C     YM=YM+XMSN*VN2
C     ZM=ZM+XMSN*VN3
C     XKE=XKE+XMSN*(VN1*VN1+VN2*VN2+VN3*VN3)
C  10 CONTINUE
CC
C     ELSE
CC
CC    RIGID BODIES PRESENT
CC
C     DO 20 N=1,NUMNP
C     XMSN=1./XMST(N)
C     VN1=RBDYN(N)*VT(1,N)
C     VN2=RBDYN(N)*VT(2,N)
C     VN3=RBDYN(N)*VT(3,N)
C     XM=XM+XMSN*VN1
C     YM=YM+XMSN*VN2
C     ZM=ZM+XMSN*VN3
C     XKE=XKE+XMSN*(VN1*VN1+VN2*VN2+VN3*VN3)
C  20 CONTINUE
C     IF (NDOF.EQ.6) THEN
C     DO 30 N=1,NUMNP
```

```
C      XMSN=1./XMSR(N)
C      VN1=RBDYN(N)*VR(1,N)
C      VN2=RBDYN(N)*VR(2,N)
C      VN3=RBDYN(N)*VR(3,N)
C      XM=XM+XMSN*VN1
C      YM=YM+XMSN*VN2
C      ZM=ZM+XMSN*VN3
C      XKE=XKE+XMSN*(VN1*VN1+VN2*VN2+VN3*VN3)
C   30 CONTINUE
C      ENDIF
CC
C      ENDIF
       RETURN
       END
CC
CC.....TOTAL KINETIC ENERGY
C       XKE=.5*XKE
CC.....TOTAL INTERNAL ENERGY
C       XIE=.XPE+DEENG
CC.....TOTAL ENERGY
C       XTE=XKE+XPE+DEENG
CC.....TOTAL X-RIGID BODY VELOCITY
C       XRBV=XM/TOTALM
CC.....TOTAL Y-RIGID BODY VELOCITY
C       YRBV=YM/TOTALM
CC.....TOTAL Z-RIGID BODY VELOCITY
C       ZRBV=ZM/TOTALM
C
       RETURN
       END
```

APPENDIX D: User Defined Interface Control

This subroutine may be provided by the user to turn the interfaces on and off. This option is activated by the *USER_INTERFACE_CONTROL keyword. The arguments are defined in the listing provided below.

```
      SUBROUTINE UCTRL2 (NSI,NTY,TIME,CYCLE,MSR,NMN,NSV,NSN,
     1     THMR,THSV,VT,XI,UT,ISKIP,IDRINT,NUMNP,DT2,NINPUT,UA)
C*******************************************************************
C|     LIVERMORE SOFTWARE TECHNOLOGY CORPORATION (LSTC)            |
C|     ------------------------------------------------------------|
C|     COPYRIGHT 1987, 1988, 1989 JOHN O. HALLQUIST, LSTC          |
C|     ALL RIGHTS RESERVED                                         |
C*******************************************************************
C
      INTEGER CYCLE
C
C
C     USER SUBROUTINE FOR INTERFACE CONTROL
C
C     NOTE:  LS-DYNA USED AN INTERNAL NUMBERING SYSTEM TO
C            ACCOMODATE ARBITRARY NODE NUMBERING.  TO ACCESS
C            INFORMATION FOR USER NODE N, ADDRESS ARRAY LOCATION M,
C            M=LQF(N,1).  TO OBTAIN USER NODE NUMBER, N,
C            CORRESPONDING TO ARRAY ADDRESS M, SET N=LQFINV(M,1)
C
C     ARGUMENTS:
C        NSI          =NUMBER OF SLIDING INTERFACE
C        NTY          =INTERFACE TYPE.
C                     .EQ.4:SINGLE SURFACE
C                     .NE.4:SURFACE TO SURFACE
C        TIME         =CURRENT SOLUTION TIME
C        CYCLE        =CYCLE NUMBER
C        MSR(NMN)     =LIST OF MASTER NODES NUMBERS IN INTERNAL
C                      NUMBERING SCHEME
C        NMN          =NUMBER OF MASTER NODES
C        NSV(NSN)     =LIST OF SLAVE NODES NUMBERS IN INTERNAL
C                      NUMBERING SCHEME
C        NSN          =NUMBER OF SLAVE NODES
C        THMR(NMN)    =MASTER NODE THICKNESS
C        THSV(NSN)    =SLAVE NODE THICKNESS
C        VT(3,NUMNP)  =NODAL TRANSLATIONAL VELOCITY VECTOR
C        XI(3,NUMNP)  =INITIAL COORDINATES AT TIME=0
C        UT(3,NUMNP)  =NODAL TRANSLATIONAL DISPLACEMENT VECTOR
C        IDRINT       =FLAG FOR DYNAMIC RELAXATION PHASE
C                     .NE.0:DYNAMIC RELAXATION IN PROGRESS
C                     .EQ.0:SOLUTION PHASE
C        NUMNP        =NUMBER OF NODAL POINTS
C        DT2          =TIME STEP SIZE AT N+1/2
C        NINPUT       =NUMBER OF VARIABLES INPUT INTO UA
C        UA(*)        =USER'S ARRAY, FIRST NINPUT LOCATIONS
C                      DEFINED BY USER.  THE LENGTH OF THIS
C                      ARRAY IS DEFINED ON CONTROL CARD 10.
C                      THIS ARRAY IS UNIQUE TO INTERFACE NSI.
C
C     SET FLAG FOR ACTIVE CONTACT
C       ISKIP=0 ACTIVE
```

```
C       ISKIP=1 INACTIVE
C
C*******************************************************************
      DIMENSION MSR(*),NSV(*),THMR(*),THSV(*),VT(3,*),XI(3,*),
                UT(3,*)UA(*)
C
C     THE FOLLOWING SAMPLE OF CODEING IS PROVIDED TO ILLUSTRATE HOW
C     THIS SUBROUTINE MIGHT BE USED.  HERE WE CHECK TO SEE IF THE
C     SURFACES IN THE SURFACE TO SURFACE CONTACT ARE SEPARATED.  IF
C     SO THE ISKIP=1 AND THE CONTACT TREATMENT IS SKIPPED.
C
      IF (NTY.EQ.4) RETURN
      DT2HLF=DT2/2.
      XMINS= 1.E20
      XMAXS=-XMINS
      YMINS= 1.E20
      YMAXS=-YMINS
      ZMINS= 1.E20
      ZMAXS=-ZMINS
      XMINM= 1.E20
      XMAXM=-XMINM
      YMINM= 1.E20
      YMAXM=-YMINM
      ZMINM= 1.E20
      ZMAXM=-ZMINM
      THKS=0.0
      THKM=0.0
      DO 10 I=1,NSN
      DSP1=UT(1,NSV(I))+DT2HLF*VT(1,NSV(I))
      DSP2=UT(2,NSV(I))+DT2HLF*VT(2,NSV(I))
      DSP3=UT(3,NSV(I))+DT2HLF*VT(3,NSV(I))
      X1=XI(1,NSV(I))+DSP1
      X2=XI(2,NSV(I))+DSP2
      X3=XI(3,NSV(I))+DSP3
      THKS =MAX(THSV(I),THKS)
      XMINS=MIN(XMINS,X1)
      XMAXS=MAX(XMAXS,X1)
      YMINS=MIN(YMINS,X2)
      YMAXS=MAX(YMAXS,X2)
      ZMINS=MIN(ZMINS,X3)
      ZMAXS=MAX(ZMAXS,X3)
   10 CONTINUE
      DO 20 I=1,NMN
      DSP1=UT(1,MSR(I))+DT2HLF*VT(1,MSR(I))
      DSP2=UT(2,MSR(I))+DT2HLF*VT(2,MSR(I))
      DSP3=UT(3,MSR(I))+DT2HLF*VT(3,MSR(I))
      X1=XI(1,MSR(I))+DSP1
      X2=XI(2,MSR(I))+DSP2
      X3=XI(3,MSR(I))+DSP3
      THKM =MAX(THMR(I),THKS)
      XMINS=MIN(XMINM,X1)
      XMAXS=MAX(XMAXM,X1)
      YMINS=MIN(YMINM,X2)
      YMAXS=MAX(YMAXM,X2)
      ZMINS=MIN(ZMINM,X3)
      ZMAXS=MAX(ZMAXM,X3)
   20 CONTINUE
      IF (XMAXS+THKS.LT.XMINM-THKM) GO TO 40
```

```
      IF (YMAXS+THKS.LT.YMINM-THKM) GO TO 40
      IF (ZMAXS+THKS.LT.ZMINM-THKM) GO TO 40
      IF (XMAXS+THKM.LT.XMINS-THKS) GO TO 40
      IF (YMAXS+THKM.LT.YMINS-THKS) GO TO 40
      IF (ZMAXS+THKM.LT.ZMINS-THKS) GO TO 40
      ISKIP=0
      RETURN
   40 ISKIP=1
      RETURN
      END
```

APPENDIX E: User Defined Interface Friction

This subroutine may be provided by the user to set the Coulomb friction coefficients. This option is activated by the *USER_INTERFACE_FRICTION keyword. The arguments are defined in the listing provided below.

```
      SUBROUTINE USRFRC (NSI,TIME,CYCLE,DT2,NSLAVE,AREAS,XS,YS,ZS,
     . MSN,MASTRS,AREAM,XCM,YCM,ZCM,STFSN,STFMS,FORCEN,RVX,RVY,RVZ,
     . FRIC1,FRIC2,FRIC3,FRIC4,NINPUT,UA,SIDE)
C*********************************************************************
C|    LIVERMORE SOFTWARE TECHNOLOGY CORPORATION (LSTC)              |
C|    ------------------------------------------------------------  |
C|    COPYRIGHT 1987, 1988, 1989 JOHN O. HALLQUIST, LSTC            |
C|    ALL RIGHTS RESERVED                                           |
C*********************************************************************
C
      INTEGER CYCLE
      CHARACTER*(*) SIDE
      DIMENSION UA(*),MASTRS(4),XCM(4),YCM(4),ZCM(4)
C
C
C     USER SUBROUTINE FOR INTERFACE FRICTION CONTROL
C
C     NOTE:  LS-DYNA USES AN INTERNAL NUMBERING SYSTEM TO
C            ACCOMODATE ARBITRARY NODE NUMBERING.  TO ACCESS
C            INFORMATION FOR USER NODE N, ADDRESS ARRAY LOCATION M,
C            M=LQF(N,1).  TO OBTAIN USER NODE NUMBER, N,
C            CORRESPONDING TO ARRAY ADDRESS M, SET N=LQFINV(M,1)
C
C     ARGUMENTS:
C        NSI           =NUMBER OF SLIDING INTERFACE
C        TIME          =CURRENT SOLUTION TIME
C        CYCLE         =CYCLE NUMBER
C        DT2           =TIME STEPS SIZE AT N+1/2
C        NSLAVE        =SLAVE NODE NUMBER IN LS-DYNA INTERNAL
C                       NUMBERING
C        AREAS         =SLAVE NODE AREA (INTERFACE TYPES 5&10 ONLY)
C        XS            =X-COORDINATE SLAVE NODE (PROJECTED)
C        YS            =Y-COORDINATE SLAVE NODE (PROJECTED)
C        ZS            =Z-COORDINATE SLAVE NODE (PROJECTED)
C        MSN           =MASTER SEGMENT NUMBER
C        MASTRS(4)     =MASTER SEGMENT NODE IN LS-DYNA INTERNAL
C                       NUMBERING
C        AREAM         =MASTER SEGMENT NUMBER
C        XCM(4)        =X-COORDINATES MASTER SURFACE (PROJECTED)
C        YCM(4)        =Y-COORDINATES MASTER SURFACE (PROJECTED)
C        ZCM(4)        =Z-COORDINATES MASTER SURFACE (PROJECTED)
C        STFSN         =SLAVE NODE PENALTY STIFFNESS
C        STFMS         =MASTER SEGMENT PENALTY STIFFNESS
C        FORCEN        =NORMAL FORCE
C        RVX,RVY,RVZ,=RELATIVE X,Y,Z-VELOCITY BETWEEN SLAVE NODE AND
                        MASTER SEGMENT
```

```
C*******************************************************************
C     THE FOLLOWING VALUES ARE TO BE SET BY USER
C
C        FRIC1        =STATIC FRICTION COEFFICIENT
C        FRIC2        =DYNAMIC FRICTION COEFFICIENT
C        FRIC3        =DECAY CONSTANT
C        FRIC4        =VISCOUS FRICTION COEFFICIENT (SETTING FRIC4=0
                       TURNS THIS OPTION OFF)
C
C*******************************************************************
C
C        NINPUT       =NUMBER OF VARIABLES INPUT INTO UA
C        UA(*)        =USERS' ARRAY, FIRST NINPUT LOCATIONS
C                      DEFINED BY USER.  THE LENGTH OF THIS
C                      ARRAY IS DEFINED ON CONTROL CARD 15.
C                      THIS ARRAY IS UNIQUE TO INTERFACE NSI.
C
C        SIDE         ='MASTER' FOR FIRST PASS.  THE MASTER
C                      SURFACE IS THE SURFACE DESIGNATED IN THE
C                      INPUT.
C                     ='SLAVE' FOR SECOND PASS AFTER SLAVE AND
C                      MASTER SURFACES HAVE BE SWITCHED FOR
C                      THE TYPE 3 SYMMETRIC INTERFACE TREATMENT
C
C*******************************************************************
C
      RETURN
      END
```

APPENDIX F: 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-DYNA3D. The two input decks must be provided in the same set of consistent units. This can potentially require a major conversion to either the OSP input or the LS-DYNA input. With two legitimate and consistent input decks, the coupled program should run to completion with no problems. Additional inputs are required to make the models interact between the OSP and LS-DYNA3D portions of the run.

The simplest form of a coupled simulation is simply to include a single body in an OSP run. No special modifications are needed to the OSP input deck for use in the coupled simulation. Ellipsoids and planes in the OSP are usually attached to "segments" which correspond to LS-DYNA3D "rigid bodies." Because the coupling procedure works on the basis of shared information on LS-DYNA rigid bodies with the OSP segments, the ellipsoids/planes listed in the OSP section must correspond to the segments which are to be coupled. These ellipsoids and planes may be actual geometry which is used for contact, or they may be simply artificial shapes to permit the data transfer between the OSP and LS-DYNA.

DUMMY MODELING

The dummy is typically modeled entirely within the OSP. The coupling of the dummy into LS-DYNA requires the creation of a separate LS-DYNA rigid body material for each segment of the OSP. The easiest way to create a mesh for the model is to set the LS-DYNA rigid body coupling option to 2.0. This causes LS-DYNA to search all of the ellipsoids connected to the appropriate segment and generate meshes which are then slaved to the OSP dummy. Thus, with minimal input, a complete dummy may be generated and the kinematics may be traced in LS-DYNA3D and displayed in the LS-DYNA post-processor, LS-POST

Once the basic dummy coupling has been accomplished, the deformable finite element structure can be added. Assuming that an ellipsoid is available for the steering wheel, a flat airbag can be added in the proper location. One or more nodes must be attached to the steering wheel. This is done by identifying the attached nodes as "Extra Nodes for Rigid Body" which is input in LS-DYNA3D by *CONSTRAINED_EXTRA_NODES_Option. The nodes are slaved to the LS-DYNA3D material which has been coupled to the MADYMO steering wheel model. Contact must now be identified between the airbag and the steering wheel, the windshield, and the various body parts which may be affected. This requires the use of one geometric contact entity (see *CONTACT_ENTITY) for each plane or ellipsoid which may interact with the airbag. A control volume specifying inflation properties for the airbag must be specified (see *AIRBAG_*OPTION*) to complete the model.

AIRBAG MODELING

Modeling of airbags is accomplished by use of shell or membrane elements in conjunction with a control volume (see *AIRBAG_*OPTION*) and possibly a single surface contact algorithm to eliminate interpenetrations during the inflation phase (see *CONTACT_*OPTION*). The contact types showing an "a" in front are most suited for airbag analysis. Current recommended material types for the airbags are:

*MAT_ELASTIC = Type 1. Elastic

*MAT_COMPOSITE_DAMAGE = Type 22. Layered orthotropic elastic for composites

*MAT_FABRIC = Type 34. Fabric model for folded airbags

Model 34 is a “fabric” model which can be used for flat bags. As a user option this model may or may not support compression.

The elements which can be used are as follows:

Belytschko-Tsay quadrilateral with 1 point quadrature. This element behaves rather well for folded and unfolded cases with only a small tendency to hourglass. The element tends to be a little stiff. Stiffness form hourglass control is recommended.

Belytschko-Tsay membrane. This model is softer than the normal Belytschko-Tsay element and can hourglass quite badly. Stiffness form hourglass is recommended. As a better option, the fully integrated Belytschko-Tsay membrane element can be chosen.

C0 Triangular element. The C0 triangle is very good for flat bag inflation and has no tendency to hourglass.

The best choice is a specially developed airbag membrane element with quadrilateral shape. This is an automatic choice when the fabric material is used.

As an airbag inflates, a considerable amount of energy is transferred to the surrounding air. This energy transfer decreases the kinetic energy of the bag as it inflates. In the control volume logic, this is simulated either by using either a mass weighted damping option or a back pressure on the bag based on a stagnation pressure. In both cases, the energy that is absorbed is a function of the fabric velocity relative to a rigid body velocity for the bag. For the mass weighted case, the damping force on a node is proportional to the mass times the damping factor times the velocity vector. This is quite effective in maintaining a stable system, but has little physical justification. The latter approach using the stagnation pressure method estimates the pressure needed to accelerate the surrounding air to the speed of the fabric. The formula for this is:

$$P = Area \times \alpha \times \left(\left(\vec{V}_i - \vec{V}_{cg} \right) \cdot \hat{n} \right)^2$$

This formula accomplishes a similar function and has a physical justification. Values of the damping factor, α, are limited to the range of 0 to 1, but a value of 0.1 or less is more likely to be a good value.

KNEE BOLSTER

The knee-to-knee bolster interactions are characterized by the stiffness of the knee being comparable to that of the knee bolster. Therefore, modeling the knee as a rigid body may produce large errors in the interaction forces. Calibrated force-deflection curves could be determined, but they would have no predictive value for slight changes to knee bolster designs. For this reason, a more accurate modeling of the compliance of the knee bolster and the knee is required.

The knee can be modeled as a combined rigid/deformable body. The rigid body is coupled to the OSP. Overlaying the rigid body are brick elements which model the “skin” that exists over the knees of the dummy. These brick elements use material type 6 (*MAT_VISCOELASTIC) which is a viscoelastic model that does a reasonable job of approximating the hysteretic behavior of rubbers. The inner layer of the brick elements is attached to the rigid body through the

*CONSTRAINED_EXTRA_NODES Option. Between the knee bolster is a SURFACE-TO-SURFACE contact definition.

COMMON ERRORS

1. Improper airbag inflation or no inflation.

The most common problem is inconsistency in the units used for the input constants. An inflation load curve must also be specified. The normals for the airbag segments must all be consistent and facing outwards. If a negative volume results, this can sometimes be quickly cured by using the "flip" flag on the control volume definition to force inward facing normals to face outwards.

2. Excessive airbag distortions.

Check the material constants. Triangular elements should have less distortion problems than quadrilaterals. Overlapped elements at time zero can cause locking to occur in the contact leading to excessive distortions. The considerable energy input to the bag will create numerical noise and some damping is recommended to avoid problems.

3. The dummy passes through the airbag.

A most likely problem is that the contacts are improperly defined. Another possibility is that the models were developed in an incompatible unit system. The extra check for penetration flag if set to 1 on the contact control cards variable PENCHK in the *CONTACT_... definitions may sometimes cause nodes to be prematurely released due to the softness of the penalties. In this case the flag should be turned off.

4. The OSP fails to converge.

This may occur when excessively large forces are passed to the OSP. First, check that unit systems are consistent and then look for improperly defined contacts in the LS-DYNA input.

5. Time step approaches zero.

This is almost always in the airbag. If elastic or orthotropic (*MAT_ELASTIC or *MAT_COMPOSITE material 1 or 22) is being used, then switch to fabric material *MAT_FABRIC which is less time step size sensitive and use the fully integrated membrane element. Increasing the damping in the control volume usually helps considerably. Also, check for "cuts" in the airbag where nodes are not merged. These can allow elements to deform freely and cut the time step to zero.

APPENDIX G: Interactive Graphics Commands

Only the first four or less characterers of command are significant. These commands are available in the interactive phase of LS-DYNA. The interactive graphics 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 disenable postscript mode on the PC and eps file is written as picture is drawn; remove eofs and initgraphics for eps use.
QUIT	Same as execute.
RANGE	Set fix range for contour levels; range <minvalue> <maxvalue>.
RAX	Reflect model about xy plane; restore command will switch-off reflections.
RAY	Reflect model about yz plane; restore command will switch-off reflections.
RAZ	Reflect model about zx plane, restore command will switch-off reflections.
RESTORE	Restores model to original position, also switches off element and node numbers, slice capper, reflections and cut model.
RETURN	Exit.
RGB	Change color red green blue element <mat #> <red> <green> <blue>.
RX	Rotate model about x axis.
RY	Rotate model about y axis.
RZ	Rotate model about z axis.
SAVE	Set or unset saving of display for animation.
SEQUENCE	Periodic plot during execution; SEQ <# of cycles> <commands> EXE.
SHR	Shrink element facets towards centoids in subsequent views, shrink <value>.
SIP	Select shell integration point for contour; SIP <#>.

SLICE	Slice model a z-minimum plane; slice <value in normalized model dimension> this feature is removed after using restore. Slice enables internal details for brick elements to be used to generate new polygons on the slice plane.
SNORMAL	Set or unset display of shell direction normals to indicate topology order.
SPOT	Draw node numbers on model spot <first #> <last # for range>.
TAURUS	LS-DYNA database, TAU <state #>, or state <state #>, reads LS-TAURUS file to extract previous state data.
TRIAD	Set or unset display of axis triad.
TSHELL	Set or unset shell element thickness simulation in subsequent views.
TV	Change display type.
TX	Translates model along x axis.
TY	Translates model along y axis.
TZ	Translates model along z axis.
V	Display model using painters algorithm.
VECTOR v or d	View with vector arrows of velocity or displacement; <v> or <d>.
ZB	Switch on or off zbuffer algorithm for subsequent view; or draw commands.
ZIN	Zoom in using mouse to set display size and position.
ZMA	Set position of zmax plane; ZMAX <value in normalized model dimesions>.
ZMI	Set position of zmin plane; ZMIN <value in normalized model dimesions>.
ZOUT	Zoom out using mouse to set displays size expansion and position.

APPENDIX H: Interactive Material Model Driver

INTRODUCTION

The interactive material model driver in LS-DYNA allows calculation of the material constitutive response to a specified strain path. Since the constitutive model subroutines in LS-DYNA3D are directly called by this driver, the behavior of the constitutive model is precisely that which can be expected in actual applications. In the current implementation the constitutive subroutines for both shell elements and solid elements can be examined.

INPUT DEFINITION

The material model driver is invoked by setting the total number of beam, shell, and solid elements to zero in a standard LS-DYNA input file. The number of material model definitions should be set to one, the number of load curves should be nine, and the termination time to the desired length of the driver run. The complete state dump interval is interpreted as the time step to be used in the material model driver run. Plotting information is saved for every step of a driver run and sufficient memory is allocated to save this information in core for the interactive plotting phase.

The input deck consists only of the TITLE card, the CONTROL cards, one MATERIAL DEFINITION, and NINE LOAD CURVES describing the strain path should be defined. These nine curves define the time history of the displacement gradient components shown in Table H.1.

The velocity gradient matrix, L_{ij}, is approximated by taking the time derivative of the components in Table H.1. If these components are considered to form a tensor S_{ij} , then

$$L_{ij}(t) = \frac{S_{ij}(t) - S_{ij}(t_{k-1})}{(t - t_k)}$$

and the strain rate tensor is defined as

$$d_{ij} = \frac{L_{ij} + L_{ij}^t}{2}$$

and the spin tensor as

$$\omega_{ij} = \frac{L_{ij} - L_{ij}^t}{2}$$

Table H.1. Load Curve Definitions versus Time

Load Curve Number	Component Definition
1	$\frac{\partial u}{\partial x}$
2	$\frac{\partial v}{\partial y}$
3	$\frac{\partial w}{\partial z}$
4	$\frac{\partial u}{\partial y}$
5	$\frac{\partial v}{\partial x}$
6	$\frac{\partial u}{\partial z}$
7	$\frac{\partial w}{\partial x}$
8	$\frac{\partial v}{\partial z}$
9	$\frac{\partial w}{\partial y}$

INTERACTIVE DRIVER COMMANDS

After reading the input file and completing the calculations, LS-DYNA gives a command prompt to the terminal. A summary of the available interactive commands is given below. An on-line help package is available by typing HELP.

ACCL	Scale all abscissa data by f. Default is f=1.
ASET amin omax	Set min and max values on abscissa to amin and amax, respectively. If amin=amax=0, scaling is automatic.
CHGL n	Change label for component n. LS-DYNA prompts for new label.
CONTINUE	Re-analyze material model.
CROSS c_1 c_2	Plot component c_1 versus c_2.
ECOMP	Display component numbers on the graphics display: 1 x-stress, 2 y-stress, 3 z-stress, 4 xy-stress, 5 yz-stress, 6 zx-stress, 7 effective plastic strain, 8 pressure, 9 von Mises (effective) stress, 10 1st principal deviatoric stress, 11 2nd principal deviatoric stress, 12 3rd principal deviatoric stress, 13 maximum shear stress, 14 1st principal stress, 15 2nd principal stress, 16 3rd principal stress, 17 ln (v/v0), 18 relative volume, 19 v0/v - 1.0, 20 1st history variable, 21 2nd history variable. Adding 100 or 400 to component numbers 1-16 yields strains and strain rates, respectively.
FILE name	Change pampers filename to name for printing.
GRID	Graphics displays will be overlaid by a grid of orthogonal lines.
NOGRID	Graphics displays will not be overlaid by a grid of orthogonal lines.

OSCL	Scale all ordinate data by f. Default is f=1.
OSET omin omax	Set min and max values on ordinate to omin and omax, respectively. If omin=omax=0, scaling is automatic.
PRINT	Print plotted time history data into file "pampers." Only data plotted after this command is printed. File name can be changed with the "file" command.
QUIT, END, T	Exit the material model driver program.
RDLC m n r_1 z_1 ... r_n z_n	Redefine load curve m using n coordinate pairs (r_1,z_1) (r_2,z_2),...(r_n,z_n).
TIME c	Plot component c versus time.
TV n	Use terminal output device type n. LS-DYNA provides a list of available devices.

Presently, the material model drive is implemented for solid and shell element material models. The driver does not yet support material models for beam elements.

APPENDIX I: VDA Database

VDA surfaces describe the surface of geometric entities and are useful for the simulation of sheet forming problems. The German automobile and automotive supplier industry (VDA) has defined the VDA guidelines [VDA, 1987] for a proper surface definition used for the exchange of surface data information. In LS-DYNA, this format can be read and used directly. Some files have to be provided for proper linkage to the motion of the correlation parts/materials in LS-DYNA3D.

Linking is performed via names. To these names surfaces are attached, which in turn can be linked together from many files externally to LS-DYNA. Thus, arbitrary surfaces can be provided by a preprocessor and then can be written to various files. The so called VDA file given on the LS-DYNA3D execution line via V=**vda** contains references to all other files. It also contains several other parameters affecting the treatment in the contact subroutines; see below.

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

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

alias definitions

}

alias definitions

followed by optional runtime parameters and a final end statement.

The file, **afile,** is optional, and if given must be the name of an ASCII input file formatted in accordance with the VDA Surface Interface Definitions as defined by the German automobile and automotive supply industry. **bfile** is required, and is the name of a binary VDA file. In a first run **afile** is given and **bfile** is created. In any further run, if the definitions have not changed, **afile** can be dropped and only **bfile** is needed. The purpose of **bfile** is that it allows for much faster initialization if the same VDA surfaces are to be used in a future LS-DYNA run.

If **afile** is given, **bfile** will always be created or overwritten. The alias definitions are used for linking to LS-DYNA and between the various surface definitions in the files defined by **afile** and **bfile.**

The alias definitions are of the form

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

where **name** is any string of up to 12 characters, and el1,...,eln are the names of VDA elements as specified in **afile**. The list of elements can be empty, in which case all the SURF and FACE VDA elements in **afile** will be used. Care should be taken to ensure that the alias **name** is unique, not only among the other aliases, but among the VDA element names in **afile**. This collection of VDA elements can later be indicated by the alias **name**. In particular, **name** may appear in later alias definitions.

Often it is required that a punch or die be created by a simple offset. This can be achieved in the **vda** files in two ways, either on VDA elements directly, or on parts defined by aliases. This feature offers great capability in generating and using surface data information.

Offset version 1:

As an option, the keyword **offset** may appear in the alias list which allows a new surface to be created as a normal offset (plus translation) of a VDA element in the file. The keyword **offset** my be applied to VDA elements only, not aliases. The usage of **offset** follows the form

offset elem **normal x y z**

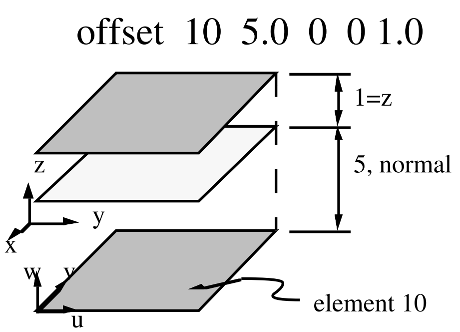
where **normal** is the amount to offset the surface along the normal direction, and **x**,**y**,**z** are the translations to be applied. The default normal direction is given by the cross product of the local u and v directions on the VDA surface, taken in that order. **normal** can be negative.

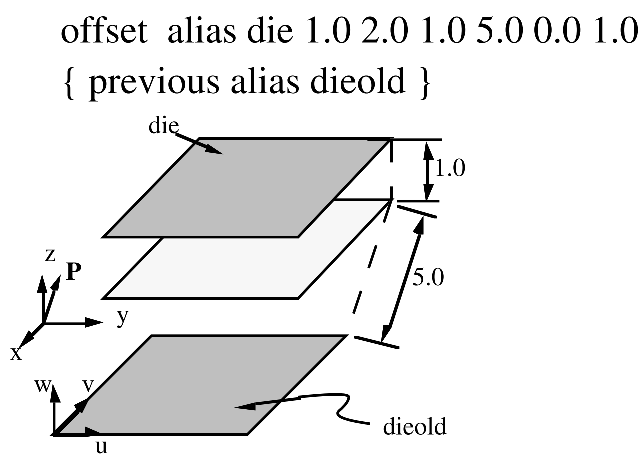
Offset version 2:

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

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

where **normal**, **x**, **y**, and **z** are defined as in the offset keyword. A reference point $\mathbf{x_c}$, $\mathbf{y_c}$, and $\mathbf{z_c}$ defines a point in space which determines the normal direction to the VDA surface, which is a vector from the origin to $P(x_c,y_c,z_c)$. 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 occuring on one edge can pass to the adjacent edge. The default value is 0.01.

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

iterate [8] Maximum number of surface iterations allowed. Since points being tracked are checked every cycle, if convergence fails it will be tried again next cycle, so setting this parameter high does not necessarily help much. On the other hand, a point converging to a crease in the VDA surface (a crease between patches with discontinuous derivative, for example) may bounce back and forth between patches up to this many times, without actually moving. Hence, this value should not be too large. The default value is 8.

el_size [t mx mn]

Controls the generation of elements where:

t =surface tolerance for mesh generation,

mx=maximum element size to generate,

mn=minimum element size to generate.

The default values are [0.25 100. 1.0]

aspect [s1 s2]

Controls the generation of elements where:

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

s2=maximum aspect ratio for any generated element.

The default values are [1.5 4.0]

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

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

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

somepatch [n] Like onepatch, but generates an element for 1 out of every [n] patches.

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
```

<u>Explanation:</u>

vda1 This file contains the sufaces/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 Δr × Δz 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 Δr × Δz window. This window is set until it is reset or the "UNFIX" command is typed.
UNFIX	Release current display window set by the "FIX", "GSET", "FSET" or "SETF" commands.
UZ a b Δl	Zoom in at point (a,b) with window Δl where a, b, and Δl are numbers between 0 and 1. The picture is assumed to lie in a unit square.
UZG	Cover currently displayed picture with a 10 by 10 square grid to aid in zooming with the unity zoom, "UZ", command.
UZOU a b Δl	Zoom out at point (a,b) with window Δl where a, b, and Δl are numbers between 0 and 1. The current window is scaled by the factor 1/Δl.The picture is assumed to lie in a unit square.

Z r z Δl	Zoom in at point (r,z) with window Δl.
ZOUT r z Δl	Zoom out at point (r,z) with window Δl. The window is enlarged by the ratio of the current window and Δl. The cursor may be used to zoom out via the cursor command DZOU and entering two points with the cursor to define the window. The ratio of the current window with the specified window determines the picture size reduction. An alternative cursor command, DZZO, may be used and only needs one point to be entered at the location where the reduction (2×) is expected.

Graphics Window Controls for x versus y plots

The following commands apply to line plots, interface plots, etc.

ASCL f_a	Scale all abscissa data by f_a. The default is $f_a = 1$.
ASET amin amax	Set minimum and maximum values on abscissa to amin and amax, respectively. If amin=amax=0.0 (default) LS-DYNA determines the minimum and maximum values.
OSCL f_o	Scale all ordinate data by f_o. The default is $f_o = 1$.
OSET omin omax	Set minimum and maximum values on ordinate to omin and omax, respectively. If omin=omax=0.0 (default) LS-DYNA determines the minimum and maximum values.
SMOOTH n	Smooth a data curve by replacing each data point by the average of the 2n adjacent points. The default is n=0.

Mesh Display Options

ELPLT	Plot element numbers on mesh of material n.
FSOFF	Turn off the “FSON” command.
FSON	Plot only free surfaces and slideline interfaces with “O” command. (Must be used before “O” command.)
G	View mesh.
GO	View mesh right of centerline and outline left of centerline.
GS	View mesh and solid fill elements to identify materials by color.
M n	Material n is to be rezoned.
MNOFF	Do not plot material numbers with the “O”, “G”, and “GO” commands (default).

MNON	Plot material numbers with "O", "G", and "GO" commands.
NDPLT	Plot node numbers on mesh of material n.
O	Plot outlines of all material.
RPHA	Reflect mesh, contour, fringe, etc., plots about horizontal axis. Retyping "RPHA" turns this option off.
RPVA	Reflect mesh, contour, fringe, etc., plots about vertical axis. Retyping "RPVA" turns this option off.
TN r z Δl	Type node numbers and coordinates of all nodes within window (r ± Δl/2, z ± Δl/2).
UG	Display undeformed mesh.
V	Display material n on graphics display. See command M.
VSF	Display material n on graphics display and solid fill elements.

Mesh Modifications

BACKUP	Restore mesh to its previous state. This command undoes the result of the last command.
BLEN s	Smooth option where s=0 and s=1 correspond to equipotential and isoparametric smoothing, respectively. By letting $0 \le s \le 1$ a combined blending is obtained.
CN m r z	Node m has new coordinate (r,z).
DEB n f_1 l_1 ... f_n l_n	Delete n element blocks consisting of element numbers f_1 to l_1, f_2 to l_2 ... , and f_o l_n inclusive. These elements will be inactive when the calculation resume.
DE e_1 e_2	Delete elements e_1 to e_2.
DMB n m_1 m_2 ... m_n	Delete n material blocks consisting of all elements with material numbers m_1, m_2,..., and m_n. These materials will be inactive when the calculations resume.
DM n m_1 m_2 ... m_n	Delete n materials including m_1, m_2,..., and m_n.
DZER k d incr nrow	Delete element row where k is the kept element, d is the deleted element, incr is the increment, and nrow is the number of elements in the row.
DZLN number n_1 n_2 n_3...n_{last}	Delete nodal row where number is the number of nodes in the row and n_1, n_2, ... n_{last} are the ordered list of deleted nodes.

DZNR l j incr	Delete nodal row where l is the first node in the row, j is the last node in the row, and incr is the increment.
R	Restore original mesh.
S	Smooth mesh of material n. To smooth a subset of elements, a window can be set via the "GSET", "FSET", OR "SETF" commands. Only the elements lying within the window are smoothed.

Boundary Modifications

A	Display all slidelines. Slave sides are plotted as dashed lines.
B	Determine boundary nodes and sides of material n and display boundary with nodes and side numbers.
BD m n	Dekink boundary from boundary node m to boundary node n (counterclockwise).
BDS s	Dekink side s.
DSL n l_1 l_2...l_n	Delete n slidelines including slideline numbers l_1 l_2..., and l_n.
ER m n	Equal space in r-direction boundary nodes m to n (counterclockwise).
ERS s	Equal space in the r-direction boundary nodes on side s.
ES m n	Equal space along boundary, boundary nodes m to n (counterclockwise).
ESS s	Equal space along boundary, boundary nodes on side s.
EZ m n	Equal space in z-direction boundary nodes m to n (counterclockwise).
EZS s	Equal space in the z-direction boundary nodes on side s.
MC n	Check master nodes of slideline n and put any nodes that have penetrated through the slave surface back on the slave surface.
MD n	Dekink master side of slideline n. After using this command, the SC or MC command is sometimes advisable.
MN n	Display slideline n with master node numbers.
SC n	Check slave nodes of slideline n and put any nodes that have penetrated through the master surface back on the master surface.

SD n	Dekink slave side of slideline n; after using this command, the SC or MC command is sometimes advisable.
SLN m n	Equal space boundary nodes between nodes m to n on a straight line connecting node m to n.
SLNS n	Equal space boundary nodes along side n on a straight line connecting the corner nodes.
SN n	Display slideline n with slave node numbers.
VS m n r	Vary the spacing of boundary nodes m to n such that r is the ratio of the first segment length to the last segment length.
VSS s r	Vary the spacing of boundary nodes on side s such that r is the ratio of the first segment length to the last segment length.

MAZE Line Definitions

B	Determine boundary nodes and sides of material n and display boundary with nodes and side numbers. See command "M".
LD n k l	Line defintion n for MAZE includes boundary nodes k to l
LDS n l	Line definition n for MAZE consists of side number l.
M n	Material n is active for the boundary command B.

Calculation Graphics Display Control Parameters

MOLP	Overlay the mesh on the contour, fringe, principal stress, and principal strain plots. Retyping "MOLP" turns this option off.
NLOC	Do not plot letters on contour lines.
NUMCON n	Plot n contour levels. The default is 9.
PLOC	Plot letters on contour lines to identify their levels (default).
RANGE r_1 r_2	Set the range of levels to be between r_1 and r_2 instead of in the range chosen automatically by LS-DYNA. To deactivate this command, type RANGE 0 0.

Calculation Graphics Display

CONTOUR c n m_1 m_2...m_n	Contour component number c on n materials including materials m_1, m_2, ..., m_n. If n is zero, only the outline of material m_1 with contours is plotted. Component numbers are given in Table 1.
FRINGE c n m_1 m_2...m_n	Fringe component number c on n materials including m_1, m_2,...,m_n. If n is zero, only the outline of material m_1 with contours is plotted. Component numbers are given in Table 1.
IFD n	Begin definition of interface n. If interface n has been previously defined, this command has the effect of destroying the old definition.
IFN l m	Include boundary nodes l to m (counterclockwise) in the interface definition. This command must follow the "B" command.
IFP c m	Plot component c of interface m. Component numbers are given in Table 2.
IFS m	Include side m in the interface definition. Side m is defined for material n by the "B" command.
IFVA r_c z_c	Plot the angular location of the interface based on the center point (r_c,z_c) along the abscissa. Positive angles are measured counterclockwise from the y axis.
IFVS	Plot the distance along the interface from the first interface node along the abscissa (default).
LINE c n m_1 m_2...m_n	Plot variation of component c along line defined with the "NLDF", "PLDF", "NSDF", or the "NSSDF" commands given below. In determining variation, consider n materials including material number m_1, m_2,...m_n.
NCOL n	Number of colors in fringe plots is n. The default value for n is 6 which includes colors magenta, blue, cyan, green, yellow, and red. An alternative value for n is 5 which eliminates the minimum value magenta.
NLDF n n_1 n_2...n_3	Define line for "LINE" command using n nodes including node numbers n_1, n_2,...n_n. This line moves with the nodes.
NSDF m	Define line for "LINE" command as side m. Side m is defined for material n by the "B" command.
NSSDF l m	Define line for "LINE" command and that includes boundary nodes l to m (counterclockwise) in the interface definitions. This command must follow the "B" command.
PLDF n r_1 z_1...r_n z_n	Define line for "LINE" command using n coordinate pairs (r_1,z_1), (r_2,z_2), ...(r_n,z_n). This line is fixed in space.

PRIN c n m_1 m_2...m_n

Plot lines of principal stress and strain in the yz plane on n materials including materials m_1, m_2,...,m_n. If n is zero, only the outline of material m_1 is plotted. The lines are plotted in the principal stress and strain directions. Permissible component numbers in Table 1 include 0, 5, 6, 100, 105, 106,...,etc. Orthogonal lines of both maximum and minimum stress are plotted if components 0, 100, 200, etc. are specified.

PROFILE c n m_1 m_2...m_n

Plot component c versus element number for n materials including materials m_1, m_2,...,m_n. If n is Ø then component c is plotted for all elements. Component numbers are given in Table 1.

VECTOR c n m_1 m_2...m_n

Make a vector plot of component c on n materials including materials m_1, m_2,...,m_n. If n is zero, only the outline of material m_1 with vectors is plotted. Component c may be set to “D” and “V” for vector plots of displacement and velocity, respectively.

No.	Component	No.	Component
1	y	21*	ln (V/Vo) (volumetric strain)
2	z	22*	y-displacement
3	hoop	23*	z-displacement
4	yz	24*	maximum displacement
5	maximum principal	25*	y-velocity, y-heat flux
6	minimum principal	26*	z-velocity, y-heat flux
7	von Mises (Appendix A)	27*	maximum velocity, maximum heat flux
8	pressure or average strain	28	ij normal
9	maximum principal-minimum principal	29	jk normal
10	y minus hoop	30	kl normal
11	maximum shear	31	li normal
12	ij and kl normal (Appendix B)	32	ij shear
13	jk and li normal	33	jk shear
14	ij and kl shear	34	kl shear
15	jk and li shear	35	li shear
16	y-deviatoric	36*	relative volume V/Vo
17	z-deviatoric	37*	Vo/V-1
18	hoop-deviatoric	38*	bulk viscosity, Q
19*	effective plastic strain	39*	P + Q
20*	temperature/internal energy density	40*	density
41*-70*	element history variables		
71*	r-peak acceleration	76*	peak value of min in plane prin. stress
72*	z-peak acceleration	77*	peak value of maximum hoop stress
73*	r-peak velocity	78*	peak value of minimum hoopstress
74*	z-peak velocity	79*	peak value of pressure
75*	peak value of max. in plane prin. stress		

Table 1. Component numbers for element variables. By adding 100, 200 300, 400, 500 and 600 to the component numbers not followed by an asterick, component numbers for infinitesimal strains, lagrange strains, almansi strains, strain rates, extensions, and residual strain are obtained. Maximum and minimum principal stresses and strains are in the rz plane. The corresponding hoop quantities must be examined to determine the overall extremum. ij, jk, etc. normal components are normal to the ij, jk, etc side. The peak value database must be flagged on Control Card 4 in columns 6-10 or components 71-79 will not be available for plotting.

No.	Component
1	pressure
2	shear stress
3	normal force
4	tangential force
5	y-force
6	z-force

Table 2. Component numbers for interface variables.
In axisymmetric geometries the force is per radian.

Cursor Commands

DBD a b — Use cursor to define points a and b on boundary. Dekink boundary starting at a, moving counterclockwise, and ending at b.

DCN a b — Use cursor to define points a and b. The node closest to point a will be moved to point b.

DCSN n a — Move nodal point n to point a defined by the cursor.

DCNM a b — Use cursor to define points a and b. The node at point a is given the coordinate at point b.

DER a b — Use cursor to define points a and b on boundary. Equal space nodes in r-direction along boundary starting at a, moving counterclockwise, and ending at b.

DES a b — Use cursor to define points a and b on boundary. Equal space nodes along boundary starting at a, moving counterclockwise, and ending at b.

DEZ a b — Use cursor to define points a and b on boundary. Equal space nodes in z-direction along boundary starting at a, moving counterclockwise, and ending at b.

DTE a b — Use cursor to define points a and b on the diagonal of a window. The element numbers and coordinates of elements lying within the window are typed on the terminal.

DTN a b — Use cursor to define points a and b on the diagonal of a window. The node numbers and coordinates of nodal points lying within the window are typed on the terminal.

DTNC a — Use cursor to define point a. The nodal point number and nodal coordinates of the node lying closest to point a will be printed.

DVS a b r	Use cursor to define points a and b on boundary. Variable space nodes along boundary starting at a, moving 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 A.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.

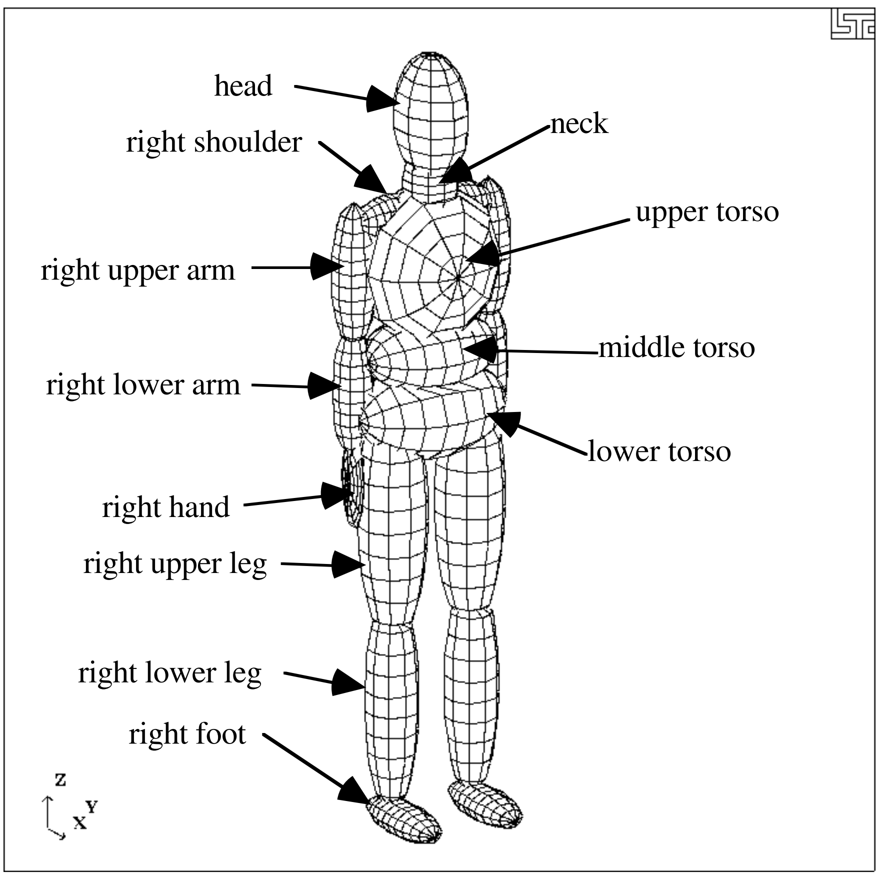


Figure K.1: 50th percentile male dummy in the nominal position.

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

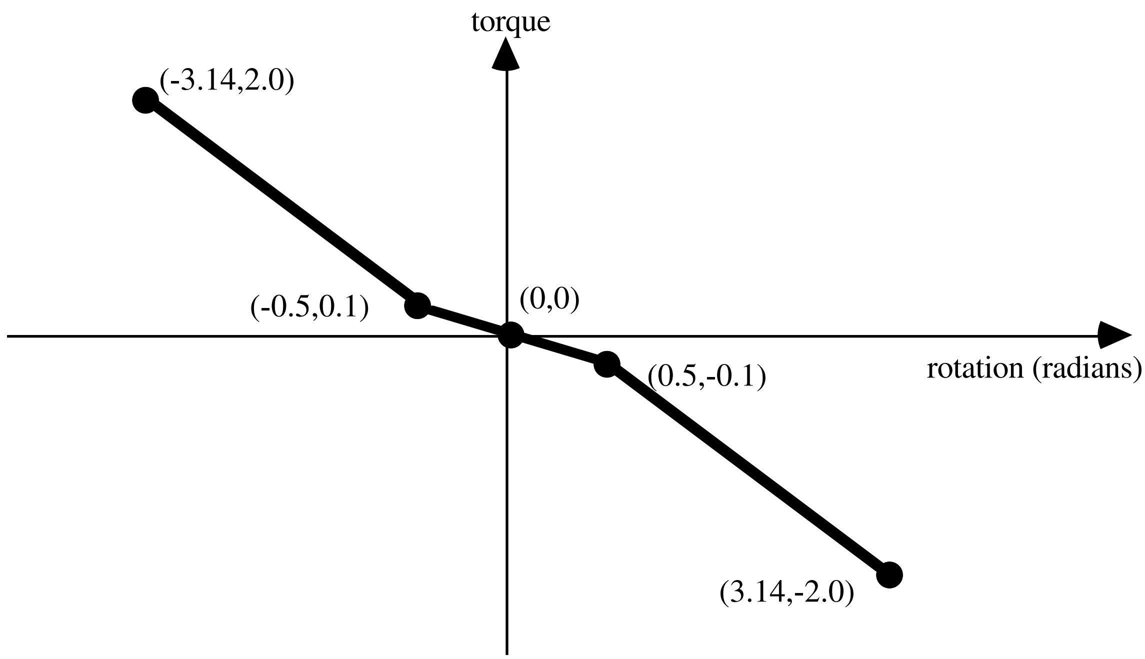


Figure K.2: Characteristic torque curve shape used by all joints.

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 lexion	+ = tilt right	0.0
upper neck	forward flexion	+ = nod fwd	0.0
upper neck	torsion	+ = twist left	0.0
left shoulder	abduction-adduction	+ = abduction	30.0
left shoulder	internal-external rotation	+ = external	-10.0
left shoulder	flexion-extension	- = fwd raise	-40.0
right shoulder	abduction-adduction	- = abduction	-30.0
right shoulder	internal-external rotation	- = external	10.0
right shoulder	flexion-extension	- = fwd raise	-40.0
left elbow	flexion-extension	+ = extension	-60.0
right elbow	flexion-extension	+ = extension	-60.0
left hip	abduction-adduction	+ = abduction	0.0
left hip	medial-lateral rotation	+ = lateral	0.0
left hip	flexion-extension	+ = extension	-80.0
right hip	abduction-adduction	- = abduction	0.0
right hip	medial-lateral rotation	- = lateral	0.0
right hip	flexion-extension	+ = extension	-80.0
left knee	flexion-extension	+ = flexion	50.0
right knee	flexion-extension	+ = flexion	50.0
left ankle	inversion-eversion	+ = eversion	0.0
left ankle	dorsi-plantar flexion	+ = plantar	0.0
left ankle	medial-lateral rotation	+ = lateral	0.0
right ankle	inversion-eversion	- = eversion	0.0
right ankle	dorsi-plantar flexion	+ = plantar	0.0
right ankle	medial-lateral rotation	- = lateral	0.0

In Figure 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.

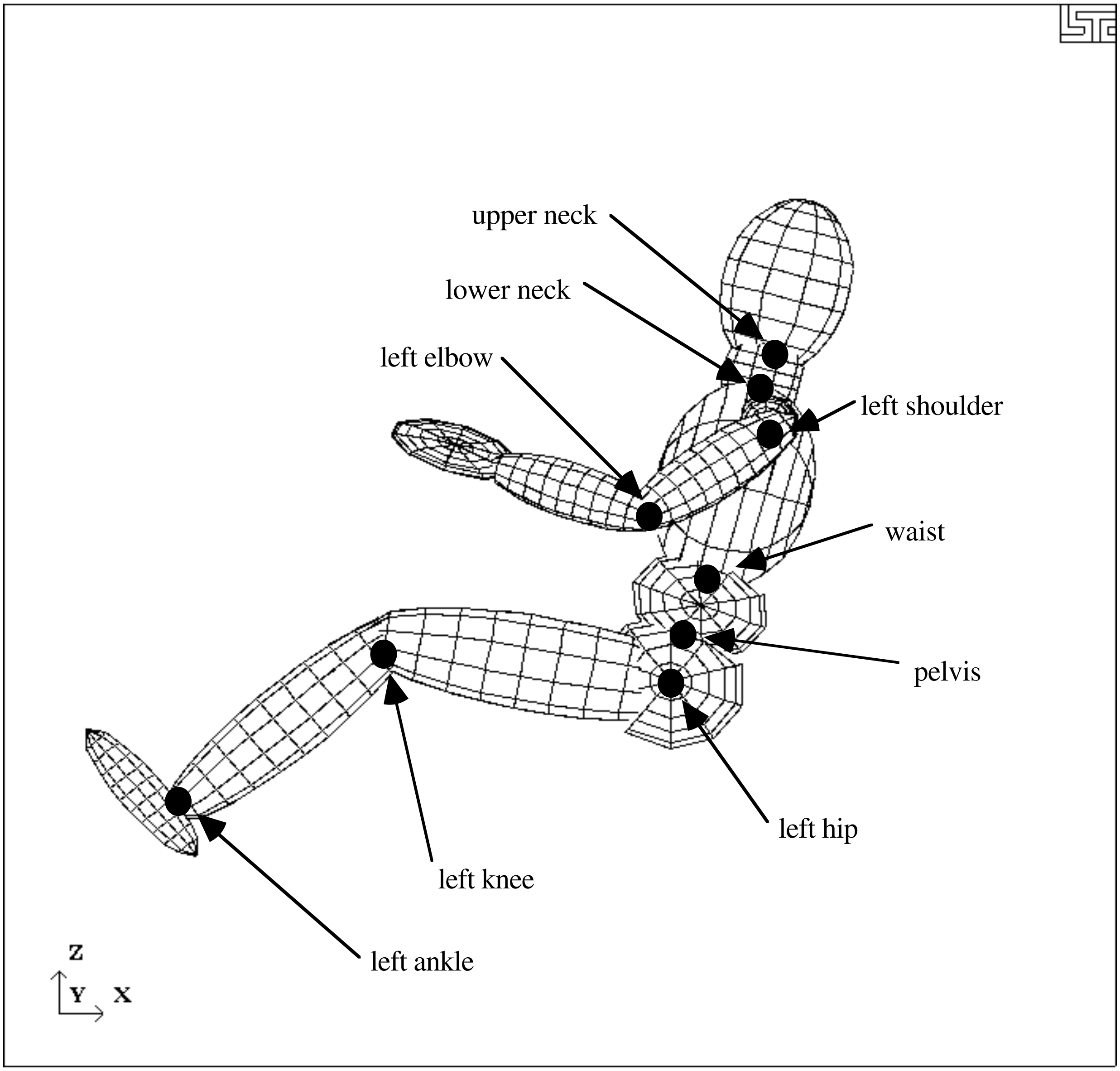


Figure K.3: Dummy seated using the file listed in Table K.3.

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
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 (z)	rotation (y)	rotation (z)

Joint springs of the *COMPONENT_HYBRIDIII dummies are formulated in the following manner.

$$T = a_{lo}(q - q_{lo}) + b_{lo}(q - q_{lo})^3 \qquad q \le q_{lo}$$

$$T = a_{hi}(q - q_{hi}) + b_{hi}(q - q_{hi})^3 \qquad q \ge q_{hi}$$

$$T = 0 \qquad q_{lo} < q < q_{hi}$$

where

T is the joint torque

q is the joint generalized coordinate

a_{lo} and b_{lo} are the linear and cubic coefficents, respectively, for the low regime

a_{hi} and b_{hi} are the linear and cubic coefficents, respectively, for the high regime

q_{lo} and q_{hi} are the activation values forhe 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_AVS
- *DATABASE_MOVIE
- *DATABASE_MPGS
- *DATABASE_TRACER
- *DATABASE_BINARY_XTFILE
- *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:

`trackpen`

If 1, any initial penetrations for slave nodes are compensated for in the contact algorithm. No nodes are moved to eliminate penetrations, and no initial penetration checking is performed. The algorithm detects these penetrations and allows for them in computing forces, so excessively large forces are avoided. As the slave node moves in such a way as to reduce or eliminate the penetration, the full contact distance/material thickness is imposed. Use of this option is encouraged as it can greatly help stability, particularly in models with many initial penetrations. By default this option is disabled.

`bucket`

Bucket sorting frequency for this contact interface

`lcbucket`

Load curve giving bucket sort frequency as a function of simulation time. Currently this option is not supported by any of the contact algorithms.

`nseg2track`

Number of contact segments to track for each slave node

`inititer`

Number of iterations for initial penetration checking

The defaults for each are taken from the corresponding options in the pfile (described below). For example, if you had the control card:

```
*CONTACT_SINGLE_SURFACE_TITLE
This is my title card
```

you could change this to

```
*CONTACT_SINGLE_SURFACE_TITLE_MPP
This is my title card
        1
```

to turn on the initial penetration tracking option. The serial/SMP code will ignore these options.

Output Files and Post-Processing

For performance reasons, many of the ASCII output files normally created by LS-DYNA have been combined into a new binary format used by MPP/LS-DYNA. There is a post-processing program 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 mes*nnnn*) and others. Most of these will be found in the local directory specified in the pfile.

The format of the d3plot file has not been changed. It will be created in the global directory, and can be directly handled with your current graphics post-processor.

Parallel Specific Options

There are a few new command line options that are specific to the MPP version of LS-DYNA.

In the serial and SMP versions of LS-DYNA, the amount of memory required to run the problem can be specified on the command line using the keyword *memory=XXX*, where *XXX* is the number of words of memory to be allocated. For the MPP code, this will result in each processor allocating *XXX* words of memory. If pre-decomposition has not been performed, one processor must perform the decomposition of the problem. This can require substantially more memory than will be required once execution has started. For this reason, there is a second memory command line option, *memory2=YYY*. If used together with the memory keyword, the decomposing processor will allocate *XXX* words of memory, and all other processors will allocate *YYY* words of memory.

For example, in order to run a 250,000 element crash problem on 4 processors, you might need *memory=80m* and *memory2=20m*. To run the same problem on 16 processors, you still need *memory=80m*, but can set *memory2=6m*. The value for *memory2* drops nearly linearly with the number of processors used to run the program, which works well for shared-memory systems.

The full deck restart capability is supported by the MPP version of LS-DYNA, but in a manner slightly different than the SMP code. Each time a restart dump file is written, a separate restart file is also written with the base name D3FULL. For example, when the third restart file d3dump03 is written (one for each processor, d3dump03.0000, d3dump03.0001, etc), there is also a single file written named d3full03. This file is required in order to do a full deck restart and the d3dump files are not used in this case by the MPP code. In order to perform a full deck restart with the MPP code, you first must prepare a full deck restart input file as for the serial/SMP version. Then, instead of giving the command line option *r=d3dump03* you would use the special option *n=d3full03*. The presence of this command line option tells the MPP code that this is a restart, not a new problem, and that the file d3full03 contains the geometry and stress data carried over from the previous run.

PFILE

There is a new command line option: p=pfile. pfile contains MPP specific parameters that affect the execution of the program. The file is split into sections, with several options in each section. Currently, these sections: **directory**, **decomposition**, **contact**, and **general** are available. First, here is a sample pfile:

```
directory {
global rundir
local /tmp/rundir
}
contact {
inititer 3
}
```

The file is case insensitive and free format input, 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 diffently 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 occuring 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 sychronization 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 effeciency 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 coodinate 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 inherient 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 tye 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:

new (x,y,z) = cylindrical coordinates (r,theta,z)

Knowing the order of the coordinates is important if combining transformations, as in the example below.

S2R x0 y0 z0 vx1 vy1 vz1 vx2 vy2 vz2

Just like the above, but for spherical coordinates. The (vx1,vy1,vz1) vector is the phi=0 axis.

New (x,y,z) = spherical coordinates (rho, theta, phi)

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, 2PI), z=[0,100]. In particular, the new y coordinates (theta) will be very small compared to the other coordinate directions. It is therefore likely that every decomposition domain will extend through the complete transformed y direction. This means that each domain will be a shell completely around the original cylinder. If you want to split the domains along radial lines, try this pair of transformations:

C2R 0 0 0 0 0 1 1 0 0

SY 5000

This will do the above C2R, but then scale y by 5000. This will result in the part appearing to be about 30,000 long in the y direction -- long enough that every decomposition domain will divide the part in this (transformed) y direction. The result will be decomposition domains that are radial "wedges" in the original part.

General combinations of transformations can be specified, and they are applied in order:

SX 5 SY .2 RZ 30

will scale x, then y, then rotate.

Execution of MPP/LS-DYNA

MPP/LS-DYNA runs under a parallel environment which provided by the hardware vendor. The execution of the program therefore varies from machine to machine. On some platforms, command line parameters can be passed directly on the command line. For others, the use of the names file is required. The names file is supported on all systems.

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

rsh <machine name> kill -INT <PID>

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

Here is a simple table to show how to run the program on various platforms. Of course, scripts are often written to mask these differences.

Platform	Execution Command
DEC Alpha	dmpirun –-np *n mpp-dyna*
Fujitsu	jobexec –vp *n* –mem *m* *mpp-dyna*
Hitachi	mpirun –-np *n mpp-dyna*
HP	*mpp-dyna* –np *n*
IBM	#!/bin./ksh export MP_PROC=*n* export MP_LABELIO=no export MP_EUILIB=us export MPI_EUIDEVICE=css0 poe *mpp-dyna*
NEC	mpirun –-np *n mpp-dyna*
SGI	mpirun –-np *n mpp-dyna*
Sun	tmrun –n-p *n mpp-dyna*

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 960 adds many additional implicit features, including new element formulations for linear and modal analysis.

SETTING UP AN IMPLICIT SIMULATION

The keyword *CONTROL_IMPLICIT_GENERAL is used to activate the implicit method. LS-DYNA can conduct either a linear or a nonlinear implicit analysis. The keyword *CONTROL_IMPLICIT_SOLUTION is used to select between these implicit analysis types. In addition, an implicit eigenvalue analysis can be performed to extract frequencies and mode shapes.

To perform a linear implicit analysis, use the *CONTROL_IMPLICIT_GENERAL keyword to activate the implicit method and to specify the time step size. Enter the termination time using the *CONTROL_TERMINATION keyword. For a single step analysis, select the step size to be equal to the termination time. Use the *CONTROL_IMPLICIT_SOLUTION keyword to request a linear analysis. Select linear element formulations using the *SECTION_SOLID and/or *SECTION_SHELL keywords. For best accuracy, a double precision version of LS-DYNA should be used for linear analysis.

To perform an eigenvalue analysis, use the *CONTROL_IMPLICIT_GENERAL keyword to activate the implicit method and to specify a time step size. Enter the termination time using the *CONTROL_TERMINATION keyword (the time step size and termination time must be nonzero, but will otherwise be ignored as LS-DYNA will presently just compute the eigenvalues and stop.) Use the *CONTROL_IMPLICIT_EIGENVALUE keyword to indicate the desired number of eigenvalues and frequency ranges of interest. For best accuracy, a double precision version of LS-DYNA should be used for eigenvalue analysis.

A nonlinear implicit simulation is typically divided into several steps. In a dynamic simulation, these are *time steps*. In a static simulation, these are *load steps*. Multiple steps may be used to divide the nonlinear behavior into manageable pieces, to obtain results at intermediate stages during the simulation, or perhaps to resolve a particular frequency of motion in dynamic simulations. In each step, an equilibrium geometry is sought which balances internal and external forces in the model. The *nonlinear equation solver* performs an iterative search using one of several Newton based methods. *Convergence* of this iterative process is obtained when norms of displacement and/or energy fall below user-prescribed tolerances.

Control parameters for the nonlinear equation solver are input using the keyword *CONTROL_IMPLICIT_SOLUTION. By default, the progress of the equilibrium search is not shown to the screen. This output can be activated either using the NLPRINT input parameter, or interactively toggled on and off by entering "<ctrl-c> nlprint". The box below shows a typical iteration sequence, where the norms of displacement (du/u) and energy (Ei/E0) are displayed. When these norms are reduced below user prescribed tolerances (default 1.0e-3 and 1.0e-2, respectively), equilibrium is reached within sufficient accuracy, the iteration process is said to have *converged*, and the solution proceeds to the next time step.

```
BEGIN time step     3

============================================================

               time =  1.50000E-01

  current step size =  5.00000E-02

Iteration:   1     *|du|/|u| =  3.4483847E-01     *Ei/E0 =  1.0000000E+00

Iteration:   2     *|du|/|u| =  1.7706435E-01     *Ei/E0 =  2.9395439E-01

Iteration:   3     *|du|/|u| =  1.6631174E-03     *Ei/E0 =  3.7030904E-02

Iteration:   4     *|du|/|u| =  9.7088516E-05     *Ei/E0 =  9.6749731E-08
```

A typical print-out showing the progress of the nonlinear equation solver. By default, the progress of the equilibrium search is not shown to the screen. This output can be activated either using the NLPRINT input parameter, or interactively toggled on and off by entering: "<ctrl-c> nlprint".

LINEAR EQUATION SOLVER

Within each equilibrium iteration, a linear system of equations of the form $\mathbf{K}\Delta\mathbf{u} = \mathbf{R}$ must be solved. To do this, the stiffness matrix $\mathbf{K}$ is inverted and applied to the out-of-balance load or residual $\mathbf{R}$, yielding a displacement increment $\Delta\mathbf{u}$. Storing and solving this linear system represents a large portion of the memory and CPU costs of an implicit analysis.

Control parameters for solving the linear system $\mathbf{K}\Delta\mathbf{u} = \mathbf{R}$ are input using the keyword *CONTROL_IMPLICIT_SOLVER. Several different linear equation solvers are available, including direct (Gaussian elimination) and iterative (conjugate gradient, Lanczos) methods. A sparse storage scheme is used to minimize memory requirements, which are still often substantial. Two options are available for matrix reordering, allowing nodes and elements to be numbered arbitrarily by the user.

NONLINEAR EQUATION SOLVER

Several different nonlinear equation solvers are available for finding equilibrium within each step. All are iterative in nature. In the *full Newton method*, a new stiffness matrix is formed and inverted each equilibrium iteration. This is the most costly method, but can require fewer iterations to reach equilibrium. In the *modified Newton method*, several iterations are performed using the same stiffness matrix. After each iteration, the geometry is updated using $\Delta\mathbf{u}$ and a new $\mathbf{R}$ is computed. This approach reduces cost by avoiding some forming and factoring of the stiffness matrix $\mathbf{K}$, but usually requires more iterations to reach equilibrium.

The default nonlinear equation solver is the BFGS solver, which uses a *quasi-Newton method*. In this method, the inverted stiffness matrix $\mathbf{K}$ is used for several iterations, but is improved after each iteration using an inexpensive rank two update. If convergence is not reached after 10 iterations, or if *divergence* (increasing $\mathbf{R}$) is detected, then a new stiffness matrix is automatically formed and inverted. This hybrid method combines the efficiency of the modified Newton method with the reliability of the full Newton method. The number of iterations between stiffness matrix reformations is a user input, defaulting to 10. If a value of one is chosen, then the full Newton method is recovered.

```
   BEGIN time step     1
============================================================
               time =  1.00000E+00
  current step size =  1.00000E+00

Iteration:   1     *|du|/|u| =  2.5517753E+00     *Ei/E0 =  1.0000000E+00

DIVERGENCE (increasing residual norm) detected:
    |{Fe}-{Fi}| ( 7.5426269E+03) exceeds |{Fe}| ( 5.0000000E+00)
automatically REFORMING stiffness matrix...

Iteration:   2     *|du|/|u| =  6.0812935E-01     *Ei/E0 =  4.0526413E-01
Iteration:   4     *|du|/|u| =  1.0974191E-02     *Ei/E0 =  2.3907781E-04
Iteration:   5     *|du|/|u| =  1.0978787E-02     *Ei/E0 =  1.7910795E-04
Iteration:   6     *|du|/|u| =  4.2201181E-03     *Ei/E0 =  4.2557768E-05
Iteration:   7     *|du|/|u| =  4.1142219E-03     *Ei/E0 =  3.0658711E-05
Iteration:   8     *|du|/|u| =  1.9794757E-03     *Ei/E0 =  9.1215551E-06
Iteration:   9     *|du|/|u| =  1.7957653E-03     *Ei/E0 =  6.1669480E-06
Iteration:  10     *|du|/|u| =  1.2022830E-03     *Ei/E0 =  2.9031284E-06

ITERATION LIMIT reached, automatically REFORMING stiffness matrix...

Iteration:  11     *|du|/|u| =  5.4011414E-04     *Ei/E0 =  1.0553019E-06
```

The print-out above shows typical behavior of the default BFGS nonlinear equation solver. Two automatic stiffness reformations are performed, initially due to divergence, and later when the default limit of 10 iterations is exceeded. By default, the progress of the equilibrium search is not shown to the screen. This output can be activated either using the NLPRINT input parameter, or interactively toggled on and off by entering “<ctrl-c> nlprint”.

$$\mathbf{K}_{n+1}^{-1} = \left(\mathbf{I} + \mathbf{w}\mathbf{v}^T\right)\mathbf{K}_n^{-1}\left(\mathbf{I} + \mathbf{v}\mathbf{w}^T\right)$$

The BFGS update: A new stiffness matrix inverse is approximated by the old stiffness matrix inverse, and the outer product of two carefully chosen vectors.

ELEMENT FORMULATIONS FOR IMPLICIT ANALYSIS

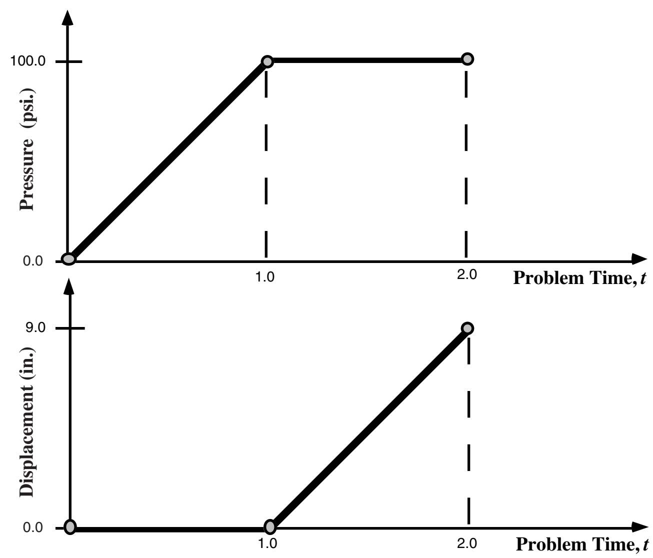
The default element formulations in LS-DYNA are highly efficient, using single point integration. For implicit analysis it is generally more effective to use more expensive element formulations which are less susceptable to hourglass instability. The Hughes-Liu brick element #2 and shell element #6, 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_DYNA3D 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. Try to get a picture of the deformed mesh during the iteration process. This can be done interactively by entering "<ctrl-c> iteration". LS-DYNA will create a binary plot database named "d3iter" containing a picture of the deformed mesh after each equilibrium iteration. View this database as you would a d3plot database. Frequently the problem will become obvious, especially as deformation is magnified.